



# ANSYS Mechanical APDL Command Reference

---



ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
ansysinfo@ansys.com  
<http://www.ansys.com>  
(T) 724-746-3304  
(F) 724-514-9494

Release 13.0  
November 2010

ANSYS, Inc. is  
certified to ISO  
9001:2008.

---

## Copyright and Trademark Information

© 2010 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

## Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

## U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

## Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

---

# Table of Contents

<b>1. About This Manual</b> .....	1
1.1. Conventions Used in this Manual .....	1
1.1.1. Product Codes .....	2
1.1.2. Applicable ANSYS Products .....	3
1.2. ANSYS Product Capabilities .....	3
1.3. Terminology .....	3
1.4. ANSYS Command Characteristics .....	5
1.4.1. Data Input .....	5
1.4.2. Free-Format Input .....	5
1.4.3. Nonrestrictive Data Input .....	5
1.4.4. Condensed Data Input .....	6
1.4.5. Units .....	6
1.4.6. Defaults .....	6
1.4.7. File Names .....	7
1.4.8. Star and Slash Commands .....	8
<b>2. Command Groupings</b> .....	9
2.1. SESSION Commands .....	9
2.2. DATABASE Commands .....	11
2.3. GRAPHICS Commands .....	14
2.4. APDL Commands .....	17
2.5. PREP7 Commands .....	20
2.6. SOLUTION Commands .....	40
2.7. POST1 Commands .....	54
2.8. POST26 Commands .....	63
2.9. AUX2 Commands .....	65
2.10. AUX3 Commands .....	66
2.11. AUX12 Commands .....	66
2.12. AUX15 Commands .....	67
2.13. OPTIMIZATION Commands .....	67
2.14. VARIATIONAL TECHNOLOGY Commands .....	69
2.15. PROBABILISTIC Design Commands .....	70
2.16. DISPLAY Program Commands .....	72
2.17. REDUCED Order Modeling Commands .....	72
2.18. Menu-Inaccessible Commands .....	73
<b>3. Command Dictionary</b> .....	79
I. Connection Commands .....	83
II. A Commands .....	91
III. B Commands .....	181
IV. C Commands .....	233
V. D Commands .....	349
VI. E Commands .....	427
VII. F Commands .....	591
VIII. G Commands .....	713
IX. H Commands .....	799
X. I Commands .....	829
XI. J Commands .....	859
XII. K Commands .....	861
XIII. L Commands .....	891
XIV. M Commands .....	965
XV. N Commands .....	1077

XVI. O Commands .....	1147
XVII. P Commands .....	1189
XVIII. Q Commands .....	1385
XIX. R Commands .....	1391
XX. S Commands .....	1461
XXI. T Commands .....	1659
XXII. U Commands .....	1765
XXIII. V Commands .....	1791
XXIV. W Commands .....	1887
XXV. X Commands .....	1901
XXVI. Y Commands .....	1905
XXVII. Z Commands .....	1907
Index .....	1909

## List of Tables

2.1. Run Controls .....	9
2.2. Processor Entry .....	10
2.3. Files .....	11
2.4. List Controls .....	11
2.5. Set Up .....	11
2.6. Selecting .....	12
2.7. Components .....	12
2.8. Working Plane .....	13
2.9. Coordinate System .....	13
2.10. Picking .....	14
2.11. Set Up .....	14
2.12. Views .....	15
2.13. Scaling .....	15
2.14. Style .....	15
2.15. Labeling .....	16
2.16. Graphs .....	16
2.17. Annotation .....	17
2.18. Parameter Definition .....	17
2.19. Macro Files .....	18
2.20. Abbreviations .....	18
2.21. Array Parameters .....	18
2.22. Process Controls .....	19
2.23. Matrix Operations (APDL Math) .....	20
2.24. Database .....	20
2.25. Element Type .....	21
2.26. Real Constants .....	21
2.27. Materials .....	21
2.28. Material Data Tables .....	22
2.29. Primitives .....	22
2.30. Keypoints .....	23
2.31. Hard Points .....	24
2.32. Lines .....	24
2.33. Areas .....	25
2.34. Volumes .....	26
2.35. Booleans .....	26



2.36. Meshing .....	27
2.37. Nodes .....	29
2.38. Elements .....	30
2.39. Superelements .....	31
2.40. Digitizing .....	31
2.41. Coupled Degrees of Freedom .....	31
2.42. Constraint Equations .....	32
2.43. Element Reordering .....	32
2.44. FLOTRAN Options .....	32
2.45. FLOTRAN Property .....	33
2.46. FLOTRAN Operating .....	33
2.47. FLOTRAN Solver .....	33
2.48. FLOTRAN Turbulence .....	34
2.49. FLOTRAN Stability .....	34
2.50. FLOTRAN Miscellaneous .....	34
2.51. FLOTRAN Multiple Species .....	35
2.52. Status .....	35
2.53. Explicit Dynamics .....	36
2.54. Sections .....	37
2.55. Ocean .....	38
2.56. Morphing .....	39
2.57. Trefftz Domain .....	39
2.58. Perfectly Matched Layers .....	39
2.59. Special Purpose .....	39
2.60. Analysis Options .....	40
2.61. Nonlinear Options .....	42
2.62. Dynamic Options .....	42
2.63. Spectrum Options .....	43
2.64. Load Step Options .....	44
2.65. Solid Constraints .....	45
2.66. Solid Forces .....	45
2.67. Solid Surface Loads .....	45
2.68. Solid Body Loads .....	46
2.69. Inertia .....	46
2.70. Miscellaneous Loads .....	47
2.71. Load Step Operations .....	47
2.72. Master Degrees of Freedom .....	48
2.73. Gap Conditions .....	48
2.74. Rezoning .....	48
2.75. Birth and Death .....	48
2.76. FE Constraints .....	48
2.77. FE Forces .....	49
2.78. FE Surface Loads .....	49
2.79. FE Body Loads .....	50
2.80. Status .....	50
2.81. Explicit Dynamics .....	51
2.82. FLOTRAN Checkout .....	52
2.83. Radiosity .....	52
2.84. ANSYS Multi-field solver Definition Commands .....	53
2.85. ANSYS Multi-field solver Global Controls .....	53
2.86. ANSYS Multi-field solver Time Controls .....	53
2.87. ANSYS Multi-field solver Load Transfer .....	54

2.88. ANSYS Multi-field solver Convergence Controls .....	54
2.89. ANSYS Multi-field solver Interface Mapping .....	54
2.90. Setup .....	54
2.91. Controls .....	55
2.92. Results .....	55
2.93. Element Table .....	56
2.94. Listing .....	56
2.95. Animation .....	57
2.96. Path Operations .....	57
2.97. Surface Operations .....	58
2.98. Load Case Calculations .....	58
2.99. Magnetics Calculations .....	59
2.100. Fatigue .....	59
2.101. Trace Points .....	60
2.102. FLOTRAN Processing .....	60
2.103. Special Purpose .....	60
2.104. Status .....	62
2.105. Failure Criteria .....	62
2.106. Set Up .....	63
2.107. Controls .....	63
2.108. Operations .....	64
2.109. Display .....	64
2.110. Listing .....	64
2.111. Special Purpose .....	65
2.112. Status .....	65
2.113. Binary Files .....	65
2.114. Results Files .....	66
2.115. General Radiation .....	66
2.116. Radiation Matrix Method .....	66
2.117. Radiosity Solver .....	66
2.118. IGES .....	67
2.119. Specifications .....	67
2.120. Operations .....	68
2.121. Files .....	68
2.122. Run .....	68
2.123. Display .....	68
2.124. DesignXplorer .....	69
2.125. Harmonic Sweep using VT Accelerator .....	70
2.126. Deterministic Model .....	70
2.127. Probabilistic Preprocessing .....	70
2.128. Probabilistic Methods .....	70
2.129. Run Probabilistic Analysis .....	71
2.130. Probabilistic Postprocessing .....	71
2.131. Probabilistic Database .....	71
2.132. Response Surface .....	71
2.133. Auxiliary Commands and Information .....	71
2.134. Setup .....	72
2.135. Driver Options .....	72
2.136. Action .....	72
2.137. Set Up .....	72
2.138. Preparation .....	72
2.139. Generation Pass .....	73

2.140. Use Pass ..... 73



---

## Chapter 1: About This Manual

---

Welcome to the *Command Reference*. This manual contains a complete dictionary of detailed ANSYS command descriptions, arranged in alphabetical order for ease of access. It is the definitive reference for correct command usage, providing comprehensive specifications for every argument of every ANSYS command.

The *Command Reference* is intended to give you information on individual ANSYS commands. Although this manual also contains lists of commands arranged by **functional grouping**, this manual is not intended to be your primary source of procedural information -- look in the appropriate analysis guides for introductory and procedural guidelines concerning when and where to use commands.

Command references given in the various analysis guides refer to the command descriptions given in *Command Dictionary* (p. 79) of this manual. These descriptions are organized in dictionary form to allow you to look up the commands conveniently. The alphabetical ordering of commands that begin with a star (\*) or a slash (/) ignores those symbols for ordering purposes.

Once you have reviewed a command in the *Command Reference*, you should also read the theoretical description of the command as given in the *Theory Reference for the Mechanical APDL and Mechanical Applications*, if indicated.

The following *Command Reference* topics are available:

- 1.1. Conventions Used in this Manual
- 1.2. ANSYS Product Capabilities
- 1.3. Terminology
- 1.4. ANSYS Command Characteristics

### 1.1. Conventions Used in this Manual

ANSYS documentation uses the following conventions to help you identify various types of information:

Type style or text	Indicates
<b>BOLD</b>	Uppercase, bold text indicates command names (such as <b>K</b> , <b>DDELE</b> ) or elements ( <b>LINK180</b> ).
<b>Bold&gt;Bold</b>	Bold text in mixed case indicates a GUI menu path, a series of menu choices for accessing a command from the GUI. One or more angle brackets (>) separate menu items in a menu path. Frequently in text, an ANSYS command is followed by its GUI equivalent in parentheses, as shown in this example: <b>*GET</b> command ( <b>Utility Menu&gt; Parameters&gt; Get Scalar Data</b> )
<i>ITALICS</i>	Uppercase italic letters indicate command arguments for <i>numeric</i> values (such as <i>VALUE, INC, TIME</i> ). On some commands, non-numeric convenience labels (for example, ALL and P) can also be entered for these arguments.

Type style or text	Indicates
<i>Italics</i>	Mixed case italic letters indicate command arguments for <i>alphanumeric</i> values (for example, <i>Lab</i> or <i>Fname</i> ). The documentation also uses italic text for emphasis.
TYPEWRITER	A typewriter font indicates command input listings and ANSYS output listings.

## Note

Any mention of a command or element name in this volume implies a reference to the appropriate command or element description (in the *Command Reference* or *Element Reference* manuals, respectively) for more detailed information.

Often it is necessary to refer to another command within a text description. Interpret the string [CMD] as "see the **CMD** command." For example, "... the element coordinate system **ESYS**] should be used to ...," means you can refer to the **ESYS** command for information related to the element coordinate system. Where several commands are referenced, the form [CMD1, CMD2, etc.] is used. In some cases only the primary command of a related group may be referenced.

### 1.1.1. Product Codes

Below and to the right of the short description of each command, you will see a list of product codes. These codes represent the products in the ANSYS Family of Products. The command is valid only for those products whose symbols are listed. A command that is valid in the entire set of products would have the following list of products:

MP ME ST PR PRN DS DSS FL EM EH DY PP VT EME MFS

The codes represent each of the products in the ANSYS suite of products:

Code	Product
MP	ANSYS Multiphysics
ME	ANSYS Mechanical
ST	ANSYS Structural
PR	ANSYS Professional - Nonlinear Thermal
PRN	ANSYS Professional - Nonlinear Structural
DS	ANSYS DesignSpace
DSS	ANSYS DesignSpace - Structural
FL	ANSYS FLOTRAN
EM	ANSYS Emag - Low Frequency
EH	ANSYS Emag - High Frequency
DY	ANSYS LS-DYNA
PP	ANSYS PrepPost
VT	ANSYS DesignXplorer
EME	ANSYS Mechanical/ANSYS Emag

Code	Product
MFS	ANSYS Mechanical/CFX-Flo

For a list of the features included in each product, see *ANSYS Product Capabilities* (p. 3).

If the symbol for a product does not appear, then that command is either not valid or not applicable in the corresponding product, and should not be issued when using that product. For example, if the PR and FL symbols are not listed, the pertinent command is not valid in the ANSYS Professional or ANSYS FLOTRAN products, but is valid in each of the remaining ANSYS products.

In most cases, commands and elements that are valid in ANSYS Multiphysics are also valid in the ANSYS ED and ANSYS Multiphysics 1, 2, and 3 products. Exceptions are noted under Product Restrictions for the applicable command or element. ANSYS ED and ANSYS Multiphysics 1, 2, and 3 are not listed as separate product codes.

### 1.1.2. Applicable ANSYS Products

This document applies to the following ANSYS products:

- ANSYS Multiphysics (includes all structural, thermal, electromagnetics, and computational fluid dynamics (CFD) capabilities, excludes explicit dynamics)
- ANSYS Mechanical (includes all structural and thermal capabilities; excludes electromagnetics, CFD, and explicit dynamics capabilities)
- ANSYS Structural (includes all structural linear and nonlinear capabilities)
- ANSYS Professional (Nonlinear Thermal and Nonlinear Structural)
- ANSYS Emag (Low Frequency and High Frequency)
- ANSYS FLOTRAN
- ANSYS LS-DYNA
- ANSYS PrepPost
- ANSYS ED
- ANSYS Mechanical/ANSYS Emag (combination of All Mechanical and Emag capabilities)
- ANSYS Mechanical/CFX-Flo (combination of All Mechanical and CFX-Flo capabilities)

Some command arguments and element key option (KEYOPT) settings have defaults in the derived products that are different from those in ANSYS Multiphysics. These cases are clearly documented under the Product Restrictions section of the affected commands and elements. If you intend to use your derived product input file in ANSYS Multiphysics, you should explicitly input these settings in the derived product, rather than letting them default; otherwise, behavior in ANSYS Multiphysics will be different.

While ANSYS connection capabilities and High Performance Computing are included as part of the ANSYS release distribution, they are separately-licensed products. Consult your ASD if you want to install and run any of the separately-licensed products at your site.

## 1.2. ANSYS Product Capabilities

A complete list of engineering capabilities available in the various ANSYS products can be found on our website at <http://www.ansys.com/assets/brochures/capabilities-brochure.pdf>.

## 1.3. Terminology

Various terms are used in the command descriptions throughout this manual. These terms are defined as follows:

**Analysis** - The set of input lines relating to a single problem. An analysis basically consists of three phases: the Preprocessing Phase, the Solution Phase, and the Postprocessing Phase.

**Command** - An instruction to supply data, or control, to the program. Commands usually begin with a prescribed name, followed by alphanumeric data. For example, the command **ET**, *ITYPE*, *Ename*, ... may be input as **ET,1,PIPE288** stating that element type 1 is defined as the **PIPE288** element. The uppercase argument name indicates that a numerical value is typically entered in that field, whereas an upper-lower case data label indicates that an alphanumeric value is typically entered in that field. Another command, for example, **/PREP7**, instructs the program to enter the PREP7 portion of the program. All valid commands are alphabetically listed in the *Command Dictionary* (p. 79) of this document. Unrecognized commands are further processed in a macro search (**\*USE**) before being ignored. Commands may be indented on a line for clarity.

**Data** - Data may be numeric (real or integer), alphabetic, or alphanumeric (containing letters and numbers). Nonnumeric data should not contain special characters such as

```
! @ # $ % ^ & * ( ) _ - +
= | \ { } [ ] " ' / < > ~ `
```

Numeric data may be input in a variety of ways as described in *Data Input* (p. 5). Some commands are switches of the form **Commandname,Key** where *Key* can be 0, NO or OFF to toggle the switch off; or 1, YES or ON to toggle the switch on.

**Degree of Freedom** - The degrees of freedom are the primary nodal unknowns determined by the analysis. They may be displacements, rotations, temperatures, pressures, voltages, etc. A degree of freedom is defined by a node number and a label, for example, 1 UX, 87 ROTZ, 4 TEMP, etc. Derived results, such as stresses, heat flows, etc. are computed from the degree of freedom results and are considered secondary unknowns.

Element types with unequal sets of degrees of freedom can be combined in a single structure. For example, a 2-D structure composed of 2-D solid elements having two degrees of freedom (UX, UY) at each node and a 2-D beam element having three degrees of freedom (UX, UY, ROTZ) at each node will have the latter three degrees of freedom at the common nodes. Nodes which do not have a beam element attached will have only two degrees of freedom with ROTZ eliminated from the solution printout.

**Field** - The command name and data items entered on a command are separated into consecutive fields. A field is assumed to be as "wide" as the number of characters specified. A comma is used to end one field and begin the next.

**Line** - A line of input is a physical record read by the computer. Input lines are limited to 640 characters (including preceding blanks, commas, and any special characters). For file input, a line is represented by one 640 column data record. For interactive input, a line is the string of characters (640 maximum) entered before the RETURN key is pressed. Several commands may be put on one line as described in *Data Input* (p. 5). Blank lines are permitted for clarity.

**Command name** - Only the first four characters of any alphabetic (or alphanumeric) command name are interpreted by the program (except as noted for certain commands, such as **/POSTn**, **/AUXn**, **\*ENDDO**, etc.). The remaining characters of the field are ignored. Names shown with fewer than four characters are assumed to have blanks up through the fourth character as part of the name. For example, this sample **ET** command may be input as **ET,1,42** or **ET ,1,42** or **ET ,1,42**, but not as **ETABCD,1,42**. Names may not contain special characters (as described in *Data Input* (p. 5)). If the command name is omitted, the name defaults to the name of the previous command, unless it was a slash (/) or star (\*) command.

**Postprocessing Phase** - The set of ANSYS commands causing further processing of the solution output. These commands consist of commands from the POST1 and POST26 processors. The postprocessing phase may consist of input for several postprocessing sessions (in series).



**Preprocessing Phase** - The set of ANSYS commands related to defining the model. The preprocessing phase consists of input from the PREP7 processor.

**Program** - The collection of all processors (preprocessing, postprocessing, auxiliary, etc.) is called the ANSYS program.

**Processor** - A group of related functions, such as model definition (PREP7) or results examination (POST1).

**Run** - The collection of all ANSYS commands between the system level commands is called an ANSYS run (or session). A session may also consist of several analyses in series (separated by a **/CLEAR** command).

**Solution Phase** - The set of ANSYS commands which apply boundary conditions to the model created in preprocessing, then performs a solution for that set of boundary conditions. The solution phase may consist of several solutions in series, such as a static solution, followed by a modal solution, etc.

## 1.4. ANSYS Command Characteristics

### 1.4.1. Data Input

The data input for each command is described in this manual. Data should not be input in any undocumented field, nor should other than documented values be input in any field. Also, the data input described in this manual should not be used with any earlier version of the ANSYS program. Some features that allow easy input of data are free-format, nonrestrictive, and condensed input.

### 1.4.2. Free-Format Input

Free-format capability allows the user to input data in consecutive fields without having to space to each field. The comma (,) character effectively ends the field so that the next character will be input in the beginning of the next field. A blank field is skipped by having no data item between the commas. Fields are assumed to be as wide as the number of characters specified. Input is converted to formatted fields when coded files are written (of width large enough (16 characters maximum) to minimize loss of accuracy).

Significant figures of output should not be expected to match that of input. Machine precision, rounding of numbers when writing internal scratch files, etc., tend to lower the precision during the analysis.

### 1.4.3. Nonrestrictive Data Input

Nonrestrictive data input allows the user to enter any form of data in a field and the ANSYS program will interpret it as required (integer, real, or alphabetic) or ignore it. Double precision is used throughout the program for all real numbers. Alphabetic data may be entered in upper or lower case. Lower case is internally converted to upper case within the program (except for case-sensitive applications, such as in comments, (text preceded by a !) titles, and file names.)

The following features are available with the nonrestrictive data input.

- No distinction is necessary between real and integer data.
- Data may be placed anywhere within the field.
- Real data input without a decimal point has the decimal point assumed after the right-most digit.
- Real number values input to integer data fields will be rounded to the nearest integer. The absolute value of integer data must fall between zero and 2,000,000,000.
- Element and node numbers (IDs) should be between 1 and 99,999,999. Element type, material reference, real constant, section, and coordinate system IDs should be between 1 and 9,999,999. IDs large than

these may not be written out correctly in listings (xLIST commands), **CDWRITE** files, **LSWRITE** files, and other text-based output files.

- Exponents may be input in the field after the number. The E (or D) character must be used (upper or lower case). The sign after the E character is assumed to be the sign associated with the exponent (the absence of a sign is assumed to be +). The absolute value of real data must either be zero, or between 1.0E-60 and 1.0E+60.
- A nonnumeric character in the numeric field (other than a valid convenience label, a parameter name, or an E or D exponent character) will be ignored and will cause the remainder of the field to be ignored. When the program can accept an alphanumeric label in a field, it will interpret any alphanumeric input that matches a valid label as the appropriate label. In other words, the program will not interpret a valid label as an identically named parameter. If you truly want to use a parameter in place of a valid label, you can use forced parametric substitution (using % signs). A non-alphanumeric character in a label field is taken as part of the label. Non-alphanumeric characters (such as CONTROL-characters, TAB-characters, and other terminal editing characters) should be avoided since they may be used directly instead of being interpreted by the terminal.

### 1.4.4. Condensed Data Input

An option for condensing the data input is available with the \$ character. This character effectively ends a command and allows the next command to begin on the same line (at the next column). The line must not extend beyond 640 characters (including all commas and \$ signs). If a command cannot be completed on the line, it should begin on the next line. Blank commands are ignored. The \$ character should not be used following any command that causes a file switch to read additional commands (such as the **/INPUT**, **\*USE**, **OPEXE**, **CDREAD**, etc. commands, any "unknown command" macro, or any do-loops or if-then-else constructs). The \$ character should not be used between a command and its required following format line (such as **\*VREAD**, **\*VWRITE**, **\*MSG**, etc.). The \$ character should not be used after the **ALLSEL** command. Using the \$ character in interactive mode may result in unexpected output behavior.

### 1.4.5. Units

The ANSYS program permits the use of any consistent set of units for length, force, time, temperature, etc. Care must be taken to ensure that all input data are in the same set of units. The **/UNITS** command may be used to note the system of units being used. Temperatures may be absolute or relative in most cases. For problems requiring absolute temperatures, such as those involving creep, swelling, or radiation, temperatures may be input as Celsius or Fahrenheit, for convenience, with a temperature shift from absolute zero [**TOFFST**].

### 1.4.6. Defaults

ANSYS commands are of two types:

- specification commands
- action commands

Specifications define how an action is to be performed. If a specification is not defined before the action, the default specification is used. If some specifications are changed after the action, these changed specifications (and the remaining previous specifications, if any) will be used for the next action. If the same specification is defined more than once before the action, the last specification is used. Note, since specification settings are easily forgotten, reset the specification to the desired value before the action command.

To minimize the data input requirements, ANSYS commands operate on a "default" principle. That is, unless otherwise stated, default specifications are used whenever needed. Two types of default are used:

- the command default
- the argument default

The command default is defined as the action taken, or the argument values used, if the entire command is omitted. For action commands, the command default is "no action" unless otherwise stated. A description of the command default, when applicable, appears at the end of the Argument section of the command description. The argument default is the value the argument takes on if the command is included, but the argument on the command is left blank. The latter case is often used when only some arguments on a command are to be specified or to restore a default specification after it has been changed. The argument default is documented as part of each argument's description.

A default specification is invoked whenever no value (or a blank field) is input for the argument. If the argument accepts a numeric value and no default is specified, a blank field defaults to a zero value. Where a default is specified, a blank *or a zero value* will produce the default value (unless otherwise specified). If a zero value is desired where a zero input produces a nonzero default, input a small number (such as 1E-10) instead of zero. Specifications are initialized to their default values. Defaults which are not obvious are described, for the most part, with each command. For cases where defaults are not obvious and are not described, enter the desired value. Such cases may occur where no default is favored or where a default is purposely not documented (so that it may change in the future).

## 1.4.7. File Names

Various files are used during an ANSYS session for reading, writing, and storing data. ANSYS-generated files are identified by a name, a system-dependent separator, and an extension (as in the form `fname.ext`).

The name is obtained from the `Jobname` (250 characters maximum, including the directory name) as defined with the ANSYS execution command.

---

### Note

Windows cannot create directory names longer than 245 characters.

Unless otherwise specified (see the [Operations Guide](#)), the `Jobname` defaults to a system dependent name (usually `FILE`). The extension is a 2 to 4-character identifier (see [Files that ANSYS Writes](#) in the [Basic Analysis Guide](#)). For example, the ANSYS error message file (which has the identifier `ERR`) with a `Jobname` of `FILE` and separator (`.`) would be generated as `FILE.ERR`. If the `Jobname` is `JOB1`, the file would be named `JOB1.ERR`. ANSYS documentation generically refers to ANSYS files as `Jobname.identifier`, such as `Jobname.ERR` for the error message file. Note, the extension may be shown upper case in the documentation but may actually be lower case on some operating systems (such as UNIX). Unless otherwise indicated, ANSYS files are written in the current working directory.

Some commands (such as `/INPUT` and `/OUTPUT`) allow you to specify a file name other than `Jobname`, and an extension other than the default extension. Such user-written files use a naming convention similar to the ANSYS-generated files except that the name (248 characters maximum, including the directory name) is user defined, and the extension (8 characters maximum) is optional. The file name can include the directory path, but it is not necessary if you want to use the default directory. File names containing directory paths must reference existing directories. File and directory names may contain blank spaces, but strings containing blank spaces must be enclosed in single quotes.

A system dependent separator is automatically written between file name and extension (if the extension exists) input on commands.

File name fields are usually restricted to 248 characters on ANSYS commands, including the directory name. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

To help ensure portability of input files across systems, the ANSYS program has certain requirements for file names. File names specified by the user should be valid both per ANSYS criteria (ASCII alpha-numeric), and per the operating system. Avoid using special characters (+, -, \*, /, \$, !, etc.) in file names unless meaningful, and begin file names with a letter. Some system-dependent special characters (such as ~) are not interpreted the same as they are at the operating system level when used in pathnames. Parameter substitution may be forced within the name, extension, or directory fields by enclosing the parameter within percent (%) signs. For example, if I=5, the name TEST%I% becomes TEST5. Only one forced substitution may be made per field.

### 1.4.8. Star and Slash Commands

The slash (/) and star (\*) commands are usually used for supplying general control instructions to the ANSYS program. The slash commands are used, for example, for entering a processor (**/PREP7**, **/POST1**, etc.). Printout controls, display controls, and file controls are available within the slash command set. Repeat, looping and macro controls are available within the star command set. Many star and slash commands are global and apply to all processors. For example, the **/TITLE** command may be used to change the title in any processor (PREP7, POST1, etc.). The star commands can be input anywhere. Most slash commands can be input anywhere, however, some can only be input at a Begin level. The latter commands are specifically noted in the command explanation section. Graphics displays are controlled with the graphics display slash commands. The graphics display slash commands can be input anywhere.

---

## Chapter 2: Command Groupings

---

Following are tables of related commands. The documentation for each command also references these tables, providing convenient access to related commands.

**Example:** To define material types in the PREP7 general preprocessor, see *PREP7 Commands* (p. 20) and examine the commands available under *Table 2.27: Materials* (p. 21).

While this document is useful as a reference, you should first look at the appropriate analysis guide to learn which commands are available for a particular type of analysis or operation. The analysis guides contain more detailed information about which commands to use for each step of a given analysis type.

Command Grouping Table Title
<i>SESSION Commands</i>
<i>DATABASE Commands</i>
<i>GRAPHICS Commands</i>
<i>APDL Commands</i>
<i>PREP7 Commands</i>
<i>SOLUTION Commands</i>
<i>POST1 Commands</i>
<i>POST26 Commands</i>
<i>AUX2 Commands</i>
<i>AUX3 Commands</i>
<i>AUX12 Commands</i>
<i>AUX15 Commands</i>
<i>OPTIMIZATION Commands</i>
<i>VARIATIONAL TECHNOLOGY Commands</i>
<i>PROBABILISTIC Design Commands</i>
<i>DISPLAY Program Commands</i>
<i>REDUCED Order Modeling Commands</i>
<i>Menu-Inaccessible Commands</i>

### 2.1. SESSION Commands

These commands provide general control to the ANSYS session. The commands are grouped by functionality.

#### Table 2.1 Run Controls

**These SESSION commands control the overall characteristics of the ANSYS session, including the jobname, Graphical User Interface behavior, and file switching.**

**/BATCH**                Sets the program mode to "batch."

**These SESSION commands control the overall characteristics of the ANSYS session, including the jobname, Graphical User Interface behavior, and file switching.**

<b>/CONFIG</b>	Assigns values to ANSYS configuration parameters.
<b>/CWD</b>	Changes the current working directory.
<b>/EOF</b>	Exits the file being read.
<b>/EXIT</b>	Stops the run and returns control to the system.
<b>/FILENAME</b>	Changes the jobname for the analysis.
<b>HELP</b>	Displays help information on ANSYS commands and element types.
<b>/INPUT</b>	Switches the input file for the commands that follow.
<b>KEYW</b>	Sets a keyword used by the GUI for context filtering (GUI).
<b>MEMM</b>	Allows the current session to keep allocated memory
<b>/MENU</b>	Activates the Graphical User Interface (GUI).
<b>/MSTART</b>	Controls the initial GUI components.
<b>/NERR</b>	Limits the number of warning and error messages displayed.
<b>/OUTPUT</b>	Redirects text output to a file or to the screen.
<b>PAUSE</b>	Temporarily releases (pauses) the currently used product license so that another application can use it.
<b>/STATUS</b>	Lists the status of items for the run.
<b>/SYP</b>	Passes a command string and arguments to the operating system.
<b>/SYS</b>	Passes a command string to the operating system.
<b>/UI</b>	Activates specified GUI dialog boxes.
<b>/UIS</b>	Controls the GUI behavior.
<b>UNPAUSE</b>	Restores use of a temporarily released (paused) product license. (Applicable only <i>after</i> a previously issued <b>PAUSE</b> command.)

## Table 2.2 Processor Entry

**These SESSION commands are used to enter and exit the various processors in the program.**

<b>/AUX2</b>	Enters the binary file dumping processor.
<b>/AUX3</b>	Enters the results file editing processor.
<b>/AUX12</b>	Enters the radiation matrix generation processor.
<b>/AUX15</b>	Enters the IGES file transfer processor.
<b>FINISH</b>	Exits normally from a processor.
<b>/OPT</b>	Enters the design optimizer.
<b>/POST1</b>	Enters the database results postprocessor.
<b>/POST26</b>	Enters the time-history results postprocessor.
<b>/PREP7</b>	Enters the model creation preprocessor.
<b>/QUIT</b>	Exits a processor.

These **SESSION** commands are used to enter and exit the various processors in the program.

**/SOLU** Enters the solution processor.

### Table 2.3 Files

These **SESSION** commands are for file operations, such as deleting, copying, and listing.

**ANSTOAQWA** Creates an AQWA-LINE input file from the current ANSYS model.

**ANSTOASAS** Creates an ASAS input file from the current ANSYS model.

**/ASSIGN** Reassigns a file name to an ANSYS file identifier.

**/CLOG** Copies the session log file to a named file.

**/COPY** Copies a file.

**/DELETE** Deletes a file.

**/FDELE** Deletes a binary file after it is used.

**LGWRITE** Writes the database command log to a file.

**\*LIST** Displays the contents of an external, coded file.

**/RENAME** Renames a file.

### Table 2.4 List Controls

These **SESSION** commands are used to control listings and printed program output.

**C\*\*\*** Places a comment in the output.

**/COM** Places a comment in the output.

**/GO** Reactivates suppressed printout.

**/GOLIST** Reactivates the suppressed data input listing.

**/GOPR** Reactivates suppressed printout.

**/NOLIST** Suppresses the data input listing.

**/NOPR** Suppresses the expanded interpreted input data listing.

## 2.2. DATABASE Commands

These commands are used to operate on the database in a global sense. The commands are grouped by functionality.

### Table 2.5 Set Up

These **DATABASE** commands can be used to initialize the database, save it to a file, or annotate it with titles and systems of units.

**/CLEAR** Clears the database.

**RESUME** Resumes the database from the database file.

**SAVE** Saves all current database information.

**/SMBC** Controls the display of solid model boundary condition symbols and labels.

**STAT** Displays the status of database settings.

**/STITLE** Defines subtitles.

**These DATABASE commands can be used to initialize the database, save it to a file, or annotate it with titles and systems of units.**

<b>/TITLE</b>	Defines a main title.
<b>UNDO</b>	Allows the user to modify or save commands issued since the last <b>RESUME</b> or <b>SAVE</b> command.
<b>/UNITS</b>	Annotates the database with the system of units used.

### Table 2.6 Selecting

**These DATABASE commands are used to select subsets of database entities for further operations.**

<b>ALLSEL</b>	Selects all entities with a single command.
<b>ASLL</b>	Selects those areas containing the selected lines.
<b>ASEL</b>	Selects a subset of areas.
<b>ASLV</b>	Selects those areas contained in the selected volumes.
<b>DOFSEL</b>	Selects a degree-of-freedom label set for reference by other commands.
<b>ESEL</b>	Selects a subset of elements.
<b>ESLA</b>	Selects those elements associated with the selected areas.
<b>ESLL</b>	Selects those elements associated with the selected lines.
<b>ESLN</b>	Selects those elements attached to the selected nodes.
<b>ESLV</b>	Selects elements associated with the selected volumes.
<b>KSEL</b>	Selects a subset of keypoints or hard points.
<b>KSLL</b>	Selects those keypoints contained in the selected lines.
<b>KSLN</b>	Selects those keypoints associated with the selected nodes.
<b>LSEL</b>	Selects a subset of lines.
<b>LSLA</b>	Selects those lines contained in the selected areas.
<b>LSLK</b>	Selects those lines containing the selected keypoints.
<b>NSEL</b>	Selects a subset of nodes.
<b>NSLA</b>	Selects those nodes associated with the selected areas.
<b>NSLE</b>	Selects those nodes attached to the selected elements.
<b>NSLK</b>	Selects those nodes associated with the selected keypoints.
<b>NSLL</b>	Selects those nodes associated with the selected lines.
<b>NSLV</b>	Selects those nodes associated with the selected volumes.
<b>PARTSEL</b>	Selects a subset of parts in an explicit dynamic analysis.
<b>VSEL</b>	Selects a subset of volumes.
<b>VSLA</b>	Selects those volumes containing the selected areas.

### Table 2.7 Components

**These DATABASE commands allow selected subsets of entities to be named as components for easy selection later on.**

<b>CM</b>	Groups geometry items into a component.
-----------	---



**These DATABASE commands allow selected subsets of entities to be named as components for easy selection later on.**

<b>CMDELE</b>	Deletes a component or assembly definition.
<b>CMEDIT</b>	Edits an existing component or assembly.
<b>CMGRP</b>	Groups components and assemblies into an assembly.
<b>CMLIST</b>	Lists the entities contained in a component or assembly.
<b>CMMOD</b>	Modifies the specification of a component.
<b>CMPLLOT</b>	Plots the entities contained in a component or assembly.
<b>CMSEL</b>	Selects a subset of components and assemblies.
<b>CMWRITE</b>	Writes components and assemblies to a file.

**Table 2.8 Working Plane**

**These DATABASE commands turn on, move, rotate, and modify the working plane, which is used for picking operations.**

<b>KWPAVE</b>	Moves the working plane origin to the average location of keypoints.
<b>KWPLAN</b>	Defines the working plane using three keypoints.
<b>LWPLAN</b>	Defines the working plane normal to a location on a line.
<b>NWPAVE</b>	Moves the working plane origin to the average location of nodes.
<b>NWPLAN</b>	Defines the working plane using three nodes.
<b>WPAVE</b>	Moves the working plane origin to the average of specified points.
<b>WPCSYS</b>	Defines the working plane location based on a coordinate system.
<b>WPLANE</b>	Defines a working plane to assist in picking operations.
<b>WPOFFS</b>	Offsets the working plane.
<b>WPROTA</b>	Rotates the working plane.
<b>WPSTYL</b>	Controls the display and style of the working plane.

**Table 2.9 Coordinate System**

**These DATABASE commands define and manipulate coordinate systems.**

<b>CLOCAL</b>	Defines a local coordinate system relative to the active coordinate system.
<b>CS</b>	Defines a local coordinate system by three node locations.
<b>CSCIR</b>	Locates the singularity for non-Cartesian local coordinate systems.
<b>CSDELE</b>	Deletes local coordinate systems.
<b>CSKP</b>	Defines a local coordinate system by three keypoint locations.
<b>CSLIST</b>	Lists coordinate systems.
<b>CSWPLA</b>	Defines a local coordinate system at the origin of the working plane.
<b>CSYS</b>	Activates a previously defined coordinate system.

**These DATABASE commands define and manipulate coordinate systems.**

**LOCAL** Defines a local coordinate system by a location and orientation.

### Table 2.10 Picking

**These DATABASE commands are generated by the GUI when picking operations are performed.**

**FITEM** Identifies items chosen by a picking operation (GUI).

**FLST** Specifies data required for a picking operation (GUI).

## 2.3. GRAPHICS Commands

These commands are used to control the graphics of the ANSYS program. The commands are grouped by functionality.

### Table 2.11 Set Up

**These GRAPHICS commands are used for general graphics control, such as window set up, color settings, specifications saving, etc.**

**/CMAP** Changes an existing or creates a new color mapping table.

**/COLOR** Specifies the color mapping for various items.

**/DEVICE** Controls graphics device options.

**DSYS** Activates a display coordinate system for geometry listings and plots.

**/DV3D** Sets 3-D device option modes.

**/ERASE** Specifies that the screen is to be erased before each display.

**ERASE** Explicitly erases the current display.

**/GCMD** Controls the type of element or graph display used for the **GPLOT** command.

**/GCOLUMN** Allows the user to apply a label to a specified curve.

**/GFILE** Specifies the pixel resolution on Z-buffered graphics files.

**GPLOT** Controls general plotting.

**/GRAPHICS** Defines type of graphics display to be used for element plots.

**/GRESUME** Sets graphics settings to the settings on a file.

**/GSAVE** Saves graphics settings to a file for later use.

**/GTYPE** Controls the entities that the **GPLOT** command displays.

**HPGL** Specifies various HP options.

**/IMAGE** Allows graphics data to be captured and saved.

**IMMED** Allows immediate display of a model as it is generated.

**JPEG** Provides JPEG File Export for ANSYS Displays.

**/MREP** Enables you to reissue the graphics command macro "name" during a replot or zoom operation.

**/NOERASE** Prevents the screen erase between displays.

**/PCOPY** Generates hard copies automatically (device dependent).

**These GRAPHICS commands are used for general graphics control, such as window set up, color settings, specifications saving, etc.**

<b>PNGR</b>	Sets file options for PNGR graphics export for ANSYS displays.
<b>PSCR</b>	Specifies various PostScript options.
<b>/PSTATUS</b>	Displays the global or window display specifications.
<b>/REPLOT</b>	Automatically reissues the last display command for convenience.
<b>/RESET</b>	Resets display specifications to their initial defaults.
<b>/SEG</b>	Allows graphics data to be stored in the local terminal memory.
<b>/SHOW</b>	Specifies the device and other parameters for graphics displays.
<b>TIFF</b>	Provides TIFF file Export for ANSYS Displays.
<b>/WINDOW</b>	Defines the window size on the screen.

**Table 2.12 Views**

**These GRAPHICS commands are used to control the view of the model.**

<b>/ANGLE</b>	Rotates the display about an axis.
<b>/AUTO</b>	Resets the focus and distance specifications to "automatically calculated."
<b>/DIST</b>	Specifies the viewing distance for magnifications and perspective.
<b>/FOCUS</b>	Specifies the focus point (center of the window).
<b>/USER</b>	Conveniently resets <b>/FOCUS</b> and <b>/DIST</b> to USER.
<b>/VCONE</b>	Defines the view cone angle for perspective displays.
<b>/VIEW</b>	Defines the viewing direction for the display.
<b>/VUP</b>	Specifies the global Cartesian coordinate system reference orientation.
<b>/XFRM</b>	Controls the centroid or the axis of dynamic rotation.
<b>/ZOOM</b>	Zooms a region of a display window.

**Table 2.13 Scaling**

**These GRAPHICS commands are used to scale various parts of the display.**

<b>/DSCALE</b>	Sets the displacement multiplier for displacement displays.
<b>/ICLWID</b>	Scales the line width of circuit builder icons.
<b>/ICSCALE</b>	Scales the icon size for elements supported in the circuit builder.
<b>/RATIO</b>	Distorts the object geometry.
<b>/SHRINK</b>	Shrinks elements, lines, areas, and volumes for display clarity.
<b>/SSCALE</b>	Sets the contour multiplier for topographic displays.
<b>/TXTRE</b>	Applies textured appearance to selected items.
<b>/VSCALE</b>	Scales the length of displayed vectors.

**Table 2.14 Style**

**These GRAPHICS commands are used to change the way a model is displayed.**

<b>/CPLANE</b>	Specifies the cutting plane for section and capped displays.
----------------	--

**These GRAPHICS commands are used to change the way a model is displayed.**

<b>/CTYPE</b>	Specifies the type of contour display.
<b>/EDGE</b>	Displays only the "edges" of an object.
<b>/ESHAPE</b>	Displays elements with shapes determined from the real constants.
<b>/FACET</b>	Specifies the facet representation used to form solid model displays.
<b>/GLINE</b>	Specifies the element outline style.
<b>/GMARKER</b>	Specifies the curve marking style.
<b>GMFACE</b>	Specifies the facet representation used to form solid models.
<b>/LIGHT</b>	Specifies the light direction for the display window.
<b>/NORMAL</b>	Allows displaying area elements by top or bottom faces.
<b>/SHADE</b>	Defines the type of surface shading used with Z-buffering.
<b>/TRLCY</b>	Specifies the level of translucency.
<b>/TYPE</b>	Defines the type of display.

**Table 2.15 Labeling****These GRAPHICS commands are used to add helpful labels and symbols to displays.**

<b>/CFORMAT</b>	Controls the graphical display of alpha character strings for parameters, components, assemblies, and tables.
<b>/CLABEL</b>	Specifies contour labeling.
<b>/CONTOUR</b>	Specifies the uniform contour values on stress displays.
<b>/CVAL</b>	Specifies nonuniform contour values on stress displays.
<b>/GFORMAT</b>	Specifies the format for the graphical display of numbers.
<b>/HBC</b>	Determines how boundary conditions are displayed in a display window.
<b>/NUMBER</b>	Specifies whether numbers, colors, or both are used for displays.
<b>/PBC</b>	Shows boundary condition symbols and values on displays.
<b>/PBF</b>	Shows body force loads as contours on displays.
<b>PGSELE</b>	Select a subset of elements for display with the PGR viewer.
<b>/PICE</b>	Shows initial conditions on elements as contours on displays.
<b>/PLOPTS</b>	Controls graphics options on subsequent displays.
<b>/PNUM</b>	Controls entity numbering/coloring on plots.
<b>/PSF</b>	Shows surface load symbols on model displays.
<b>/PSYMB</b>	Shows various symbols on displays.
<b>/TRIAD</b>	Shows the global XYZ coordinate triad on displays.
<b>/UDOC</b>	Determines position and content for the multi-legend options.

**Table 2.16 Graphs****These GRAPHICS commands are used to control the way line graphs are displayed.**

<b>/AXLAB</b>	Labels the X and Y axes on graph displays.
<b>/GRID</b>	Selects the type of grid on graph displays.
<b>/GROPT</b>	Sets various line graph display options.

**These GRAPHICS commands are used to control the way line graphs are displayed.**

<b>/GRTYP</b>	Selects single or multiple Y-axes graph displays.
<b>/GTHK</b>	Sets line thicknesses for graph lines.
<b>/XRANGE</b>	Specifies a linear abscissa (X) scale range.
<b>/YRANGE</b>	Specifies a linear ordinate (Y) scale range.

**Table 2.17 Annotation**

**These GRAPHICS commands are used to annotate a display with notes and symbols.**

<b>/AN3D</b>	Specifies 3-D annotation functions
<b>/ANNOT</b>	Activates graphics for annotating displays (GUI).
<b>/ANUM</b>	Specifies the annotation number, type, and hot spot (GUI).
<b>/LARC</b>	Creates annotation arcs (GUI).
<b>/LINE</b>	Creates annotation lines (GUI).
<b>/LSPEC</b>	Specifies annotation line attributes (GUI).
<b>/LSYMBOL</b>	Creates annotation symbols (GUI).
<b>/PCIRCLE</b>	Creates an annotation circle (GUI).
<b>/PMORE</b>	Creates an annotation polygon (GUI).
<b>/POLYGON</b>	Creates annotation polygons (GUI).
<b>/PSPEC</b>	Creates annotation polygon attributes (GUI).
<b>/PWEDGE</b>	Creates an annotation wedge (GUI).
<b>/TLABEL</b>	Creates annotation text (GUI).
<b>/TSPEC</b>	Creates annotation text attributes (GUI).

## 2.4. APDL Commands

These commands make up the ANSYS Parametric Design Language (APDL). The commands are grouped by functionality.

**Table 2.18 Parameter Definition**

**These APDL commands are used to define parameters and their values.**

<b>*AFUN</b>	Specifies units for angular functions in parameter expressions.
<b>*ASK</b>	Prompts the user to input a parameter value.
<b>*DEL</b>	Deletes a parameter (GUI).
<b>*DIM</b>	Defines an array parameter and its dimensions.
<b>*GET</b>	Retrieves a value and stores it as a user-named parameter.
<b>/INQUIRE</b>	Returns system information to a parameter.
<b>PARRES</b>	Reads parameters from a file.
<b>PARSAV</b>	Writes parameters to a file.
<b>*SET</b>	Assigns values to user-named parameters.
<b>*STATUS</b>	Lists the current parameters and abbreviations.
<b>*TAXIS</b>	Defines table index numbers.

**These APDL commands are used to define parameters and their values.**

<b>*TREAD</b>	Reads data from an external file into a table array parameter.
<b>*VFILL</b>	Fills an array parameter.
<b>*VGET</b>	Retrieves values into an array parameter.
<b>*VREAD</b>	Reads data and produces an array parameter vector or matrix.

**Table 2.19 Macro Files****These APDL commands are used to build and execute command macros.**

<b>*CFCLOS</b>	Closes the "command" file.
<b>*CFOPEN</b>	Opens a "command" file.
<b>*CFWRITE</b>	Writes an ANSYS command (or similar string) to a "command" file.
<b>*CREATE</b>	Opens (creates) a macro file.
<b>/DFLAB</b>	Changes degree of freedom labels for user custom elements.
<b>*END</b>	Closes a macro file.
<b>/MAIL</b>	Mails file to specified address.
<b>/MKDIR</b>	Creates a directory.
<b>*MSG</b>	Writes an output message via the ANSYS message subroutine.
<b>/PMACRO</b>	Specifies that macro contents be written to the session log file.
<b>/PSEARCH</b>	Specifies a directory to be searched for "unknown command" macro files.
<b>/RMDIR</b>	Removes (deletes) a directory.
<b>/TEE</b>	Writes a list of commands to a specified file at the same time that the commands are being executed.
<b>*ULIB</b>	Identifies a macro library file.
<b>*USE</b>	Executes a macro file.

**Table 2.20 Abbreviations****These APDL commands can be used to define abbreviations for longer commands, and to create user-defined commands.**

<b>*ABBR</b>	Defines an abbreviation.
<b>ABBRES</b>	Reads abbreviations from a coded file.
<b>ABBSAV</b>	Writes the current abbreviation set to a coded file.
<b>/UCMD</b>	Assigns a user-defined command name.

**Table 2.21 Array Parameters****These APDL commands are used to operate on parameter arrays (i.e., vectors and matrices).**

<b>/DIRECTORY</b>	Put the file names in the current directory into a string parameter array.
<b>*MFOURI</b>	Calculates the coefficients for, or evaluates, a Fourier series.
<b>*MFUN</b>	Copies or transposes an array parameter matrix.

**These APDL commands are used to operate on parameter arrays (i.e., vectors and matrices).**

<b>*MOPER</b>	Performs matrix operations on array parameter matrices.
<b>*MWRITE</b>	Writes a matrix to a file in a formatted sequence.
<b>*SREAD</b>	Reads a file into a string array parameter.
<b>*TOPER</b>	Operates on table parameters.
<b>*VABS</b>	Applies the absolute value function to array parameters.
<b>*VCOL</b>	Specifies the number of columns in matrix operations.
<b>*VCUM</b>	Allows array parameter results to add to existing results.
<b>*VEDIT</b>	Allows numerical array parameters to be graphically edited.
<b>*VFACT</b>	Applies a scale factor to array parameters.
<b>*VFUN</b>	Performs a function on a single array parameter.
<b>*VITRP</b>	Forms an array parameter by interpolation of a table.
<b>*VLEN</b>	Specifies the number of rows to be used in array parameter operations.
<b>*VMASK</b>	Specifies an array parameter as a masking vector.
<b>*VOPER</b>	Operates on two array parameters.
<b>*VPLOT</b>	Graphs columns (vectors) of array parameters.
<b>*VPUT</b>	Restores array parameter values into the ANSYS database.
<b>*VSCFUN</b>	Determines properties of an array parameter.
<b>*VSTAT</b>	Lists the current specifications for the array parameters.
<b>*VWRITE</b>	Writes data to a file in a formatted sequence.

**Table 2.22 Process Controls**

**These APDL commands can be used to control the order in which other commands are processed.**

<b>*CYCLE</b>	Bypasses commands within a do-loop.
<b>*DO</b>	Defines the beginning of a do-loop.
<b>*DOWHILE</b>	Loops repeatedly through the next <b>*ENDDO</b> command.
<b>*ELSE</b>	Separates the final if-then-else block.
<b>*ELSEIF</b>	Separates an intermediate if-then-else block.
<b>*ENDDO</b>	Ends a do-loop and starts the looping action.
<b>*ENDIF</b>	Ends an if-then-else.
<b>*EXIT</b>	Exits a do-loop.
<b>*GO</b>	Causes a specified line on the input file to be read next.
<b>*IF</b>	Conditionally causes commands to be read.
<b>*REPEAT</b>	Repeats the previous command.
<b>*RETURN</b>	Returns input stream to a higher level.

**These APDL commands can be used to control the order in which other commands are processed.**

**/WAIT** Introduces a delay before reading the next command.

**Table 2.23 Matrix Operations (APDL Math)**

**These APDL Math commands can be used to create, manipulate, and solve matrices.**

<b>*AXPY</b>	Performs the matrix operation $M2 = v * M1 + w * M2$ .
<b>*COMP</b>	Compresses the columns of a matrix using a specified algorithm.
<b>*DMAT</b>	Creates a dense matrix.
<b>*EIGEN</b>	Performs a modal solution with unsymmetric or damping matrices.
<b>*EXPORT</b>	Exports a matrix to a file in the specified format.
<b>*FREE</b>	Deletes a matrix or a solver object and frees its memory allocation.
<b>*ITENGINE</b>	Performs a solution using an iterative solver.
<b>*LSBAC</b>	Performs the solve (forward/backward substitution) of a factorized linear system.
<b>*LSENGINE</b>	Creates a linear solver engine.
<b>*LSFACTOR</b>	Performs the numerical factorization of a linear solver system.
<b>*MULT</b>	Performs the matrix multiplication $M3 = M1(T1) * M2(T2)$ .
<b>*NRM</b>	Computes the norm of the specified matrix or vector.
<b>*PRINT</b>	Prints the matrix values to a file.
<b>*SMAT</b>	Creates a sparse matrix.
<b>*VEC</b>	Creates a vector.

## 2.5. PREP7 Commands

These commands are used to create and set up the model. The commands are grouped by functionality.

**Table 2.24 Database**

**These PREP7 commands are used to read model data into the database, list out the database, and control the numbering of entities in the database.**

<b>AFLIST</b>	Lists the current data in the database.
<b>CDREAD</b>	Reads a file of solid model and database information into the database.
<b>CDWRITE</b>	Writes geometry and load database items to a file.
<b>CDOPT</b>	Specifies format to be used for archiving geometry.
<b>CECHECK</b>	Check constraint equations and couplings for rigid body motions.
<b>CHECK</b>	Checks current database items for completeness.
<b>CNCHECK</b>	Provides and/or adjusts the initial status of contact pairs.
<b>FC</b>	Provides failure criteria information and activates a data table to input temperature-dependent stress and strain limits.
<b>FCHECK</b>	Checks both the strain and stress input criteria for all materials.



**These PREP7 commands are used to read model data into the database, list out the database, and control the numbering of entities in the database.**

<b>FCDELE</b>	Deletes previously defined failure criterion data for the given material.
<b>FCLIST</b>	To list what the failure criteria is that you have input.
<b>IGESOUT</b>	Writes solid model data to a file in IGES Version 5.1 format.
<b>MFIMPORT</b>	Imports a new field into a current ANSYS Multi-field solver analysis.
<b>NOOFFSET</b>	Prevents the <b>CDREAD</b> command from offsetting specified data items.
<b>NUMCMP</b>	Compresses the numbering of defined items.
<b>NUMMRG</b>	Merges coincident or equivalently defined items.
<b>NUMOFF</b>	Adds a number offset to defined items.
<b>NUMSTR</b>	Establishes starting numbers for automatically numbered items.
<b>/PREP7</b>	Enters the model creation preprocessor.

### Table 2.25 Element Type

**These PREP7 commands define the type of elements to be used in the model.**

<b>DOF</b>	Adds degrees of freedom to the current degree-of-freedom set.
<b>ELBOW</b>	Specifies degrees of freedom to be coupled for end release and applies section constraints to elbow elements.
<b>ET</b>	Defines a local element type from the element library.
<b>ETCHG</b>	Changes element types to their corresponding types.
<b>ETCONTROL</b>	Controls the element technologies used in element formulation (for applicable elements).
<b>ETDELE</b>	Deletes element types.
<b>ETLIST</b>	Lists currently defined element types.
<b>KEYOPT</b>	Sets element key options.
<b>NSVR</b>	Defines the number of variables for user-programmable element options.

### Table 2.26 Real Constants

**These PREP7 commands define the model real constants.**

<b>R</b>	Defines the element real constants.
<b>RDELE</b>	Deletes real constant sets.
<b>RLIST</b>	Lists the real constant sets.
<b>RMODIF</b>	Modifies real constant sets.
<b>RMORE</b>	Adds real constants to a set.
<b>SETFGAP</b>	Updates real constant table for squeeze film elements.

### Table 2.27 Materials

**These PREP7 commands are used to define the linear material properties.**

<b>EMUNIT</b>	Specifies the system of units for magnetic field problems.
---------------	--

**These PREP7 commands are used to define the linear material properties.**

<b>MP</b>	Defines a linear material property.
<b>MPAMOD</b>	Modifies temperature-dependent coefficients of thermal expansion.
<b>MPCHG</b>	Changes the material number attribute of an element.
<b>MPCOPY</b>	Copies material model data from one material reference number to another.
<b>MPDATA</b>	Defines property data to be associated with the temperature table.
<b>MPDELE</b>	Deletes linear material properties.
<b>MPDRES</b>	Reassembles existing material data with the temperature table.
<b>/MPLIB</b>	Sets the default material library read and write paths.
<b>MPLIST</b>	Lists linear material properties.
<b>MPPLOT</b>	Plots linear material properties as a function of temperature.
<b>MPREAD</b>	Reads a file containing material properties.
<b>MPTEMP</b>	Defines a temperature table for material properties.
<b>MPTGEN</b>	Adds temperatures to the temperature table by generation.
<b>MPTRES</b>	Restores a temperature table previously defined.
<b>MPWRITE</b>	Writes linear material properties in the database to a file (if the Lib option is not specified) or writes both linear and nonlinear material properties (if Lib is specified) from the database to a file.
<b>TBFT</b>	Performs curve-fitting operations on materials.
<b>UIMP</b>	Defines constant material properties (GUI).

**Table 2.28 Material Data Tables****These PREP7 commands create and modify the data tables, used to define nonlinear materials, for example.**

<b>TB</b>	Activates a material data table.
<b>TBCOPY</b>	Copies a data table from one material to another.
<b>TBDATA</b>	Defines data for the material data table.
<b>TBDELE</b>	Deletes previously defined material data tables.
<b>TBEO</b>	Sets special options or parameters for material data tables.
<b>TBFIELD</b>	Defines values of field variables for the material data tables.
<b>TBLIST</b>	Lists the material data tables.
<b>TBMODIF</b>	Modifies data for the material data table (GUI).
<b>TBPLOT</b>	Displays the material data table.
<b>TBPT</b>	Defines a point on a stress-strain or B-H curve.
<b>TBTEMP</b>	Defines a temperature for the material data table.

**Table 2.29 Primitives****These PREP7 commands are used to create primitive shapes for modeling.**

<b>BLC4</b>	Creates a rectangular area or block volume by corner points.
-------------	--

**These PREP7 commands are used to create primitive shapes for modeling.**

<b>BLC5</b>	Creates a rectangular area or block volume by center and corner points.
<b>BLOCK</b>	Creates a block volume based on working plane coordinates.
<b>CON4</b>	Creates a conical volume anywhere on the working plane.
<b>CONE</b>	Creates a conical volume centered about the working plane origin.
<b>CYL4</b>	Creates a circular area or cylindrical volume anywhere on the working plane.
<b>CYL5</b>	Creates a circular area or cylindrical volume by end points.
<b>CYLIND</b>	Creates a cylindrical volume centered about the working plane origin.
<b>PCIRC</b>	Creates a circular area centered about the working plane origin.
<b>POLY</b>	Creates a polygonal area based on working plane coordinate pairs.
<b>PRI2</b>	Creates a polygonal area or a prism volume by vertices (GUI).
<b>PRISM</b>	Creates a prism volume based on working plane coordinate pairs.
<b>PTXY</b>	Defines coordinate pairs for use in polygons and prisms.
<b>RECTNG</b>	Creates a rectangular area anywhere on the working plane.
<b>RPOLY</b>	Creates a regular polygonal area centered about the working plane origin.
<b>RPR4</b>	Creates a regular polygonal area or prism volume anywhere on the working plane.
<b>RPRISM</b>	Creates a regular prism volume centered about the working plane origin.
<b>SPH4</b>	Creates a spherical volume anywhere on the working plane.
<b>SPH5</b>	Creates a spherical volume by diameter end points.
<b>SPHERE</b>	Creates a spherical volume centered about the working plane origin.
<b>TORUS</b>	Creates a toroidal volume.

**Table 2.30 Keypoints**

**These PREP7 commands are used to create, modify, list, etc., keypoints.**

<b>GSUM</b>	Calculates and prints geometry items.
<b>K</b>	Defines a keypoint.
<b>KBETW</b>	Creates a keypoint between two existing keypoints.
<b>KCENTER</b>	Creates a keypoint at the center of a circular arc defined by three locations.
<b>KDELE</b>	Deletes unmeshed keypoints.
<b>KDIST</b>	Calculates and lists the distance between two keypoints.
<b>KFILL</b>	Generates keypoints between two keypoints.
<b>KGEN</b>	Generates additional keypoints from a pattern of keypoints.
<b>KL</b>	Generates a keypoint at a specified location on an existing line.
<b>KLIST</b>	Lists the defined keypoints or hard points.
<b>KMODIF</b>	Modifies an existing keypoint.

**These PREP7 commands are used to create, modify, list, etc., keypoints.**

<b>KMOVE</b>	Calculates and moves a keypoint to an intersection.
<b>KNODE</b>	Defines a keypoint at an existing node location.
<b>KPLOT</b>	Displays the selected keypoints.
<b>KPSCALE</b>	Generates a scaled set of (meshed) keypoints from a pattern of keypoints.
<b>KSCALE</b>	Generates a scaled pattern of keypoints from a given keypoint pattern.
<b>KSUM</b>	Calculates and prints geometry statistics of the selected keypoints.
<b>KSYMM</b>	Generates a reflected set of keypoints.
<b>KTRAN</b>	Transfers a pattern of keypoints to another coordinate system.
<b>SOURCE</b>	Defines a default location for undefined nodes or keypoints.

**Table 2.31 Hard Points****These PREP7 commands are used to create, modify, list, etc., hard points.**

<b>HPTCREATE</b>	Defines a hard point.
<b>HPTDELETE</b>	Deletes selected hard points.

**Table 2.32 Lines****These PREP7 commands are used to create, modify, list, etc., lines.**

<b>BSPLIN</b>	Generates a single line from a spline fit to a series of keypoints.
<b>CIRCLE</b>	Generates circular arc lines.
<b>GSUM</b>	Calculates and prints geometry items.
<b>L</b>	Defines a line between two keypoints.
<b>L2ANG</b>	Generates a line at an angle with two existing lines.
<b>L2TAN</b>	Generates a line tangent to two lines.
<b>LANG</b>	Generates a straight line at an angle with a line.
<b>LARC</b>	Defines a circular arc.
<b>LAREA</b>	Generates the shortest line between two keypoints on an area.
<b>LCOMB</b>	Combines adjacent lines into one line.
<b>LDELE</b>	Deletes unmeshed lines.
<b>LDIV</b>	Divides a single line into two or more lines.
<b>LDRAG</b>	Generates lines by sweeping a keypoint pattern along path.
<b>LEXTND</b>	Extends a line at one end by using its slope.
<b>LFILLT</b>	Generates a fillet line between two intersecting lines.
<b>LGEN</b>	Generates additional lines from a pattern of lines.
<b>LLIST</b>	Lists the defined lines.
<b>LPLOT</b>	Displays the selected lines.
<b>LREVERSE</b>	Reverses the normal of a line, regardless of its connectivity or mesh status.

**These PREP7 commands are used to create, modify, list, etc., lines.**

<b>LROTAT</b>	Generates circular lines by rotating a keypoint pattern about an axis.
<b>LSSCALE</b>	Generates a scaled set of lines from a pattern of lines.
<b>LSTR</b>	Defines a straight line irrespective of the active coordinate system.
<b>LSUM</b>	Calculates and prints geometry statistics of the selected lines.
<b>LSYMM</b>	Generates lines from a line pattern by symmetry reflection.
<b>LTAN</b>	Generates a line at the end of, and tangent to, an existing line.
<b>LTRAN</b>	Transfers a pattern of lines to another coordinate system.
<b>SPLINE</b>	Generates a segmented spline through a series of keypoints.
<b>SSLN</b>	Selects and displays small lines in the model.

**Table 2.33 Areas****These PREP7 commands are used to create, modify, list, etc., areas.**

<b>A</b>	Defines an area by connecting keypoints.
<b>AATT</b>	Associates element attributes with the selected, unmeshed areas.
<b>ADELE</b>	Deletes unmeshed areas.
<b>ADGL</b>	Lists keypoints of an area that lie on a parametric degeneracy.
<b>ADRAG</b>	Generates areas by dragging a line pattern along a path.
<b>AFILLT</b>	Generates a fillet at the intersection of two areas.
<b>AGEN</b>	Generates additional areas from a pattern of areas.
<b>AL</b>	Generates an area bounded by previously defined lines.
<b>ALIST</b>	Lists the defined areas.
<b>ANORM</b>	Reorients area normals.
<b>AOFFST</b>	Generates an area, offset from a given area.
<b>APLOT</b>	Displays the selected areas.
<b>AREVERSE</b>	Reverses the normal of an area, regardless of its connectivity or mesh status.
<b>AROTAT</b>	Generates cylindrical areas by rotating a line pattern about an axis.
<b>ARSCALE</b>	Generates a scaled set of areas from a pattern of areas.
<b>ARSYM</b>	Generates areas from an area pattern by symmetry reflection.
<b>ASKIN</b>	Generates an area by "skinning" a surface through guiding lines.
<b>ASUB</b>	Generates an area using the shape of an existing area.
<b>ASUM</b>	Calculates and prints geometry statistics of the selected areas.
<b>ATRAN</b>	Transfers a pattern of areas to another coordinate system.
<b>GSUM</b>	Calculates and prints geometry items.

**These PREP7 commands are used to create, modify, list, etc., areas.**

<b>SPLOT</b>	Displays the selected areas and a faceted view of their underlying surfaces
--------------	---

**Table 2.34 Volumes****These PREP7 commands are used to create, modify, list, etc., volumes.**

<b>EXTOPT</b>	Controls options relating to the generation of volume elements from area elements.
<b>GSUM</b>	Calculates and prints geometry items.
<b>V</b>	Defines a volume through keypoints.
<b>VA</b>	Generates a volume bounded by existing areas.
<b>VDELE</b>	Deletes unmeshed volumes.
<b>VDGL</b>	Lists keypoints of a volume that lie on a parametric degeneracy.
<b>VDRAG</b>	Generates volumes by dragging an area pattern along a path.
<b>VEXT</b>	Generates additional volumes by extruding areas.
<b>VGEN</b>	Generates additional volumes from a pattern of volumes.
<b>VLIST</b>	Lists the defined volumes.
<b>VLSCALE</b>	Generates a scaled set of volumes from a pattern of volumes.
<b>VOFFST</b>	Generates a volume, offset from a given area.
<b>VPLOT</b>	Displays the selected volumes.
<b>VROTAT</b>	Generates cylindrical volumes by rotating an area pattern about an axis.
<b>VSUM</b>	Calculates and prints geometry statistics of the selected volumes.
<b>VSymm</b>	Generates volumes from a volume pattern by symmetry reflection.
<b>VTRAN</b>	Transfers a pattern of volumes to another coordinate system.

**Table 2.35 Booleans****These PREP7 commands are used to perform Boolean operations on solid model entities.**

<b>AADD</b>	Adds separate areas to create a single area.
<b>AGLUE</b>	Generates new areas by "gluing" areas.
<b>AINA</b>	Finds the intersection of areas.
<b>AINP</b>	Finds the pairwise intersection of areas.
<b>AINV</b>	Finds the intersection of an area with a volume.
<b>AOVLAP</b>	Overlaps areas.
<b>APTN</b>	Partitions areas.
<b>ASBA</b>	Subtracts areas from areas.
<b>ASBL</b>	Subtracts lines from areas.
<b>ASBV</b>	Subtracts volumes from areas.
<b>ASBW</b>	Subtracts the intersection of the working plane from areas (divides areas).
<b>BOPTN</b>	Specifies Boolean operation options.

**These PREP7 commands are used to perform Boolean operations on solid model entities.**

<b>BTOL</b>	Specifies the Boolean operation tolerances.
<b>LCSL</b>	Divides intersecting lines at their point(s) of intersection.
<b>LGLUE</b>	Generates new lines by "gluing" lines.
<b>LINA</b>	Finds the intersection of a line with an area.
<b>LINL</b>	Finds the common intersection of lines.
<b>LINP</b>	Finds the pairwise intersection of lines.
<b>LINV</b>	Finds the intersection of a line with a volume.
<b>LOVLAP</b>	Overlaps lines.
<b>LPTN</b>	Partitions lines.
<b>LSBA</b>	Subtracts areas from lines.
<b>LSBL</b>	Subtracts lines from lines.
<b>LSBV</b>	Subtracts volumes from lines.
<b>LSBW</b>	Subtracts the intersection of the working plane from lines (divides lines).
<b>VADD</b>	Adds separate volumes to create a single volume.
<b>VGLUE</b>	Generates new volumes by "gluing" volumes.
<b>VINP</b>	Finds the pairwise intersection of volumes.
<b>VINV</b>	Finds the intersection of volumes.
<b>VOVLAP</b>	Overlaps volumes.
<b>VPTN</b>	Partitions volumes.
<b>VSBA</b>	Subtracts areas from volumes.
<b>VSBV</b>	Subtracts volumes from volumes.
<b>VSBW</b>	Subtracts intersection of the working plane from volumes (divides volumes).

### Table 2.36 Meshing

**These PREP7 commands are used to mesh solid models with nodes and elements.**

<b>ACCAT</b>	Concatenates multiple areas in preparation for mapped meshing.
<b>ACLEAR</b>	Deletes nodes and area elements associated with selected areas.
<b>AESIZE</b>	Specifies the element size to be meshed onto areas.
<b>AMAP</b>	Generates a 2-D mapped mesh based on specified area corners.
<b>AMESH</b>	Generates nodes and area elements within areas.
<b>AREFINE</b>	Refines the mesh around specified areas.
<b>CHKMSH</b>	Checks area and volume entities for previous meshes.
<b>CLRMSHLN</b>	Clears meshed entities.
<b>CPCYC</b>	Couples the two side faces of a cyclically symmetric model for loadings that are the same on every segment.
<b>CZDEL</b>	Edits or clears cohesive zone sections.
<b>CZMESH</b>	Creates and meshes a cohesive zone group of elements.
<b>DESIZE</b>	Controls default element sizes.

**These PREP7 commands are used to mesh solid models with nodes and elements.**

<b>EORIENT</b>	Reorients solid element normals.
<b>EREFINE</b>	Refines the mesh around specified elements.
<b>ESIZE</b>	Specifies the default number of line divisions.
<b>ESYS</b>	Sets the element coordinate system attribute pointer.
<b>FVMESH</b>	Generates nodes and tetrahedral volume elements from detached exterior area elements (facets).
<b>GSGDATA</b>	Specifies the reference point and defines the geometry in the fiber direction for the generalized plane strain element option.
<b>IMESH</b>	Generates nodes and interface elements along lines or areas.
<b>KATT</b>	Associates attributes with the selected, unmeshed keypoints.
<b>KCLEAR</b>	Deletes nodes and point elements associated with selected keypoints.
<b>KESIZE</b>	Specifies the edge lengths of the elements nearest a keypoint.
<b>KMESH</b>	Generates nodes and point elements at keypoints.
<b>KREFINE</b>	Refines the mesh around specified keypoints.
<b>KSCON</b>	Specifies a keypoint about which an area mesh will be skewed.
<b>LATT</b>	Associates element attributes with the selected, unmeshed lines.
<b>LCCAT</b>	Concatenates multiple lines into one line for mapped meshing.
<b>LCLEAR</b>	Deletes nodes and line elements associated with selected lines.
<b>LESIZE</b>	Specifies the divisions and spacing ratio on unmeshed lines.
<b>LMESH</b>	Generates nodes and line elements along lines.
<b>LREFINE</b>	Refines the mesh around specified lines.
<b>MAT</b>	Sets the element material attribute pointer.
<b>MCHECK</b>	Checks mesh connectivity.
<b>MODMSH</b>	Controls the relationship of the solid model and the FE model.
<b>MOPT</b>	Specifies meshing options.
<b>MSHAPE</b>	For elements that support multiple shapes, specifies the element shape to be used for meshing.
<b>MSHCOPY</b>	Simplifies the generation of meshes that have matching node element patterns on two different line groups (in 2-D) or area groups (3-D).
<b>MSHKEY</b>	Specifies whether free meshing or mapped meshing should be used to mesh a model.
<b>MSHMID</b>	Specifies placement of midside nodes.
<b>MSHPATTERN</b>	Specifies pattern to be used for mapped triangle meshing.
<b>NREFINE</b>	Refines the mesh around specified nodes.
<b>PSMESH</b>	Splits an initially continuous group of beam, shell, plane, or solid elements into two unconnected groups, tied together with <b>PRETS179</b> pretension elements.
<b>REAL</b>	Sets the element real constant set attribute pointer.
<b>RTHICK</b>	Defines variable thickness at nodes for shell elements.



**These PREP7 commands are used to mesh solid models with nodes and elements.**

<b>SHPP</b>	Controls element shape checking.
<b>SMRTSIZE</b>	Specifies meshing parameters for automatic (smart) element sizing.
<b>TCHG</b>	Converts 20-node degenerate tetrahedral elements to their 10-node non-degenerate counterparts.
<b>TIMP</b>	Improves the quality of tetrahedral elements that are not associated with a volume.
<b>TYPE</b>	Sets the element type attribute pointer.
<b>VATT</b>	Associates element attributes with the selected, unmeshed volumes.
<b>VCLEAR</b>	Deletes nodes and volume elements associated with selected volumes.
<b>VIMP</b>	Improves the quality of the tetrahedral elements in the selected volume(s).
<b>VMESH</b>	Generates nodes and volume elements within volumes.
<b>VEORIENT</b>	Specifies brick element orientation for volume mapped (hexahedron) meshing.
<b>VSWEEP</b>	Fills an existing unmeshed volume with elements by sweeping the mesh from an adjacent area throughout the volume.

**Table 2.37 Nodes****These PREP7 commands are used to create, modify, list, etc., nodes.**

<b>CENTER</b>	Defines a node at the center of curvature of 2 or 3 nodes.
<b>FILL</b>	Generates a line of nodes between two existing nodes.
<b>MOVE</b>	Calculates and moves a node to an intersection.
<b>N</b>	Defines a node.
<b>NAXIS</b>	Generates or clears nodes for general axisymmetric element sections.
<b>NANG</b>	Rotates a nodal coordinate system by direction cosines.
<b>NDELE</b>	Deletes nodes.
<b>NDIST</b>	Calculates and lists the distance between two nodes.
<b>NGEN</b>	Generates additional nodes from a pattern of nodes.
<b>NKPT</b>	Defines a node at an existing keypoint location.
<b>NLIST</b>	Lists nodes.
<b>NMODIF</b>	Modifies an existing node.
<b>NORA</b>	Rotates nodal coordinate systems to surface normal.
<b>NPLOT</b>	Displays nodes.
<b>NREAD</b>	Reads nodes from a file.
<b>NROTAT</b>	Rotates nodal coordinate systems into the active system.
<b>NRANG</b>	Specifies the range of nodes to be read from the node file.
<b>NSCALE</b>	Generates a scaled set of nodes from a pattern of nodes.
<b>NSMOOTH</b>	Smooths selected nodes among selected elements.
<b>NSYM</b>	Generates a reflected set of nodes.

**These PREP7 commands are used to create, modify, list, etc., nodes.**

<b>NWRITE</b>	Writes nodes to a file.
<b>QUAD</b>	Generates a quadratic line of nodes from three nodes.
<b>SOURCE</b>	Defines a default location for undefined nodes or keypoints.
<b>TRANSFER</b>	Transfers a pattern of nodes to another coordinate system.

**Table 2.38 Elements****These PREP7 commands are used to create, modify, list, etc., elements.**

<b>AFSURF</b>	Generates surface elements overlaid on the surface of existing solid elements and assigns the extra node as the closes fluid element node.
<b>E</b>	Defines an element by node connectivity.
<b>EDELE</b>	Deletes selected elements from the model.
<b>EGEN</b>	Generates elements from an existing pattern.
<b>EINTF</b>	Defines two-node elements between coincident nodes.
<b>ELIST</b>	Lists the elements.
<b>EMID</b>	Adds or removes midside nodes.
<b>EMODIF</b>	Modifies a previously defined element.
<b>EMORE</b>	Adds more nodes to the just-defined element.
<b>EMTGEN</b>	Generates a set of TRANS126 elements
<b>EN</b>	Defines an element by its number and node connectivity.
<b>ENDRELEASE</b>	Specifies degrees of freedom to be decoupled for end release.
<b>ENGEN</b>	Generates elements from an existing pattern.
<b>ENORM</b>	Reorients shell element normals or line element node connectivity.
<b>ENSYM</b>	Generates elements by symmetry reflection.
<b>EORIENT</b>	Reorients solid element normals.
<b>EPLLOT</b>	Produces an element display.
<b>EREAD</b>	Reads elements from a file.
<b>EREINF</b>	Generates reinforcing elements from selected existing (base) elements.
<b>ERRANG</b>	Specifies the element range to be read from a file.
<b>ESURF</b>	Generates elements overlaid on the free faces of existing elements.
<b>ESYM</b>	Generates elements from a pattern by a symmetry reflection.
<b>ESYS</b>	Sets the element coordinate system attribute pointer.
<b>EWRITE</b>	Writes elements to a file.
<b>FIPLLOT</b>	Plots a <b>FiberSIM</b> model.
<b>INISTATE</b>	Applies initial state data to an element or a selection of elements.
<b>LAYLIST</b>	Lists real constants material properties for layered elements.
<b>LAYPLOT</b>	Displays the layer stacking sequence for layered elements.

**These PREP7 commands are used to create, modify, list, etc., elements.**

<b>LFSURF</b>	Generates surface elements overlaid on the edge of existing solid elements and assigns the extra node as the closest fluid element node.
<b>MAT</b>	Sets the element material attribute pointer.
<b>NDSURF</b>	Generates surface elements overlaid on the edge of existing elements and assigns the extra node as the closest fluid element node.
<b>REAL</b>	Sets the element real constant set attribute pointer.
<b>SWADD</b>	Adds more surfaces to an existing spot weld set.
<b>SWDEL</b>	Deletes spot weld sets.
<b>SWGGEN</b>	Creates a new spot weld set.
<b>SWLIST</b>	Lists spot weld sets.
<b>TSHAP</b>	Defines simple 2-D and 3-D geometric surfaces for target segment elements.
<b>TYPE</b>	Sets the element type attribute pointer.
<b>UPGEOM</b>	Adds displacements from a previous analysis and updates the geometry to the deformed configuration.
<b>USRDOF</b>	Specifies the degrees of freedom for the user-defined element <a href="#">USER300</a> .
<b>USRELEM</b>	Specifies the characteristics of the user-defined element <a href="#">USER300</a> .

**Table 2.39 Superelements**

**These PREP7 commands are used to create and modify superelements.**

<b>SE</b>	Defines a superelement.
<b>SEDLIST</b>	Lists the degree-of-freedom solution of a superelement after the use pass.
<b>SELIST</b>	Lists the contents of a superelement matrix file.
<b>SESYMM</b>	Performs a symmetry operation on a superelement within the use pass.
<b>SETRAN</b>	Creates a superelement from an existing superelement.

**Table 2.40 Digitizing**

**These PREP7 commands are used to define nodes by tablet digitizing.**

<b>DIG</b>	Digitizes nodes to a surface.
<b>DMOVE</b>	Digitizes nodes on surfaces and along intersections.
<b>DSET</b>	Sets the scale and drawing plane orientation for a digitizing tablet.
<b>DSURF</b>	Defines the surface upon which digitized nodes lie.

**Table 2.41 Coupled Degrees of Freedom**

**These PREP7 commands are used to define, modify, list, etc., coupled degrees of freedom.**

<b>CP</b>	Defines (or modifies) a set of coupled degrees of freedom.
<b>CPDELE</b>	Deletes coupled degree of freedom sets.

**These PREP7 commands are used to define, modify, list, etc., coupled degrees of freedom.**

<b>CPINTF</b>	Defines coupled degrees of freedom at an interface.
<b>CPLGEN</b>	Generates sets of coupled nodes from an existing set.
<b>CPLIST</b>	Lists the coupled degree of freedom sets.
<b>CPNGEN</b>	Defines, modifies, or adds to a set of coupled degrees of freedom.
<b>CPSGEN</b>	Generates sets of coupled nodes from existing sets.

**Table 2.42 Constraint Equations**

**These PREP7 commands are used to define, modify, list, etc., constraint equations.**

<b>CE</b>	Defines a constraint equation relating degrees of freedom.
<b>CECYC</b>	Generates the constraint equations for a cyclic symmetry analysis.
<b>CEDELE</b>	Deletes constraint equations.
<b>CEINTF</b>	Generates constraint equations at an interface.
<b>CELIST</b>	Lists the constraint equations.
<b>CERIG</b>	Defines a rigid region.
<b>CESGEN</b>	Generates a set of constraint equations from existing sets.
<b>RBE3</b>	Distributes the force/moment applied at the master node to a set of slave nodes, taking into account the geometry of the slave nodes as well as weighting factors.

**Table 2.43 Element Reordering**

**These PREP7 commands are used to reorder the model wavefront.**

<b>NOORDER</b>	Re-establishes the original element ordering.
<b>WAVES</b>	Initiates reordering.
<b>WERASE</b>	Erases all reordering wave lists.
<b>WFRONT</b>	Estimates wavefront statistics.
<b>WMID</b>	Specifies reordering options for the <b>WAVES</b> command.
<b>WMORE</b>	Adds more nodes to the starting wave list.
<b>WSORT</b>	Initiates element reordering based upon a geometric sort.
<b>WSTART</b>	Defines a starting wave list.

**Table 2.44 FLOTRAN Options**

**These PREP7 commands are used to specify the options and output controls for a FLOTRAN CFD analysis.**

<b>FLDATA</b>	Sets up a FLOTRAN analysis.
<b>FLDATA1</b>	Controls which features of the solution algorithm are activated.
<b>FLDATA2</b>	Sets iteration and output controls for steady state analyses.
<b>FLDATA3</b>	Sets the convergence monitors for the degree of freedom set.
<b>FLDATA4</b>	Sets controls for transient analyses based on transient time and convergence monitors or sets time integration method.

These PREP7 commands are used to specify the options and output controls for a FLOTRAN CFD analysis.

<b>FLDATA4A</b>	Sets controls for transient analyses based on the number of time steps.
<b>FLDATA5</b>	Sets output and storage controls.
<b>FLDATA6</b>	Controls the output of the convergence monitor.

**Table 2.45 FLOTRAN Property**

These PREP7 commands are used to specify the fluid properties for a FLOTRAN CFD analysis.

<b>FLDATA7</b>	Specifies the type of fluid property.
<b>FLDATA8</b>	Specifies the NOM1 coefficient of the fluid property equation.
<b>FLDATA9</b>	Specifies the COF1 coefficient of the fluid property equation.
<b>FLDATA10</b>	Specifies the COF2 coefficient of the fluid property equation.
<b>FLDATA11</b>	Specifies the COF3 coefficient of the fluid property equation.
<b>FLDATA12</b>	Sets the property update frequency flag.
<b>FLDATA13</b>	Sets the property variation flag.

**Table 2.46 FLOTRAN Operating**

These PREP7 commands are used to specify the operating conditions for a FLOTRAN CFD analysis.

<b>FLDATA14</b>	Specifies the reference temperature.
<b>FLDATA15</b>	Specifies the reference pressure.
<b>FLDATA16</b>	Specifies the bulk modulus parameter.
<b>FLDATA17</b>	Specifies the specific heat ratio.

**Table 2.47 FLOTRAN Solver**

These PREP7 commands are used to control the solver for a FLOTRAN CFD analysis.

<b>FLDATA18</b>	Selects the algebraic solver.
<b>FLDATA19</b>	Specifies the number of TDMA sweeps.
<b>FLDATA20</b>	Specifies the number of conjugate direction search vectors.
<b>FLDATA20A</b>	Specifies the amount of fill-in when preconditioning the coefficient matrix.
<b>FLDATA20B</b>	Specifies the number of fill-ins for the ILU preconditioner.
<b>FLDATA21</b>	Specifies the convergence monitor.
<b>FLDATA22</b>	Specifies the maximum number of semi-direct iterations.

**These PREP7 commands are used to control the solver for a FLOTRAN CFD analysis.**

**FLDATA23** Specifies the solver minimum normalized rate of change.

### Table 2.48 FLOTRAN Turbulence

**These PREP7 commands are used to control the turbulence setting for a FLOTRAN CFD analysis.**

**FLDATA24** Sets the turbulence model and the constants used in the Standard k- $\epsilon$  Model and the Zero Equation Turbulence Model.

**FLDATA24A** Sets constants for the Re-Normalized Group Turbulence Model (RNG).

**FLDATA24B** Sets constants for the New k- $\epsilon$  Model due to Shih (NKE).

**FLDATA24C** Sets constants for the Nonlinear Model of Girimaji (GIR).

**FLDATA24D** Sets constants for the Shih, Zhu, Lumley Model (SZL).

**FLDATA24E** Sets constants for the k- $\omega$  Model.

**FLDATA24F** Sets the turbulent production clip factor for the Shear Stress Transport (SST) model.

**FLDATA24G** Sets constants in the k- $\omega$  regime for the Shear Stress Transport (SST) model.

**FLDATA24H** Sets constants in the k- $\epsilon$  regime for the Shear Stress Transport (SST) model.

**FLDATA40** Controls activation of thermal stabilization near walls.

### Table 2.49 FLOTRAN Stability

**These PREP7 commands are used to control the stability settings for a FLOTRAN CFD analysis.**

**FLDATA25** Sets solution and property relaxation factors.

**FLDATA26** Sets stability controls.

**FLDATA34** Sets modified inertial relaxation factors.

### Table 2.50 FLOTRAN Miscellaneous

**These PREP7 commands are used to control miscellaneous settings for a FLOTRAN CFD analysis.**

**FLDATA27** Controls dependent variable printing.

**FLDATA28** Specifies that variable results are to be replaced.

**FLDATA29** Re-initializes a results variable.

**FLDATA30** Controls the quadrature orders.

**FLDATA31** Specifies dependent variable caps.

**FLDATA32** Controls restart options.

**FLDATA33** Specifies the approach to discretize the advection term.

**FLDATA35** Specifies tolerances for the lower and upper bound of the volume fraction.

**FLDATA36** Specifies ambient reference values outside of the fluid for the volume of fluid (VOF) method.

**These PREP7 commands are used to control miscellaneous settings for a FLOTRAN CFD analysis.**

<b>FLDATA37</b>	Specifies the segregated solution algorithm.
<b>FLDATA38</b>	Specifies the mass type for a fluid transient analysis.
<b>FLDATA39</b>	Specifies remeshing parameters for transient fluid flow and fluid-solid interaction analyses.
<b>ICVFRC</b>	Sets the initial volume fraction field for a geometry.
<b>PERI</b>	Specifies periodic boundary conditions in an incompressible flow analysis.
<b>PLVFRC</b>	Displays volume fractions in a volume of fluid (VOF) analysis.

**Table 2.51 FLOTRAN Multiple Species**

**These PREP7 commands are used for multiple species transport in a FLOTRAN CFD analysis.**

<b>MSADV</b>	Specifies the approach to discretize the advection term in a species transport equation.
<b>MSCAP</b>	Activates and controls mass fraction capping for a species.
<b>MSDATA</b>	Defines multiple species data applicable to all species.
<b>MSMASS</b>	Specifies the mass type for a transient species analysis.
<b>MSMETH</b>	Specifies the method of solution of the species transport equations.
<b>MSMIR</b>	Sets modified inertial relaxation factors for multiple species.
<b>MSNOMF</b>	Specifies the initial value of nominal mass fraction for a species.
<b>MSPROP</b>	Defines the fluid properties of a species.
<b>MSQUAD</b>	Specifies the quadrature order for multiple species elements.
<b>MSRELAX</b>	Specifies relaxation factors for a multiple species transport analysis.
<b>MSSOLU</b>	Specifies solution options for multiple species transport.
<b>MSSPEC</b>	Specifies the name, molecular weight, and Schmidt number of a species.
<b>MSTERM</b>	Sets the convergence monitors for species.
<b>MSVARY</b>	Allows species properties to vary between global iterations.

**Table 2.52 Status**

**These PREP7 commands are for use with the STAT command.**

<b>AREAS</b>	Specifies "Areas" as the subsequent status topic.
<b>BOOL</b>	Specifies "Booleans" as the subsequent status topic.
<b>CEQN</b>	Specifies "Constraint equations" as the subsequent status topic.
<b>COUPLE</b>	Specifies "Node coupling" as the subsequent status topic.
<b>DIGIT</b>	Specifies "Node digitizing" as the subsequent status topic.
<b>ELEM</b>	Specifies "Elements" as the subsequent status topic.
<b>ETYPE</b>	Specifies "Element types" as the subsequent status topic.
<b>FATIGUE</b>	Specifies "Fatigue data status" as the subsequent status topic.
<b>FEBODY</b>	Specifies "Body loads on elements" as the subsequent status topic.

**These PREP7 commands are for use with the STAT command.**

<b>FECONS</b>	Specifies "Constraints on nodes" as the subsequent status topic.
<b>FEFOR</b>	Specifies "Forces on nodes" as the subsequent status topic.
<b>FESURF</b>	Specifies "Surface loads on elements" as the subsequent status topic.
<b>FLOTRAN</b>	Specifies "FLOTRAN data settings" as the subsequent status topic.
<b>GEOMETRY</b>	Specifies "Geometry" as the subsequent status topic.
<b>KEYPTS</b>	Specifies "Keypoints" as the subsequent status topic.
<b>LINE</b>	Specifies "Lines" as the subsequent status topic.
<b>MATER</b>	Specifies "Material properties" as the subsequent status topic.
<b>MESHING</b>	Specifies "Meshing" as the subsequent status topic.
<b>NODES</b>	Specifies "Nodes" as the subsequent status topic.
<b>PRIM</b>	Specifies "Solid model primitives" as the subsequent status topic.
<b>RCON</b>	Specifies "Real constants" as the subsequent status topic.
<b>REORDER</b>	Specifies "Model reordering" as the subsequent status topic.
<b>SELM</b>	Specifies "Superelements" as the subsequent status topic.
<b>TBLE</b>	Specifies "Data table properties" as the subsequent status topic.
<b>VOLUMES</b>	Specifies "Volumes" as the subsequent status topic.

**Table 2.53 Explicit Dynamics****These PREP7 commands are used for an explicit dynamic analysis.**

<b>EDASMP</b>	Creates a part assembly to be used in an explicit dynamic analysis.
<b>EDBOUND</b>	Defines a boundary plane for sliding or cyclic symmetry.
<b>EDBX</b>	Creates a box shaped volume to be used in a contact definition.
<b>EDCGEN</b>	Specifies contact parameters for an explicit dynamic analysis.
<b>EDCLIST</b>	Lists all contact entity specifications in an explicit dynamic analysis.
<b>EDCMORE</b>	Specifies additional contact parameters for a given contact definition in an explicit dynamic analysis.
<b>EDCNSTR</b>	Defines various types of constraints for an explicit dynamic analysis.
<b>EDCONTACT</b>	Specifies contact surface controls for an explicit dynamic analysis.
<b>EDCRB</b>	Constrains two rigid bodies to act as one in an explicit dynamic analysis.
<b>EDCURVE</b>	Specifies data curves for an explicit dynamic analysis.
<b>EDDBL</b>	Sets the single or double precision version of LS-DYNA into effect.
<b>EDDC</b>	Deletes or deactivates/reactivates contact entity specifications in an explicit dynamic analysis.
<b>EDIPART</b>	Defines inertia for rigid parts in an explicit dynamic analysis.
<b>EDLCS</b>	Defines a local coordinate system for use in explicit dynamic analysis.
<b>EDMP</b>	Defines material properties for an explicit dynamic analysis.
<b>EDNB</b>	Defines a nonreflecting boundary in an explicit dynamic analysis.



**These PREP7 commands are used for an explicit dynamic analysis.**

<b>EDNDTSD</b>	Allows smoothing of noisy data for explicit dynamic analyses and provides a graphical representation of the data.
<b>EDNROT</b>	Applies a rotated coordinate nodal constraint in an explicit dynamic analysis.
<b>EDPART</b>	Configures parts for an explicit dynamic analysis.
<b>EDPC</b>	Selects and plots explicit dynamic contact entities.
<b>EDSP</b>	Specifies small penetration checking for contact entities in an explicit dynamic analysis.
<b>EDWELD</b>	Defines a massless spotweld or generalized weld for use in an explicit dynamic analysis.

**Table 2.54 Sections****These PREP7 commands manage sections.**

<b>BSAX</b>	Specifies axial strain and force for a <a href="#">nonlinear general beam section</a> .
<b>BSM1</b>	Specifies bending moment and curvature in plane XZ for a <a href="#">nonlinear general beam section</a> .
<b>BSM2</b>	Specifies bending moment and curvature in plane XY for a <a href="#">nonlinear general beam section</a> .
<b>BSMD</b>	Specifies mass density for a <a href="#">nonlinear general beam section</a> .
<b>BSS1</b>	Specifies transverse shear force and strain in plane XZ for a <a href="#">nonlinear general beam section</a> .
<b>BSS2</b>	Specifies transverse shear force and strain in plane XY for a <a href="#">nonlinear general beam section</a> .
<b>BSTE</b>	Specifies a thermal expansion coefficient for a <a href="#">nonlinear general beam section</a> .
<b>BSTQ</b>	Specifies cross section torque and twist for a <a href="#">nonlinear general beam section</a> .
<b>CBMD</b>	Specifies preintegrated mass-density matrix for <a href="#">composite-beam sections</a> .
<b>CBMX</b>	Specifies preintegrated cross-section stiffness for <a href="#">composite-beam sections</a> .
<b>CBTE</b>	Specifies a thermal expansion coefficient for a <a href="#">composite-beam section</a> .
<b>CBTMP</b>	Specifies a temperature for the <a href="#">composite-beam section</a> matrix.
<b>SDELETE</b>	Deletes sections from the database.
<b>SECCONTROLS</b>	Supplements or overrides default section properties.
<b>SECDATA</b>	Describes the geometry of a section.
<b>SECJOINT</b>	Defines local coordinate systems at joint element nodes and other data for joint elements.
<b>/SECLIB</b>	Sets the default section library path for the <b>SECREAD</b> command.
<b>SECLOCK</b>	Specifies locks on the element degrees of freedom.
<b>SECNUM</b>	Sets the element section attribute pointer.

**These PREP7 commands manage sections.**

<b>SECOFFSET</b>	Defines the section offset for cross sections.
<b>SECPLOT</b>	Plots the geometry of a section to scale.
<b>SECREAD</b>	Reads a custom section library or a user-defined section mesh into ANSYS.
<b>SECSTOP</b>	Specifies stops on the element degrees of freedom.
<b>SECTYPE</b>	Associates section type information with a section ID number.
<b>SECWRITE</b>	Creates an ASCII file containing user mesh section information.
<b>SFLEX</b>	Sets section flexibility factors for pipe elements.
<b>SLIST</b>	Summarizes the section properties for all defined sections in the current session of ANSYS.
<b>SOCEAN</b>	Associates an ocean environment with the most recently defined element section. Used with the <a href="#">ocean</a> family of commands.
<b>SSBT</b>	Specifies the bending thermal effects quantity for a <a href="#">preintegrated shell section</a> .
<b>SSMT</b>	Specifies the membrane thermal effects quantity for a <a href="#">preintegrated shell section</a> .
<b>SSPA</b>	Specifies the membrane stiffness quantity for a <a href="#">preintegrated shell section</a> .
<b>SSPB</b>	Specifies the coupling stiffness quantity for a <a href="#">preintegrated shell section</a> .
<b>SSPD</b>	Specifies the bending stiffness quantity for a <a href="#">preintegrated shell section</a> .
<b>SSPE</b>	Specifies the transverse shear stiffness quantity for a <a href="#">preintegrated shell section</a> .
<b>SSPM</b>	Specifies the mass density and stress-free initial temperature for a <a href="#">preintegrated shell section</a> .

**Table 2.55 Ocean****These PREP7 commands define ocean environment data.**

To associate your ocean environment with an element section, issue the **SOCEAN** command.

<b>OCDATA</b>	Defines an ocean environment using non-table data.
<b>OCDELETE</b>	Deletes all ocean environment data associated with the specified ocean ID number ( <i>OCID</i> ) from the ANSYS database.
<b>OCLIST</b>	Summarizes all currently defined ocean environments.
<b>OCTABLE</b>	Defines an ocean environment using table data.

These PREP7 commands define ocean environment data.

To associate your ocean environment with an element section, issue the **SOCEAN** command.

**OCTYPE** Specifies the type of ocean environment data to follow (basic, current, or wave).

### Table 2.56 Morphing

These PREP7 commands are used to adjust the finite element mesh in the non-structural regions to coincide with the deflections of the structural regions.

**MORPH** Specifies morphing and remeshing controls.

**DAMORPH** Move nodes in selected areas to conform to structural displacements.

**DEMORPH** Move nodes in selected elements to conform to structural displacements.

**DVMORPH** Move nodes in selected volumes to conform to structural displacements.

### Table 2.57 Trefftz Domain

These PREP7 commands are used in the Trefftz method for modeling the open domain.

**TZAMESH** Meshes the areas of a volume to create Trefftz nodes.

**TZDELE** Deletes the Trefftz superelement, associated constraint equations and all supporting Trefftz files.

**TZEGEN** Generates a Trefftz domain substructure and defines a Trefftz superelement for use in electrostatic analysis.

### Table 2.58 Perfectly Matched Layers

This PREP7 command is used to create perfectly matched layers (PMLs) designed to absorb high frequency waves.

**PMLOPT** Defines perfectly matched layers (PMLs) for a high-frequency analysis.

**PMLSIZE** Determines number of PML layers.

### Table 2.59 Special Purpose

These PREP7 commands are used for special-purpose operations.

**/CYCEXPAND** Verifies a cyclically symmetric model by graphically expanding it partially or into the full 360 degrees.

**CYCLIC** Specifies a cyclic symmetry analysis.

**CYCOPT** Specifies solution options for a cyclic symmetry analysis.

**EMSYM** Specifies circular symmetry for electromagnetic sources.

**HFEREFINE** Automatically refines high-frequency tetrahedral elements (**HF119**) or lists high-frequency brick elements (**HF120**) with the largest error.

**MSTOLE** Adds two extra nodes from **FLUID116** elements to **SURF152** elements for convection analyses.

**These PREP7 commands are used for special-purpose operations.**

<b>PERBC2D</b>	Generates periodic constraints for 2-D planar magnetic field analyses.
<b>PHYSICS</b>	Writes, reads, or lists all element information.
<b>RACE</b>	Defines a "racetrack" current source.

**2.6. SOLUTION Commands**

These commands are used to load and solve the model. The commands are grouped by functionality.

**Table 2.60 Analysis Options****These SOLUTION commands are used to set general analysis options.**

<b>ABEXTRACT</b>	Extracts the alpha-beta damping multipliers for Rayleigh damping.
<b>ACCOPTION</b>	Specifies GPU accelerator capability options.
<b>ADAMS</b>	Performs solutions and writes flexible body information to a modal neutral file.
<b>ADAPT</b>	Adaptively meshes and solves a model.
<b>ANTYPE</b>	Specifies the analysis type and restart status.
<b>BCSOPTION</b>	Sets memory option for the sparse solver.
<b>CECHECK</b>	Check constraint equations and couplings for rigid body motions.
<b>CHECK</b>	Checks current database items for completeness.
<b>CINT</b>	Defines parameters associated with contour integral calculations.
<b>CMATRIX</b>	Performs electrostatic field solutions and calculates the self and mutual capacitances between multiple conductors.
<b>CMSOPT</b>	Specifies <b>component mode synthesis</b> (CMS) analysis options.
<b>CNCHECK</b>	Provides and/or adjusts the initial status of contact pairs.
<b>CNKMOD</b>	Modifies contact element key options.
<b>CUTCONTROL</b>	Controls time-step cutback during a nonlinear solution.
<b>CYCOPT</b>	Specifies the harmonic index solution for a cyclic symmetry analysis.
<b>DDOPTION</b>	Sets domain decomposer option for Distributed ANSYS.
<b>DMPEXT</b>	Extracts modal damping coefficients in a specified frequency range.
<b>DSPOPTION</b>	Sets the memory option for the distributed sparse solver.
<b>EMATWRITE</b>	Forces the writing of all the element matrices to File.EMAT.
<b>EQSLV</b>	Specifies the type of equation solver.
<b>ERESX</b>	Specifies extrapolation of integration point results.
<b>ESCHECK</b>	Perform element shape checking for a selected element set.
<b>ESSOLV</b>	Performs a coupled electrostatic-structural analysis.
<b>EXPASS</b>	Specifies an expansion pass of an analysis.
<b>GAUGE</b>	Gauges the problem domain for an edge-element formulation.
<b>GMATRIX</b>	Performs electric field solutions and calculates the self and mutual conductances between multiple conductors.
<b>HFADP</b>	Turns a high-frequency adaptive error calculation on or off.

**These SOLUTION commands are used to set general analysis options.**

<b>HFEIGOPT</b>	Specifies high frequency electromagnetic modal analysis options.
<b>HFMODPRT</b>	Calculates electromagnetic field distribution for a modal port.
<b>HFPA</b>	Specifies a radiation scan angle for a phased array antenna analysis.
<b>HFPCSWP</b>	Calculates the propagating constants of a transmission line or waveguide over a frequency range.
<b>HFSCAT</b>	Specifies a high-frequency scattering analysis.
<b>LMATRIX</b>	Calculates the differential inductance matrix and the total flux linkage in each coil for an N-winding system.
<b>LPRT</b>	Defines impedance and calibration lines for excitation eigenfield.
<b>LUMPM</b>	Specifies a lumped mass matrix formulation.
<b>MADAPT</b>	Adaptively meshes and solves an edge-based model.
<b>MONITOR</b>	Controls contents of three variable fields in nonlinear solution monitor file.
<b>MSAVE</b>	Sets the memory saving feature for the PCG and DPCG solvers.
<b>OPNCONTROL</b>	Sets decision parameter for automatically increasing the time step interval.
<b>PCGOPT</b>	Controls PCG solver options.
<b>PERTURB</b>	Sets linear perturbation analysis options.
<b>PSCONTROL</b>	Turns off shared-memory parallel operations during solution.
<b>PSOLVE</b>	Directs the program to perform a partial solution.
<b>RATE</b>	Specifies whether the effect of creep strain rate will be used in the solution of a load step.
<b>RESVEC</b>	Calculates residual vectors in a modal analysis, or includes residual vectors in a modal superposition transient/harmonic analysis.
<b>RSTOFF</b>	Offsets node or element IDs in the FE geometry record.
<b>SEEXP</b>	Specifies options for the substructure expansion pass.
<b>SEGEN</b>	Automatically generate superelements.
<b>SEOPT</b>	Specifies substructure analysis options.
<b>SNOPTION</b>	Specifies Supernode (SNODE) eigensolver options.
<b>SOLCONTROL</b>	Specifies whether to use optimized nonlinear solution defaults and some enhanced internal solution algorithms.
<b>/SOLU</b>	Enters the solution processor.
<b>SOLVE</b>	Starts a solution.
<b>SPADP</b>	Automatically refines a <b>HF119</b> tetrahedral element mesh based on S-parameter convergence.
<b>SPFSS</b>	Performs frequency sweep or angle sweep calculations to determine reflection and transmission parameters of a frequency selective surface.
<b>SPSCAN</b>	Performs a harmonic analysis of a unit cell over a range of angles and extracts the S-parameter.
<b>SPSWP</b>	Computes S-parameters over a frequency range and writes them to a file.

**These SOLUTION commands are used to set general analysis options.**

<b>STABILIZE</b>	Activates stabilization for all elements that support nonlinear stabilization.
<b>STAOPT</b>	Specifies static analysis options.
<b>TOFFST</b>	Specifies the temperature offset from absolute zero to zero.

**Table 2.61 Nonlinear Options****These SOLUTION commands are used to define options for nonlinear analyses.**

<b>ARCLEN</b>	Activates the arc-length method.
<b>ARCTRM</b>	Controls termination of the solution when the arc-length method ( <b>ARCLEN,ON</b> ) is used.
<b>BUCOPT</b>	Specifies buckling analysis options.
<b>CNVTOL</b>	Sets convergence values for nonlinear analyses.
<b>CRPLIM</b>	Specifies the creep criterion for automatic time stepping.
<b>/GST</b>	Turns Graphical Solution Tracking (GST) on or off.
<b>LNSRCH</b>	Activates a line search to be used with Newton-Raphson.
<b>MXPAND</b>	Specifies the number of modes to expand and write for a modal or buckling analysis.
<b>NCNV</b>	Sets the key to terminate an analysis.
<b>NEQIT</b>	Specifies the maximum number of equilibrium iterations for nonlinear analyses.
<b>NLDIAG</b>	Sets nonlinear diagnostics functionality.
<b>NLGEOM</b>	Includes large deformation effects in a static or full transient analysis.
<b>NLHIST</b>	Specify result items to track during solution.
<b>NROPT</b>	Specifies the Newton-Raphson options in a static or full transient analysis.
<b>PRED</b>	Activates a predictor in a nonlinear analysis.
<b>PSTRES</b>	Specifies whether prestress effects are calculated or included.
<b>SSTIF</b>	Activates stress stiffness effects in a nonlinear analysis.

**Table 2.62 Dynamic Options****These SOLUTION commands are used to define options for dynamic analyses.**

<b>ALPHAD</b>	Defines the mass matrix multiplier for damping.
<b>BETAD</b>	Defines the stiffness matrix multiplier for damping.
<b>DMPRAT</b>	Sets a constant damping ratio.
<b>FRQSCL</b>	Turns on automatic scaling of the entire mass matrix and frequency range for modal analyses using the Block Lanczos, PCG Lanczos, or Supernode mode extraction method.
<b>HARFRQ</b>	Defines the frequency range in the harmonic response analysis.
<b>HREXP</b>	Specifies the phase angle for the harmonic analysis expansion pass.
<b>HROPT</b>	Specifies harmonic analysis options.

**These SOLUTION commands are used to define options for dynamic analyses.**

<b>HROUT</b>	Specifies the harmonic analysis output options.
<b>LVSCALE</b>	Scales the load vector for mode superposition analyses.
<b>MDAMP</b>	Defines the damping ratios as a function of mode.
<b>MDPLOT</b>	Plots frequency-dependent modal damping coefficients.
<b>MIDTOL</b>	Sets middle step residual criterion values for structural transient analysis.
<b>MODCONT</b>	Creates multiple load vectors in a modal analysis.
<b>MODEPT</b>	Specifies modal analysis options.
<b>MXPAND</b>	Specifies the number of modes to expand and write for a modal or buckling analysis.
<b>QRDOPT</b>	Specifies additional QRDAMP modal analysis option.
<b>RIGID</b>	Specifies known rigid body modes (if any) of the model.
<b>TIMINT</b>	Turns on transient effects.
<b>TINTP</b>	Defines transient integration parameters.
<b>TRNOPT</b>	Specifies transient analysis options.

**Table 2.63 Spectrum Options**

**These SOLUTION commands are used to define options for spectrum analyses.**

<b>ADDAM</b>	Specifies the acceleration spectrum computation constants for the analysis of shock resistance of shipboard structures.
<b>COVAL</b>	Defines PSD cospectral values.
<b>CQC</b>	Specifies the complete quadratic mode combination method.
<b>DSUM</b>	Specifies the double sum mode combination method.
<b>FREQ</b>	Defines the frequency points for the SV vs. FREQ tables.
<b>GRP</b>	Specifies the grouping mode combination method.
<b>MMASS</b>	Specifies the missing mass response calculation.
<b>NRLSUM</b>	Specifies the Naval Research Laboratory (NRL) sum mode combination method.
<b>PFACT</b>	Calculates participation factors for the PSD or multi-point response spectrum table.
<b>PSDCOM</b>	Specifies the power spectral density mode combination method.
<b>PSDFRQ</b>	Defines the frequency points for the input spectrum vs. FREQ tables of PSD and multi-point spectrum analyses.
<b>PSDGRAPH</b>	Displays input PSD curves
<b>PSDRES</b>	Controls solution output written to the results file from a PSD analysis.
<b>PSDSPL</b>	Defines a partially correlated excitation in a PSD analysis.
<b>PSDUNIT</b>	Defines the type of PSD or multi-point response spectrum.
<b>PSDVAL</b>	Defines PSD or multi-point response spectrum values.
<b>PSDWAV</b>	Defines a wave propagation excitation in a PSD analysis.
<b>QDVAL</b>	Defines PSD quadspectral values.



**These SOLUTION commands are used to define options for spectrum analyses.**

<b>ROCK</b>	Specifies a rocking response spectrum.
<b>ROSE</b>	Specifies the Rosenblueth mode combination method.
<b>RIGRESP</b>	Selects the spectrum type and other spectrum options.
<b>SED</b>	Defines the excitation direction for a single-point response spectrum.
<b>SPDAMP</b>	Defines input spectrum damping in a multi-point response spectrum analysis.
<b>SPFREQ</b>	Defines the frequency points for the input spectrum tables <b>SPVAL</b> vs. <b>SPFREQ</b> for multi-point spectrum analyses.
<b>SPGRAPH</b>	Displays input spectrum curves for MPRS analysis.
<b>SPOPT</b>	Selects the spectrum type and other spectrum options.
<b>SPUNIT</b>	Defines the type of multi-point response spectrum.
<b>SPVAL</b>	Defines multi-point response spectrum values.
<b>SRSS</b>	Specifies the square root of sum of squares mode combination method.
<b>SV</b>	Defines spectrum values to be associated with frequency points.
<b>SVPLOT</b>	Displays input spectrum curves.
<b>SVTYP</b>	Defines the type of single-point response spectrum.
<b>VDDAM</b>	Specifies the velocity spectrum computation constants for the analysis of shock resistance of shipboard structures.

**Table 2.64 Load Step Options****These SOLUTION commands are used to define options for individual load steps.**

<b>AUTOTS</b>	Specifies whether to use automatic time stepping or load stepping.
<b>CAMPBELL</b>	<b>Campbell diagram</b> Prepares the result file for a subsequent analysis.
<b>CECMOD</b>	Modifies the constant term of a constraint equation during solution.
<b>DELTIM</b>	Specifies the time step sizes to be used for this load step.
<b>EXPSOL</b>	Specifies the solution to be expanded for reduced analyses.
<b>HMAGSOLV</b>	Specifies 2-D or axisymmetric harmonic magnetic solution options and initiates the solution.
<b>KBC</b>	Specifies stepped or ramped loading within a load step.
<b>KUSE</b>	Specifies whether or not to reuse the factorized matrix.
<b>MAGOPT</b>	Specifies options for a 3-D magnetostatic field analysis.
<b>MAGSOLV</b>	Specifies magnetic solution options and initiates the solution.
<b>MODE</b>	Specifies the harmonic loading term for this load step.
<b>NSUBST</b>	Specifies the number of substeps to be taken this load step.
<b>NUMEXP</b>	Specifies solutions to be expanded from reduced analyses.
<b>TIME</b>	Sets the time for a load step.
<b>TREF</b>	Defines the reference temperature for the thermal strain calculations.



**These SOLUTION commands are used to define options for individual load steps.**

<b>TSRES</b>	Defines an array of key times at which the time-stepping strategy changes.
<b>UPCOORD</b>	Modifies the coordinates of the active set of nodes, based on the current displacements.
<b>USRCAL</b>	Allows user-solution subroutines to be activated or deactivated.
<b>WRFULL</b>	Stops solution after assembling global matrices.

**Table 2.65 Solid Constraints**

**These SOLUTION commands are used to define constraints on the solid model.**

<b>DA</b>	Defines symmetry or antisymmetry degree-of-freedom constraints on areas.
<b>DADELE</b>	Deletes degree-of-freedom constraints on an area.
<b>DALIST</b>	Lists the degree-of-freedom constraints on an area.
<b>DK</b>	Defines degree-of-freedom constraints at keypoints.
<b>DKDELE</b>	Deletes degree-of-freedom constraints at a keypoint.
<b>DKLIST</b>	Lists the degree-of-freedom constraints at keypoints.
<b>DL</b>	Defines symmetry or antisymmetry degree-of-freedom constraints on lines.
<b>DLDELE</b>	Deletes degree-of-freedom constraints on a line.
<b>DLLIST</b>	Lists degree-of-freedom constraints on a line.
<b>DTRAN</b>	Transfers solid model degree-of-freedom constraints to the finite element model.

**Table 2.66 Solid Forces**

**These SOLUTION commands are used to define forces on the solid model.**

<b>FK</b>	Defines force loads at keypoints.
<b>FKDELE</b>	Deletes force loads at a keypoint.
<b>FKLIST</b>	Lists the forces at keypoints.
<b>FTRAN</b>	Transfers solid model forces to the finite element model.

**Table 2.67 Solid Surface Loads**

**These SOLUTION commands are used to define surface loads on the solid model.**

<b>SFA</b>	Specifies surface loads on the selected areas.
<b>SFADELE</b>	Deletes surface loads from areas.
<b>SFALIST</b>	Lists the surface loads for the specified area.
<b>SFL</b>	Specifies surface loads on lines of an area.
<b>SFLDELE</b>	Deletes surface loads from lines.
<b>SFLLIST</b>	Lists the surface loads for lines.

**These SOLUTION commands are used to define surface loads on the solid model.**

**SFTRAN** Transfer the solid model surface loads to the finite element model.

**Table 2.68 Solid Body Loads**

**These SOLUTION commands are used to define body loads on the solid model.**

<b>BFA</b>	Defines a body force load on an area.
<b>BFADELE</b>	Deletes body force loads on an area.
<b>BFALIST</b>	Lists the body force loads on an area.
<b>BFK</b>	Defines a body force load at a keypoint.
<b>BFKDELE</b>	Deletes body force loads at a keypoint.
<b>BFKLIST</b>	Lists the body force loads at keypoints.
<b>BFL</b>	Defines a body force load on a line.
<b>BFLDELE</b>	Deletes body force loads on a line.
<b>BFLLIST</b>	Lists the body force loads on a line.
<b>BFTRAN</b>	Transfers solid model body force loads to the finite element model.
<b>BFV</b>	Defines a body force load on a volume.
<b>BFVDELE</b>	Deletes body force loads on a volume.
<b>BFVLIST</b>	Lists the body force loads on a volume.

**Table 2.69 Inertia**

**These SOLUTION commands are used to define inertial loads on the model.**

<b>ACEL</b>	Specifies the linear acceleration of the structure.
<b>CGLOC</b>	Specifies the origin location of the acceleration coordinate system.
<b>CGOMGA</b>	Specifies the rotational velocity of the global origin.
<b>CMACEL</b>	Specifies the translational acceleration of an element component.
<b>CMDOMEGA</b>	Specifies the rotational acceleration of an element component about a user-defined rotational axis.
<b>CMOMEGA</b>	Specifies the rotational velocity of an element component about a user-defined rotational axis.
<b>CMROTATE</b>	Specifies the rotational velocity of an element component about a user-defined rotational axis.
<b>CORIOLIS</b>	Applies the Coriolis effect to a rotating structure.
<b>DCGOMG</b>	Specifies the rotational acceleration of the global origin.
<b>DOMEGA</b>	Specifies the rotational acceleration of the structure.
<b>IRLF</b>	Specifies that inertia relief calculations are to be performed.
<b>OMEGA</b>	Specifies the rotational velocity of the structure.

**These SOLUTION commands are used to define inertial loads on the model.**

**SYNCHRO** Specifies whether the excitation frequency is synchronous or asynchronous with the rotational velocity of the structure.

**Table 2.70 Miscellaneous Loads**

**These SOLUTION commands are for miscellaneous load definition and control.**

<b>BIOT</b>	Calculates the Biot-Savart source magnetic field intensity.
<b>FMAGBC</b>	Applies force and torque boundary conditions to an element component.
<b>HFPOR</b>	Specifies input data for waveguide or transmission line ports or an incident plane wave.
<b>IC</b>	Specifies initial conditions at nodes.
<b>ICDELE</b>	Deletes initial conditions at nodes.
<b>ICE</b>	Specifies initial conditions on elements.
<b>ICEDELE</b>	Deletes initial conditions on elements.
<b>ICELIST</b>	Lists initial conditions on elements.
<b>ICLIST</b>	Lists the initial conditions.
<b>INISTATE</b>	Applies initial state data to an element or a selection of elements.
<b>MPCHG</b>	Changes the material number attribute of an element.
<b>OUTPR</b>	Controls the solution printout.
<b>OUTRES</b>	Controls the solution data written to the database.
<b>PGRAPH</b>	Specifies the location from which graphics data will be retrieved for viewing.
<b>PGSAVE</b>	Creates a PowerGraphics (PGR) file from results data.
<b>PGWRITE</b>	Writes selected solution data to the PGR file for faster postprocessing access.
<b>PLWAVE</b>	Specifies a free-space time-harmonic incident plane electromagnetic wave in the global Cartesian coordinate system.
<b>RESCONTROL</b>	Controls file writing for multiframe restarts.
<b>SBCLIST</b>	Lists solid model boundary conditions.
<b>SBCTRAN</b>	Transfers solid model loads and boundary conditions to the FE model.
<b>WSPRINGS</b>	Creates weak springs on corner nodes of a bounding box of the currently selected elements.

**Table 2.71 Load Step Operations**

**These SOLUTION commands are used to write and solve multiple load steps.**

<b>LSCLEAR</b>	Clears loads and load step options from the database.
<b>LSDELE</b>	Deletes load step files.
<b>LSREAD</b>	Reads load and load step option data into the database.
<b>LSSOLVE</b>	Reads and solves multiple load steps.

**These SOLUTION commands are used to write and solve multiple load steps.**

**LSWRITE** Writes load and load step option data to a file.

### Table 2.72 Master Degrees of Freedom

**These SOLUTION commands are used to define master degrees of freedom.**

**M** Defines master degrees of freedom for reduced and superelement generation analyses.

**MDELE** Deletes master degrees of freedom.

**MGEN** Generates additional master degrees of freedom from a previously defined set.

**MLIST** Lists the master degrees of freedom.

**TOTAL** Specifies automatic master degree-of-freedom generation.

### Table 2.73 Gap Conditions

**These SOLUTION commands are used to define gaps for transient dynamic analyses.**

**GP** Defines a gap condition for transient analyses.

**GPDELE** Deletes gap conditions.

**GPLIST** Lists the gap conditions.

### Table 2.74 Rezoning

**These SOLUTION commands are used in analyses that employ rezoning.**

**REZONE** Initiates a rezoning operation, sets rezoning options, and rebuilds the database.

**MAPSOLVE** Maps the solved node and element solutions from an original mesh to a new mesh.

**REMESH** Specifies the starting and ending remeshing points.

**AREMESH** Generates an area for creating a new mesh for rezoning.

### Table 2.75 Birth and Death

**These SOLUTION commands are used for the birth and death option for elements.**

**EALIVE** Reactivates an element (for the birth and death capability).

**EKILL** Deactivates an element (for the birth and death capability).

**ESTIF** Specifies the matrix multiplier for deactivated elements.

### Table 2.76 FE Constraints

**These SOLUTION commands are used to define constraints on the finite element model.**

**D** Defines degree-of-freedom constraints at nodes.

**DCUM** Specifies that degree-of-freedom constraint values are to be accumulated.

**DDELE** Deletes degree-of-freedom constraints.

**DJ** Specify boundary conditions on the components of relative motion of a joint element.

**These SOLUTION commands are used to define constraints on the finite element model.**

<b>DJDELE</b>	Deletes boundary conditions on the components of relative motion of a joint element.
<b>DJLIST</b>	Lists boundary conditions applied to joint elements.
<b>DLIST</b>	Lists degree-of-freedom constraints.
<b>DSCALE</b>	Scales degree-of-freedom constraint values.
<b>DSYM</b>	Specifies symmetry or antisymmetry degree-of-freedom constraints on nodes.
<b>DVAL</b>	Defines values at enforced group.
<b>GSBDATA</b>	Specifies the constraints or applies the load at the ending point for the generalized plane strain element option.
<b>GSLIST</b>	When using generalized plane strain, lists the input data or solutions.
<b>LDREAD</b>	Reads results from the results file and applies them as loads.

**Table 2.77 FE Forces****These SOLUTION commands are used to define nodal loads on the finite element model.**

<b>F</b>	Specifies force loads at nodes.
<b>FCUM</b>	Specifies that force loads are to be accumulated.
<b>FDELE</b>	Deletes force loads on nodes.
<b>FJ</b>	Specify forces or moments on the components of the relative motion of a joint element.
<b>FJDELE</b>	Deletes forces (or moments) on the components of the relative motion of a joint element.
<b>FJLIST</b>	Lists forces and moments applied on joint elements.
<b>FLIST</b>	Lists force loads on the nodes.
<b>FSCALE</b>	Scales force load values in the database.

**Table 2.78 FE Surface Loads****These SOLUTION commands are used to define surface loads on the finite element model.**

<b>SF</b>	Specifies surface loads on nodes.
<b>SFBEAM</b>	Specifies surface loads on beam elements.
<b>SFCUM</b>	Specifies that surface loads are to be accumulated.
<b>SFDELE</b>	Deletes surface loads.
<b>SFE</b>	Specifies surface loads on elements.
<b>SFDELE</b>	Deletes surface loads from elements.
<b>SFELIST</b>	Lists the surface loads for elements.
<b>SFFUN</b>	Specifies a varying surface load.
<b>SFGRAD</b>	Specifies a gradient (slope) for surface loads.
<b>SFLIST</b>	Lists surface loads.

**These SOLUTION commands are used to define surface loads on the finite element model.**

**SFSCALE** Scales surface loads on elements.

**Table 2.79 FE Body Loads**

**These SOLUTION commands are used to define body loads on the finite element model.**

<b>BF</b>	Defines a nodal body force load.
<b>BFCUM</b>	Specifies that nodal body force loads are to be accumulated.
<b>BFDELE</b>	Deletes nodal body force loads.
<b>BFE</b>	Defines an element body force load.
<b>BFECUM</b>	Specifies whether to ignore subsequent element body force loads.
<b>BFEDELE</b>	Deletes element body force loads.
<b>BFELIST</b>	Lists the element body force loads.
<b>BFESCAL</b>	Scales element body force loads.
<b>BFLIST</b>	Lists the body force loads on nodes.
<b>BFSCALE</b>	Scales body force loads at nodes.
<b>BFUNIF</b>	Assigns a uniform body force load to all nodes.
<b>LDREAD</b>	Reads results from the results file and applies them as loads.
<b>RIMPORT</b>	Imports initial stresses from an explicit run into ANSYS.
<b>TUNIF</b>	Assigns a uniform temperature to all nodes.

**Table 2.80 Status**

**These SOLUTION commands are for use with the STAT command.**

<b>ATYPE</b>	Specifies "Analysis types" as the subsequent status topic.
<b>BIOOPT</b>	Specifies "Biot-Savart options" as the subsequent status topic.
<b>DEACT</b>	Specifies "Element birth and death" as the subsequent status topic.
<b>DYNOPT</b>	Specifies "Dynamic analysis options" as the subsequent status topic.
<b>GAP</b>	Specifies "Reduced transient gap conditions" as the subsequent status topic.
<b>GENOPT</b>	Specifies "General options" as the subsequent status topic.
<b>INRTIA</b>	Specifies "Inertial loads" as the subsequent status topic.
<b>LSOPER</b>	Specifies "Load step operations" as the subsequent status topic.
<b>MASTER</b>	Specifies "Master DOF" as the subsequent status topic.
<b>NLOPT</b>	Specifies "Nonlinear analysis options" as the subsequent status topic.
<b>OUTOPT</b>	Specifies "Output options" as the subsequent status topic.
<b>SMBODY</b>	Specifies "Body loads on the solid model" as the subsequent status topic.
<b>SMCONS</b>	Specifies "Constraints on the solid model" as the subsequent status topic.

**These SOLUTION commands are for use with the STAT command.**

<b>SMFOR</b>	Specifies "Forces on the solid model" as the subsequent status topic.
<b>SMSURF</b>	Specifies "Surface loads on the solid model" as the subsequent status topic.
<b>SOLUOPT</b>	Specifies "Solution options" as the subsequent status topic.
<b>SPTOPT</b>	Specifies "Spectrum analysis options" as the subsequent status topic.

**Table 2.81 Explicit Dynamics****These SOLUTION commands are used for an explicit dynamic analysis.**

<b>EDADAPT</b>	Activates adaptive meshing in an explicit dynamic analysis.
<b>EDALE</b>	Assigns mesh smoothing to explicit dynamic elements that use the ALE formulation.
<b>EDBVIS</b>	Specifies global bulk viscosity coefficients for an explicit dynamic analysis.
<b>EDCADAPT</b>	Specifies adaptive meshing controls for an explicit dynamic analysis.
<b>EDCPU</b>	Specifies CPU time limit for an explicit dynamic analysis.
<b>EDCSC</b>	Specifies whether to use subcycling in an explicit dynamic analysis.
<b>EDCTS</b>	Specifies mass scaling and scale factor of computed time step for an explicit dynamic analysis.
<b>EDDAMP</b>	Defines mass weighted (Alpha) or stiffness weighted (Beta) damping for an explicit dynamics model.
<b>EDDRELAX</b>	Activates initialization to a prescribed geometry or dynamic relaxation for the explicit analysis.
<b>EDDUMP</b>	Specifies output frequency for the explicit dynamic restart file (d3dump).
<b>EDENERGY</b>	Specifies energy dissipation controls for an explicit dynamic analysis.
<b>EDFPLOT</b>	Allows plotting of explicit dynamics forces.
<b>EDGCALE</b>	Defines global ALE controls for an explicit dynamic analysis.
<b>EDHGLS</b>	Specifies the hourglass coefficient for an explicit dynamic analysis.
<b>EDHIST</b>	Specifies time-history output for an explicit dynamic analysis.
<b>EDHTIME</b>	Specifies the time-history output interval for an explicit dynamic analysis.
<b>EDINT</b>	Specifies number of integration points for explicit shell and beam output.
<b>EDIS</b>	Specifies stress initialization in an explicit dynamic full restart analysis.
<b>EDLOAD</b>	Specifies loads for an explicit dynamic analysis.
<b>EDOPT</b>	Specifies the type of output for an explicit dynamic analysis.
<b>EDOUT</b>	Specifies time-history output (ASCII format) for an explicit dynamic analysis.
<b>EDPL</b>	Plots a time dependent load curve in an explicit dynamic analysis.

**These SOLUTION commands are used for an explicit dynamic analysis.**

<b>EDPVEL</b>	Applies initial velocities to parts or part assemblies in an explicit dynamic analysis.
<b>EDRC</b>	Specifies rigid/deformable switch controls in an explicit dynamic analysis.
<b>EDRD</b>	Switches a part from deformable to rigid or from rigid to deformable in an explicit dynamic analysis.
<b>EDRI</b>	Defines inertia properties for a new rigid body that is created when a deformable part is switched to rigid in an explicit dynamic analysis.
<b>EDRST</b>	Specifies the output interval for an explicit dynamic analysis.
<b>EDRUN</b>	Specifies serial or parallel processing for an explicit dynamic analysis.
<b>EDSHELL</b>	Specifies shell computation controls for an explicit dynamic analysis.
<b>EDSOLV</b>	Specifies "explicit dynamics solution" as the subsequent status topic.
<b>EDSTART</b>	Specifies status (new or restart) of an explicit dynamic analysis.
<b>EDTERM</b>	Specifies termination criteria for an explicit dynamic analysis.
<b>EDTP</b>	Plots explicit elements based on their time step size.
<b>EDVEL</b>	Applies initial velocities to nodes or node components in an explicit dynamic analysis.
<b>EDWRITE</b>	Writes explicit dynamics input to an LS-DYNA input file.
<b>REXPOR</b>	Exports displacements from an implicit run to ANSYS LS-DYNA.

**Table 2.82 FLOTRAN Checkout****These SOLUTION commands are used for a FLOTRAN analysis.**

<b>FLOCHECK</b>	Sets up and runs a zero-iteration FLOTRAN analysis.
-----------------	---

**Table 2.83 Radiosity****These SOLUTION commands are used to calculate the radiation view factors and to specify the solution parameters for the Radiosity solver method.**

<b>HEMIOPT</b>	Specifies options for Hemicube view factor calculation.
<b>RADOPT</b>	Specifies Gauss-Seidel Radiosity Solver options.
<b>RDEC</b>	Defines the decimation parameters.
<b>RSOPT</b>	Creates or loads the radiosity mapping data file for <a href="#">SURF251</a> or <a href="#">SURF252</a> element types.
<b>RSURF</b>	Generates the radiosity surface elements and stores them in the database.
<b>RSYMM</b>	Defines the plane of symmetry or center of rotation for the radiosity method.
<b>SPCNOD</b>	Defines a space node for radiation using the Radiosity method.
<b>SPCTEMP</b>	Defines a free-space ambient temperature for radiation using the Radiosity method.
<b>STEF</b>	Specifies Stefan-Boltzmann radiation constant.



These **SOLUTION** commands are used to calculate the radiation view factors and to specify the solution parameters for the Radiosity solver method.

<b>V2DOPT</b>	Specifies 2-D/axisymmetric view factor calculation options.
<b>VFOPT</b>	Specifies options for view factor file.
<b>QSOPT</b>	Specifies quasi static radiation options.

**Table 2.84 ANSYS Multi-field solver Definition Commands**

These **SOLUTION** commands are used to define the fields for an ANSYS Multi-field solver analysis.

<b>MFCMMAND</b>	Captures field solution options in a command file.
<b>MFELEM</b>	Defines a field by grouping element types.
<b>MFEM</b>	Adds more element types to a previously defined field number.
<b>MFEXTER</b>	Defines external fields for an ANSYS Multi-field solver analysis.
<b>MFFNAME</b>	Specifies a file name for a field in an ANSYS Multi-field solver analysis.

**Table 2.85 ANSYS Multi-field solver Global Controls**

These **SOLUTION** commands set global controls for an ANSYS Multi-field solver analysis.

<b>MFANALYSIS</b>	Turns an ANSYS Multi-field solver analysis on or off.
<b>MFCLEAR</b>	Deletes ANSYS Multi-field solver analysis settings.
<b>MFFR</b>	Sets up Multi-Field relaxation factors for field solutions.
<b>MFINTER</b>	Specifies the interface load transfer interpolation option for an ANSYS Multi-field solver analysis.
<b>MFLIST</b>	Lists the settings for an ANSYS Multi-field solver analysis.
<b>MFORDER</b>	Specifies field solution order for an ANSYS Multi-field solver analysis.
<b>MFPSIMUL</b>	Sets up a field solver group to simultaneously process with code coupling analyses.
<b>MFSORDER</b>	Sets up the solution sequence of simultaneous field solver groups for code coupling analyses.
<b>MFWRITE</b>	Writes an ANSYS master input file for MFX multiple code coupling.

**Table 2.86 ANSYS Multi-field solver Time Controls**

These **SOLUTION** commands set time controls for an ANSYS Multi-field solver analysis.

<b>MFCALC</b>	Specifies a calculation frequency for a field in an ANSYS Multi-field solver analysis.
<b>MFDTIME</b>	Sets time step increment for an ANSYS Multi-field solver analysis.
<b>MFOUTPUT</b>	Specifies results file output frequency for an ANSYS Multi-field solver analysis.
<b>MFRSTART</b>	Specifies a restart time for an ANSYS Multi-field solver analysis.

**These SOLUTION commands set time controls for an ANSYS Multi-field solver analysis.**

**MFTIME** Sets end time for an ANSYS Multi-field solver analysis.

### Table 2.87 ANSYS Multi-field solver Load Transfer

**These SOLUTION commands are used to define load transfer for an ANSYS Multi-field solver analysis.**

**MFLCOMM** Defines a load transfer for code coupling analyses.

**MFSURFACE** Defines a surface load transfer for an ANSYS Multi-field solver analysis.

**MFVOLUME** Defines volume load transfer across interface for an ANSYS Multi-field solver analysis.

### Table 2.88 ANSYS Multi-field solver Convergence Controls

**These SOLUTION commands are used to define convergence controls for an ANSYS Multi-field solver analysis.**

**MFCONV** Sets convergence values for an ANSYS Multi-field solver analysis.

**MFITER** Sets the maximum number of stagger iterations for an ANSYS Multi-field solver analysis.

**MFRELAX** Sets relaxation values for an ANSYS Multi-field solver analysis.

### Table 2.89 ANSYS Multi-field solver Interface Mapping

**These SOLUTION commands are used to define mapping details for an ANSYS Multi-field solver analysis.**

**MFBUCKET** Turns a bucket search on or off.

**MFCI** Sets the pixel resolution used by the CPP interpolation scheme.

**MFMAP** Calculates, saves, resumes, or deletes mapping data in an ANSYS Multi-field solver analysis.

**MFTOL** Turns normal distance checking on for surface mapping in an ANSYS Multi-field solver analysis.

## 2.7. POST1 Commands

These commands are used to postprocess the results with the database processor. The commands are grouped by functionality.

### Table 2.90 Setup

**These POST1 commands are used to put data into the database for postprocessing.**

**APPEND** Reads data from the results file and appends it to the database.

**DESOL** Defines or modifies solution results at a node of an element.

**DNSOL** Defines or modifies solution results at a node.

**FILE** Specifies the data file where results are to be found.

**HRCPLX** Computes and stores in the database the time-harmonic solution at a prescribed phase angle.

**These POST1 commands are used to put data into the database for postprocessing.**

<b>PGRSET</b>	Defines the data set to be read from the PGR file.
<b>/POST1</b>	Enters the database results postprocessor.
<b>RESET</b>	Resets all POST1 or POST26 specifications to initial defaults.
<b>SET</b>	Defines the data set to be read from the results file.
<b>SUBSET</b>	Reads results for the selected portions of the model.

**Table 2.91 Controls**

**These POST1 commands are used to control the calculations done for other commands.**

<b>AVPRIN</b>	Specifies how principal and vector sums are to be calculated.
<b>AVRES</b>	Specifies how results data will be averaged when PowerGraphics is enabled.
<b>/EFACET</b>	Specifies the number of facets per element edge for PowerGraphics displays.
<b>ERNORM</b>	Controls error estimation calculations.
<b>FORCE</b>	Selects the element nodal force type for output.
<b>INRES</b>	Identifies the data to be retrieved from the results file.
<b>LAYER</b>	Specifies the element layer for which data are to be processed.
<b>RSYS</b>	Activates a coordinate system for printout or display of results.
<b>SHELL</b>	Selects a shell element or shell layer location for results output.

**Table 2.92 Results**

**These POST1 commands are used to process results, such as degree-of-freedom results, nodal stresses, and element summable and nonsummable miscellaneous data.**

<b>NSORT</b>	Sorts nodal data.
<b>NUSORT</b>	Restores original order for nodal data.
<b>PLCINT</b>	Plots the contour integral ( <b>CINT</b> ) result data.
<b>PLDISP</b>	Displays the displaced structure.
<b>PLESOL</b>	Displays the solution results as discontinuous element contours.
<b>PLNSOL</b>	Displays results as continuous contours.
<b>PLORB</b>	Displays the orbital motion of a rotating structure undergoing vibration.
<b>PREENERGY</b>	Prints the total energies of a model.
<b>PRORB</b>	Prints the orbital motion characteristics of a rotating structure undergoing vibration.
<b>PLVECT</b>	Displays results as vectors.
<b>PRCINT</b>	Lists the contour integral ( <b>CINT</b> ) result data.
<b>PRESOL</b>	Prints the solution results for elements.
<b>PRJSOL</b>	Print joint element output.
<b>PRNLD</b>	Prints the summed element nodal loads.
<b>PRNSOL</b>	Prints the nodal solution results.

**These POST1 commands are used to process results, such as degree-of-freedom results, nodal stresses, and element summable and nonsummable miscellaneous data.**

<b>PRRFOR</b>	Used with the <b>FORCE</b> command. Prints the constrained node reaction solution.
<b>PRRSOL</b>	Prints the constrained node reaction solution.
<b>PRVECT</b>	Prints results as vector magnitude and direction cosines.
<b>SUMTYPE</b>	Sets the type of summation to be used in the following load case operations.

### Table 2.93 Element Table

**These POST1 commands are used to operate with the element table, which in turn is used to process results stored for each element, such as average stresses, heat fluxes, etc.**

<b>DETAB</b>	Modifies element table results in the database.
<b>ESORT</b>	Sorts the element table.
<b>ETABLE</b>	Fills a table of element values for further processing.
<b>EUSORT</b>	Restores original order of the element table.
<b>PLETAB</b>	Displays element table items.
<b>PLLS</b>	Displays element table items as contoured areas along elements.
<b>PLVECT</b>	Displays results as vectors.
<b>PRETAB</b>	Prints the element table items.
<b>PRVECT</b>	Prints results as vector magnitude and direction cosines.
<b>SABS</b>	Specifies absolute values for element table operations.
<b>SADD</b>	Forms an element table item by adding two existing items.
<b>SALLOW</b>	Defines the allowable stress table for safety factor calculations.
<b>SEXP</b>	Forms an element table item by exponentiating and multiplying.
<b>SFACT</b>	Allows safety factor or margin of safety calculations to be made.
<b>SFCALC</b>	Calculates the safety factor or margin of safety.
<b>SMAX</b>	Forms an element table item from the maximum of two other items.
<b>SMIN</b>	Forms an element table item from the minimum of two other items.
<b>SMULT</b>	Forms an element table item by multiplying two other items.
<b>SSUM</b>	Calculates and prints the sum of element table items.
<b>TALLOW</b>	Defines the temperature table for safety factor calculations.
<b>VCROSS</b>	Forms element table items from the cross product of two vectors.
<b>VDOT</b>	Forms an element table item from the dot product of two vectors.

### Table 2.94 Listing

**These POST1 commands are used to control printed listings of results.**

<b>/FORMAT</b>	Specifies format controls for tables.
<b>/HEADER</b>	Sets page and table heading print controls.
<b>IRLIST</b>	Prints inertia relief summary table.

**These POST1 commands are used to control printed listings of results.**

<b>/PAGE</b>	Defines the printout and screen page size.
<b>PRERR</b>	Prints SEPC and TEPC.
<b>PRITER</b>	Prints solution summary data.

**Table 2.95 Animation**

**These POST1 commands are used to animate results.**

<b>ANCNTR</b>	Produces an animated sequence of a contoured deformed shape.
<b>ANCUT</b>	Produces an animated sequence of Q-slices.
<b>ANDATA</b>	Produces a sequential contour animation over a range of results data.
<b>ANDSCL</b>	Produces an animated sequence of a deformed shape.
<b>ANCYC</b>	Applies a traveling wave animation to graphics data in a modal cyclic symmetry analysis.
<b>ANDYNA</b>	Produces an animated sequence of contour values through sub-steps.
<b>/ANFILE</b>	Saves or resumes an animation sequence to or from a file.
<b>ANFLOW</b>	Produces an animated sequence of particle flow in a flowing fluid or a charged particle traveling in an electric or magnetic field.
<b>ANHARM</b>	Produces a time-transient animated sequence of time-harmonic results ( <b>ANTYPE,HARMIC</b> ) or complex mode shapes ( <b>ANTYPE,MOD-AL</b> ).
<b>ANIM</b>	Displays graphics data in animated form.
<b>ANISOS</b>	Produces an animated sequence of an isosurface.
<b>ANMODE</b>	Produces an animated sequence of a mode shape.
<b>ANMRES</b>	Performs animation of results over multiple results files in an explicit dynamic structural analysis or fluid flow analysis with remeshing.
<b>ANTIME</b>	Produces a sequential contour animation over a range of time.
<b>TRTIME</b>	Defines the options used for the <b>PLTRAC</b> (particle flow or charged particle trace) command.

**Table 2.96 Path Operations**

**These POST1 commands are used for path operations.**

<b>PADELE</b>	Deletes a defined path.
<b>PAGET</b>	Writes current path information into an array variable.
<b>PAPUT</b>	Retrieves path information from an array variable.
<b>PARESU</b>	Restores previously saved paths from a file.
<b>PASAVE</b>	Saves selected paths to an external file.
<b>PATH</b>	Defines a path name and establishes parameters for the path.
<b>PCALC</b>	Forms additional labeled path items by operating on existing path items.
<b>PCROSS</b>	Calculates the cross product of two path vectors along the current path.

**These POST1 commands are used for path operations.**

<b>PDEF</b>	Interpolates an item onto a path.
<b>PDOT</b>	Calculates the dot product of two path vectors along the current path.
<b>PLPAGM</b>	Displays path items along the path geometry.
<b>PLPATH</b>	Displays path items on a graph.
<b>PLSECT</b>	Displays membrane and membrane-plus-bending linearized stresses.
<b>PMPAP</b>	Creates mapping of the path geometry by defining path interpolation division points.
<b>PPATH</b>	Defines a path by picking or defining nodes, or locations on the currently active working plane, or by entering specific coordinate locations.
<b>PRANGE</b>	Determines the path range.
<b>PRPATH</b>	Prints path items along a geometry path.
<b>PRSECT</b>	Calculates and prints linearized stresses along a section path.
<b>PSEL</b>	Selects a path or paths.
<b>PVECT</b>	Interpolates a set of items onto a path.

**Table 2.97 Surface Operations****These POST1 commands are used to define an arbitrary surface and to develop results information for that surface.**

<b>SUCALC</b>	Create new result data by operating on two existing result datasets on a given surface.
<b>SUCR</b>	Create a surface.
<b>SUDEL</b>	Delete geometry information as well as any mapped results for specified surface or for all selected surfaces.
<b>SUEVAL</b>	Perform operations on a mapped item and store result in a scalar parameter.
<b>SUGET</b>	Create and dimension an NPT row array parameter named PARM, where NPT is the number of geometry points in SurfName.
<b>SUMAP</b>	Map results onto selected surface(s).
<b>SUPL</b>	Plot specified SetName result data on all selected surfaces or on the specified surface.
<b>SUPR</b>	Print surface information.
<b>SURESU</b>	Resume surface definitions from a specified file.
<b>SUSAVE</b>	Save surface definitions and result items to a file.
<b>SUSEL</b>	Selects a subset of surfaces
<b>SUVECT</b>	Operate between two mapped result vectors.

**Table 2.98 Load Case Calculations****These POST1 commands are used for combining results from different load steps.**

<b>LCABS</b>	Specifies absolute values for load case operations.
<b>LCASE</b>	Reads a load case into the database.

**These POST1 commands are used for combining results from different load steps.**

<b>LCDEF</b>	Creates a load case from a set of results on a results file.
<b>LCFACT</b>	Defines scale factors for load case operations.
<b>LCFILE</b>	Creates a load case from an existing load case file.
<b>LCOPER</b>	Performs load case operations.
<b>LCSEL</b>	Selects a subset of load cases.
<b>LCSUM</b>	Specifies whether to process nonsummable items in load case operations.
<b>LCWRITE</b>	Creates a load case by writing results to a load case file.
<b>LCZERO</b>	Zeroes the results portion of the database.
<b>RAPPND</b>	Appends results data from the database to the results file.

**Table 2.99 Magnetics Calculations**

**These POST1 commands are used for special purpose magnetics postprocessing.**

<b>CURR2D</b>	Calculates current flow in a 2-D conductor.
<b>EMAGERR</b>	Calculates the relative error in an electrostatic or electromagnetic field analysis.
<b>EMF</b>	Calculates the electromotive force (emf), or voltage drop along a predefined path.
<b>EMFT</b>	Summarizes electromagnetic forces and torques on a selected set of nodes.
<b>FLUXV</b>	Calculates the flux passing through a closed contour.
<b>FMAGSUM</b>	Summarizes electromagnetic force calculations on element components.
<b>FOR2D</b>	Calculates magnetic forces on a body.
<b>IMPD</b>	Calculates the impedance of a conductor at a reference plane.
<b>MMF</b>	Calculates the magnetomotive force along a path.
<b>PLF2D</b>	Generates a contour line plot of equipotentials.
<b>POWERH</b>	Calculates the rms power loss in a conductor or lossy dielectric.
<b>QFACT</b>	Calculates the quality factor for high-frequency electromagnetic resonators.
<b>SENERGY</b>	Determines the stored magnetic energy or co-energy.
<b>SPARM</b>	Calculates scattering (S) parameters between ports of a waveguide.
<b>TORQ2D</b>	Calculates torque on a body in a magnetic field.
<b>TORQC2D</b>	Calculates torque on a body in a magnetic field based on a circular path.
<b>TORQSUM</b>	Summarizes electromagnetic torque calculations on element components.

**Table 2.100 Fatigue**

**These POST1 commands are used for fatigue analyses.**

<b>FE</b>	Defines a set of fatigue event parameters.
-----------	--

**These POST1 commands are used for fatigue analyses.**

<b>FELIST</b>	Lists the fatigue event parameters.
<b>FL</b>	Defines a set of fatigue location parameters.
<b>FLLIST</b>	Lists the fatigue location parameters.
<b>FP</b>	Defines the fatigue S vs. N and Sm vs. T tables.
<b>FPLIST</b>	Lists the property table stored for fatigue evaluation.
<b>FS</b>	Stores fatigue stress components at a node.
<b>FSDELE</b>	Deletes a stress condition for a fatigue location, event, and loading.
<b>FSLIST</b>	Lists the stresses stored for fatigue evaluation.
<b>FSNODE</b>	Calculates and stores the stress components at a node for fatigue.
<b>FSPLOT</b>	Displays a fatigue stress item for a fatigue location and event.
<b>FSSECT</b>	Calculates and stores total linearized stress components.
<b>FTCALC</b>	Performs fatigue calculations for a particular node location.
<b>FTSIZE</b>	Defines the fatigue data storage array.
<b>FTWRITE</b>	Writes all currently stored fatigue data on a file.

**Table 2.101 Trace Points****These POST1 commands are used to trace particle motions in a flow stream.**

<b>PLTRAC</b>	Displays a particle flow or charged particle trace on an element display.
<b>TRPDEL</b>	Deletes particle flow or charged particle trace points.
<b>TRPLIS</b>	Lists the particle flow or charged particle trace points.
<b>TRPOIN</b>	Defines a point through which a particle flow or charged particle trace will travel.

**Table 2.102 FLOTRAN Processing****These POST1 commands are used to postprocess the FLOTRAN CFD results.**

<b>FLREAD</b>	Reads the residual file written by the FLOTRAN CFD option.
---------------	--

**Table 2.103 Special Purpose****These POST1 commands are used for various special purposes.**

<b>BFINT</b>	Activates the body force interpolation operation.
<b>CBDOF</b>	Activates cut boundary interpolation (for submodeling).
<b>CMSFILE</b>	Specifies the <a href="#">component mode synthesis</a> (CMS) results files to include when plotting the mode shape of an assembly.
<b>/CYCEXPAND</b>	Graphically expands displacements, stresses and strains of a cyclically symmetric model partially or through the full 360 degrees by combining the <i>real</i> (original nodes and elements) and <i>imaginary</i> (duplicate nodes and elements) parts of the solution.
<b>CYCPHASE</b>	Provides tools for determining minimum and maximum possible result values from frequency couplets produced in a modal cyclic symmetry analysis.



**These POST1 commands are used for various special purposes.**

<b>/EXPAND</b>	Allows the creation of a larger graphic display than that which is represented by the actual finite element analysis model.
<b>EXPAND</b>	Displays the results of a modal cyclic symmetry analysis.
<b>EXPROFILE</b>	Exports ANSYS interface loads to a CFX Profile file.
<b>EXUNIT</b>	Indicates units assumed for an interface load for ANSYS to CFX transfer.
<b>FSSPARM</b>	Calculates reflection and transmission properties of a frequency selective surface.
<b>FSUM</b>	Sums the nodal force and moment contributions of elements.
<b>HFANG</b>	Defines or displays spatial angles of a spherical radiation surface for antenna parameter calculations.
<b>HFARRAY</b>	Defines phased array antenna characteristics.
<b>HFDEEM</b>	Calibrates S-parameter phase shift.
<b>HFPOWER</b>	Calculates power terms of a multi-port network.
<b>HFSYM</b>	Sets symmetry planes for the computation of high-frequency electromagnetic fields in the near and far field domains (beyond the finite element region).
<b>INTSRF</b>	Integrates nodal results on an exterior surface.
<b>KCALC</b>	Calculates stress intensity factors in fracture mechanics analyses.
<b>NFORCE</b>	Sums the nodal forces and moments of elements attached to nodes.
<b>NLDPOST</b>	Gets element component information from <a href="#">nonlinear diagnostic</a> files.
<b>PLCAMP</b>	Plots a Campbell diagram for applications involving rotating structure dynamics.
<b>PLCRACK</b>	Displays cracking and crushing locations in SOLID65 elements.
<b>PLFAR</b>	Plots electric far fields and far field parameters.
<b>PLFSS</b>	Plots reflection and transmission parameters of a frequency selective surface solution.
<b>PLNEAR</b>	Plots the electric field in the near zone exterior to the equivalent source surface.
<b>PLSCH</b>	Converts and plots scattering, admittance, or impedance parameters on a Smith chart.
<b>PLSYZ</b>	Converts and plots scattering, admittance, or impedance parameters as a function of frequency.
<b>PLTD</b>	Displays TDR/TDT waveforms, an impedance profile, or a total waveform.
<b>PLTLINE</b>	Plots port transmission line data generated by the HFPCSWP or SPSWP macros.
<b>PRCAMP</b>	Prints a Campbell diagram for applications involving rotating structure dynamics.
<b>PRFAR</b>	Prints electric far fields and far field parameters.
<b>PRNEAR</b>	Prints the electric field in the near zone exterior to the equivalent source surface.

**These POST1 commands are used for various special purposes.**

<b>PRSYZ</b>	Converts and lists scattering, admittance, or impedance parameters.
<b>RESWRITE</b>	Appends results data from the database to a results file in a <a href="#">top-down substructured</a> analysis.
<b>RMFLVEC</b>	Writes eigenvectors of fluid nodes to a file for use in damping parameter extraction.
<b>RSPLIT</b>	Creates one or more results file(s) from the current results file based on subsets of elements.
<b>SPICE</b>	Generates a SPICE subcircuit model using S-parameters from a Touchstone file.
<b>SPMWRITE</b>	Calculates state-space matrices and writes them to the SPM file.
<b>SPOINT</b>	Defines a point for moment summations.

**Table 2.104 Status****These POST1 commands are for use with the STAT command.**

<b>CALC</b>	Specifies "Calculation settings" as the subsequent status topic.
<b>DATADF</b>	Specifies "Directly defined data status" as the subsequent status topic.
<b>DEFINE</b>	Specifies "Data definition settings" as the subsequent status topic.
<b>DISPLAY</b>	Specifies "Display settings" as the subsequent status topic.
<b>LCCALC</b>	Specifies "Load case settings" as the subsequent status topic.
<b>POINT</b>	Specifies "Point flow tracing settings" as the subsequent status topic.
<b>PRINT</b>	Specifies "Print settings" as the subsequent status topic.
<b>SORT</b>	Specifies "Sort settings" as the subsequent status topic.
<b>SPEC</b>	Specifies "Miscellaneous specifications" as the subsequent status topic.

**Table 2.105 Failure Criteria****These POST1 commands are for use with the failure criteria.**

<b>FC</b>	Provides failure criteria information and activates a data table to input temperature-dependent stress and strain limits.
<b>FCHECK</b>	Checks both the strain and stress input criteria for all materials.
<b>FCDELE</b>	Deletes previously defined failure criterion data for the given material.
<b>FCLIST</b>	Lists the failure criteria that has been input.
<b>FCTYP</b>	Activates or removes failure-criteria types for postprocessing.

## 2.8. POST26 Commands

These commands are used to postprocess the results with the time-history processor. The commands are grouped by functionality.

**Table 2.106 Set Up**

**These POST26 commands are used to store data for processing.**

<b>ANSOL</b>	Specifies averaged nodal data to be stored from the results file in the solution coordinate system.
<b>CISOL</b>	Stores J-integral information in a variable.
<b>DATA</b>	Reads data records from a file into a variable.
<b>EDREAD</b>	Reads explicit dynamics output into variables for time-history postprocessing.
<b>ENERSOL</b>	Specifies the total energies to be stored.
<b>ESOL</b>	Specifies element data to be stored from the results file.
<b>FILE</b>	Specifies the data file where results are to be found.
<b>GAPF</b>	Defines the gap force data to be stored in a variable.
<b>GSSOL</b>	Specifies which results to store from the results file when using generalized plane strain.
<b>JSOL</b>	Specifies result items to be stored for the joint element.
<b>NSOL</b>	Specifies nodal data to be stored from the results file.
<b>NSTORE</b>	Defines which time points are to be stored.
<b>NUMVAR</b>	Specifies the number of variables allowed in POST26.
<b>/POST26</b>	Enters the time-history results postprocessor.
<b>RESET</b>	Resets all POST1 or POST26 specifications to initial defaults.
<b>RFORCE</b>	Specifies the total reaction force data to be stored.
<b>/RGB</b>	Specifies the RGB color values for indices and contours.
<b>SOLU</b>	Specifies solution summary data per substep to be stored.
<b>STORE</b>	Stores data in the database for the defined variables.
<b>TIMERANGE</b>	Specifies the time range for which data are to be stored.
<b>VARDEL</b>	Deletes a variable (GUI).
<b>VARNAM</b>	Names (or renames) a variable.

**Table 2.107 Controls**

**These POST26 commands are used to control the calculations of other commands.**

<b>CFACT</b>	Defines complex scaling factors to be used with operations.
<b>FORCE</b>	Selects the element nodal force type for output.
<b>LAYERP26</b>	Specifies the element layer for which data are to be stored.
<b>SHELL</b>	Selects a shell element or shell layer location for results output.

**These POST26 commands are used to control the calculations of other commands.**

**TVAR** Changes time to the cumulative iteration number.

### Table 2.108 Operations

**These POST26 commands are used to perform operations on the stored variables.**

<b>ABS</b>	Forms the absolute value of a variable.
<b>ADD</b>	Adds variables.
<b>ATAN</b>	Forms the arctangent of a complex variable.
<b>CLOG</b>	Forms the common log of a variable
<b>CONJUG</b>	Forms the complex conjugate of a variable.
<b>DERIV</b>	Differentiates a variable.
<b>EXP</b>	Forms the exponential of a variable.
<b>FILLDATA</b>	Fills a variable by a ramp function.
<b>IMAGIN</b>	Forms an imaginary variable from a complex variable.
<b>INT1</b>	Integrates a variable.
<b>LARGE</b>	Finds the largest (the envelope) of three variables.
<b>NLOG</b>	Forms the natural log of a variable.
<b>PROD</b>	Multiplies variables.
<b>QUOT</b>	Divides two variables.
<b>REALVAR</b>	Forms a variable using only the real part of a complex variable.
<b>SMALL</b>	Finds the smallest of three variables.
<b>SQRT</b>	Forms the square root of a variable.

### Table 2.109 Display

**These POST26 commands are used to display the results.**

<b>KEEP</b>	Stores POST26 definitions and data during active session.
<b>PLCPLX</b>	Specifies the part of a complex variable to display.
<b>PLTIME</b>	Defines the time range for which data are to be displayed.
<b>PLVAR</b>	Displays up to ten variables in the form of a graph.
<b>SPREAD</b>	Turns on a dashed tolerance curve for the subsequent curve plots.
<b>XVAR</b>	Specifies the X variable to be displayed.

Certain [graphics commands](#) also apply, such as **/AXLAB**, **/XRANGE**, and **/YRANGE**.

### Table 2.110 Listing

**These POST26 commands are used to produce tabular listings of the results.**

<b>EXTREM</b>	Lists the extreme values for variables.
<b>LINES</b>	Specifies the length of a printed page.
<b>NPRINT</b>	Defines which time points stored are to be listed.
<b>PRCPLX</b>	Defines the output form for complex variables.
<b>PRTIME</b>	Defines the time range for which data are to be listed.

**These POST26 commands are used to produce tabular listings of the results.**

**PRVAR** Lists variables vs. time (or frequency).

### Table 2.111 Special Purpose

**These POST26 commands are used for various special purposes.**

**CVAR** Computes covariance between two quantities.

**PMGTRAN** Summarizes electromagnetic results from a transient analysis.

**RESP** Generates a response spectrum.

**RPSD** Computes response power spectral density (PSD).

**SMOOTH** Allows smoothing of noisy data and provides a graphical representation of the data.

**VGET** Moves a variable into an array parameter vector.

**VPUT** Moves an array parameter vector into a variable.

### Table 2.112 Status

**These POST26 commands are for use with the STAT command.**

**DEFINE** Specifies "Data definition settings" as the subsequent status topic.

**OPERATE** Specifies "Operation data" as the subsequent status topic.

**PLOTTING** Specifies "Plotting settings" as the subsequent status topic.

**PRINT** Specifies "Print settings" as the subsequent status topic.

## 2.9. AUX2 Commands

These commands are used to examine the contents of binary files produced by the program. The commands are grouped by functionality.

### Table 2.113 Binary Files

**These AUX2 commands are used to dump the contents of binary files.**

**/AUX2** Enters the binary file dumping processor.

**DUMP** Dumps the contents of a binary file.

**FILEAUX2** Specifies the binary file to be dumped.

**FORM** Specifies the format of the file dump.

**HBMAT** Writes an assembled global matrix in Harwell-Boeing format.

**PSMAT** Writes an assembled global matrix to a postscript format that graphically displays nonzero matrix values.

**PTR** Dumps the record of a binary file.

## 2.10. AUX3 Commands

The auxiliary processor **/AUX3** allows you to operate on results files by deleting sets or by changing values.

**Table 2.114 Results Files**

**These commands are related to the **/AUX3** command.**

<b>/AUX3</b>	Enters the results file editing processor.
<b>COMPRESS</b>	Deletes all specified sets.
<b>DELETE</b>	Specifies sets in the results file to be deleted before postprocessing.
<b>FILEAUX3</b>	Specifies the results file to be edited.
<b>LIST</b>	Lists out the sets in the results file.
<b>MODIFY</b>	Changes the listed values of the data in a set.
<b>UNDELETE</b>	Removes results sets from the group of sets selected for editing.

## 2.11. AUX12 Commands

These commands are used to define radiation options for use in thermal analyses. The commands are grouped by functionality.

**Table 2.115 General Radiation**

**These AUX12 commands are used to specify general radiation parameters.**

<b>/AUX12</b>	Enters the radiation matrix generation processor.
<b>STEF</b>	Specifies Stefan-Boltzmann radiation constant.
<b>TOFFST</b>	Specifies the temperature offset from absolute zero to zero.

**Table 2.116 Radiation Matrix Method**

**These AUX12 commands are used to calculate radiation view factors and to create a radiation substructure for the Radiation Matrix method.**

<b>EMIS</b>	Specifies the emissivity as a material property.
<b>GEOM</b>	Defines the geometry specifications for the radiation matrix calculation.
<b>MPRINT</b>	Specifies that radiation matrices are to be printed.
<b>SPACE</b>	Defines a space node for radiation.
<b>VTTYPE</b>	Specifies the viewing procedure used to determine the form factors.
<b>WRITE</b>	Writes the radiation matrix file.

**Table 2.117 Radiosity Solver**

**These AUX12 commands are used to calculate the radiation view factors and to specify the solution parameters for the Radiosity solver method.**

<b>HEMIOPT</b>	Specifies options for Hemicube view factor calculation.
<b>RADOPT</b>	Specifies Gauss-Seidel Radiosity Solver options.
<b>SPCNOD</b>	Defines a space node for radiation using the Radiosity method.

**These AUX12 commands are used to calculate the radiation view factors and to specify the solution parameters for the Radiosity solver method.**

<b>SPCTEMP</b>	Defines a free-space ambient temperature for radiation using the Radiosity method.
<b>V2DOPT</b>	Specifies 2-D/axisymmetric view factor calculation options.
<b>VFOPT</b>	Specifies options for view factor file.
<b>VFQUERY</b>	Queries and prints element Hemicube view factors and average view factor.

## 2.12. AUX15 Commands

These commands are used to read in an IGES file for analysis in ANSYS. The commands are grouped by functionality.

**Table 2.118 IGES**

**These AUX15 commands read an IGES format file into PREP7 data.**

<b>/AUX15</b>	Enters the IGES file transfer processor.
<b>IGESIN</b>	Transfers IGES data from a file into ANSYS.
<b>IOPTN</b>	Controls options relating to importing a model.

## 2.13. OPTIMIZATION Commands

These commands are used for design optimization analyses. The commands are grouped by functionality.

**Table 2.119 Specifications**

**These OPTIMIZATION commands set up the design optimization variables.**

<b>OPEQN</b>	Controls curve fitting for the subproblem approximation method.
<b>OPFACT</b>	Defines the type of factorial evaluation to be performed.
<b>OPFRST</b>	Defines specifications for the first order optimization method.
<b>OPGRAD</b>	Specifies which design set will be used for gradient evaluation.
<b>OPKEEP</b>	Specifies whether to save the best-set results and database file.
<b>OPLOOP</b>	Specifies controls for optimization looping.
<b>OPPRNT</b>	Activates detailed optimization summary printout.
<b>OPRAND</b>	Defines the number of iterations for a random optimization.
<b>OPSUBP</b>	Defines number of iterations for subproblem approximation method.
<b>OPSWEEP</b>	Specifies the reference point and number of evaluation points for a sweep generation.
<b>/OPT</b>	Enters the design optimizer.
<b>OPTYPE</b>	Specifies the optimization method to be used.
<b>OPUSER</b>	Defines specifications for user-supplied external optimization.
<b>OPVAR</b>	Specifies the parameters to be treated as optimization variables.
<b>TOCOMP</b>	Defines single or multiple compliance as the topological optimization function.

**These OPTIMIZATION commands set up the design optimization variables.**

<b>TODEF</b>	Defines parameters for and initializes topological optimization.
<b>TOFREQ</b>	Defines single or mean frequency formulation as the topological optimization function.
<b>TOTYPE</b>	Specifies solution method for topological optimization.
<b>TOVAR</b>	Specifies the objective and constraints for the topological optimization problem.

**Table 2.120 Operations****These OPTIMIZATION commands operate on the design optimization variables.**

<b>OPADD</b>	Forms a set of optimization parameters by adding two sets.
<b>OPCLR</b>	Clears the optimization database.
<b>OPDEL</b>	Deletes optimization design sets.
<b>OPMAKE</b>	Creates a design set using active scalar parameter values.
<b>OPSEL</b>	Selects design sets for subsequent optimization looping.

**Table 2.121 Files****These OPTIMIZATION commands operate on the design optimization files.**

<b>OPANL</b>	Defines the analysis file to be used for optimization looping.
<b>OPDATA</b>	Identifies the file where optimization data is to be saved.
<b>OPRESU</b>	Reads optimization data into the optimization database.
<b>OPSAVE</b>	Writes all optimization data to a file.

**Table 2.122 Run****These OPTIMIZATION commands perform the design optimization analysis.**

<b>OPEXE</b>	Initiates optimization looping.
<b>TOEXE</b>	Executes one topological optimization iteration.
<b>TOLOOP</b>	Execute several topological optimizations iterations.

**Table 2.123 Display****These OPTIMIZATION commands are used to display the optimization results as plots.**

<b>OPLFA</b>	Displays the results of a factorial evaluation.
<b>OPLGR</b>	Graphs the results of a gradient evaluation.
<b>OPLIST</b>	Displays the parameters for design sets.
<b>OPLSW</b>	Graphs the results of a global sweep generation.
<b>OPRFA</b>	Prints the results of a factorial evaluation.
<b>OPRGR</b>	Prints the results of a gradient evaluation.
<b>OPRSW</b>	Prints the results of a global sweep generation.
<b>PLVAROPT</b>	Displays up to ten parameters in the form of a graph.
<b>PRVAROPT</b>	Lists up to ten optimization parameters.
<b>TOGRAPH</b>	Plots iteration solution of topological optimization.



These **OPTIMIZATION** commands are used to display the optimization results as plots.

<b>TOLIST</b>	Lists all topological optimization functions currently defined.
<b>TOPLOT</b>	Plot current topological density distribution.
<b>TOPRINT</b>	Print iteration solution history of topological optimization.
<b>TOSTAT</b>	Displays topological optimization status and results information.
<b>XVAROPT</b>	Specifies the parameter to be used as the X-axis variable.

## 2.14. VARIATIONAL TECHNOLOGY Commands

These commands are used for the Variational Technology products.

**Table 2.124 DesignXplorer**

These **Variational Technology** commands are part of the **ANSYS DesignXplorer**.

<b>/VT</b>	Enters the Variational Technology preprocessor.
<b>VTCLR</b>	Specifies the part of the Variational Technology database to be cleared.
<b>VTDISC</b>	Defines an element component as a discrete input variable for the DesignXplorer.
<b>VTEVAL</b>	Triggers evaluation of generated results based on input variables specified via the <b>VTVMOD</b> command.
<b>VTFREQ</b>	Defines the frequency as input variable for the harmonic sweep functionality of VT Accelerator.
<b>VTGEOM</b>	Defines a geometry parameter created with ANSYS MeshMorpher as a DesignXplorer input variable.
<b>VTIN</b>	Defines an inertial load as an input variable for DesignXplorer.
<b>VTMETH</b>	Defines the solution methods options for the DesignXplorer.
<b>VTMP</b>	Defines a material property as an input variable for the DesignXplorer.
<b>VTOP</b>	Defines options value for the DesignXplorer.
<b>VTPOST</b>	Launches the DesignXplorer postprocessing application.
<b>VTREAL</b>	Defines a real constant property as an input variable for the DesignXplorer.
<b>VTRFIL</b>	Specifies the file to which DesignXplorer results are written.
<b>VTSLT</b>	Defines a result quantity for the DesignXplorer.
<b>VTSEC</b>	Defines a section property as an input variable for DesignXplorer.
<b>VTSEF</b>	Defines a surface load as an input variable for the DesignXplorer.
<b>VTSL</b>	Selects a subset of elements associated with an VT input variable.
<b>VTSTAT</b>	Print the status of the DesignXplorer definitions and settings into a separate window.
<b>VTTEMP</b>	Defines the temperature as input variable for the DesignXplorer.

**These Variational Technology commands are part of the ANSYS DesignXplorer.**

**VTVMOD**            Modifies the status or current value of an input variable for the DesignXplorer.

### Table 2.125 Harmonic Sweep using VT Accelerator

**These Variational Technology commands are part of the harmonic sweep functionality of VT Accelerator.**

**VTFREQ**            Defines the frequency as input variable for the harmonic sweep capability of VT Accelerator or the DesignXplorer.

## 2.15. PROBABILISTIC Design Commands

These commands are used for probabilistic design analyses. The commands are grouped by functionality.

### Table 2.126 Deterministic Model

**The following PDS commands allow the specification of the deterministic model.**

**PDANL**            Defines the analysis file to be used for probabilistic looping.

### Table 2.127 Probabilistic Preprocessing

**The following PDS commands allow the specification and visualization of the probabilistic model.**

**PDCORR**            Specifies the correlation between two random input variables.

**PDINQR**            Retrieves a value and stores it as a user-parameter.

**PDPLOT**            Plots the distribution curves of a defined random input variable.

**PDVAR**            Specifies the parameters to be treated as probabilistic design variables.

### Table 2.128 Probabilistic Methods

**The following PDS commands allow the specification of the probabilistic methods and their options used in a probabilistic analysis.**

**PDDOEL**            Defines design of experiment levels for an individual random input variable.

**PDDMCS**            Specifies options for Monte Carlo Simulations.

**PDMETH**            Specifies the probabilistic analysis method.

The following PDS commands allow the specification of the probabilistic methods and their options used in a probabilistic analysis.

**PDUSER** Specifies options for user-specified sampling methods.

### Table 2.129 Run Probabilistic Analysis

The following PDS commands allow the execution of a probabilistic analysis.

**PDEXE** Executes the probabilistic analysis.

### Table 2.130 Probabilistic Postprocessing

The following PDS commands allow the probabilistic postprocessing and the visualization of the probabilistic results

**PDCDF** Plots the cumulative distribution function.  
**PDCMAT** Prints the correlation coefficient matrix.  
**PDHIST** Plots the frequency histogram.  
**PDPINV** Prints the result of the inversion of a probability.  
**PDPROB** Prints a probability result.  
**PDROPT** Specifies the options for an HTML report.  
**PDSCAT** Plots a scatter graph.  
**PDSENS** Plots the probabilistic sensitivities.  
**PDSHIS** Plots sample history values.  
**PDWRITE** Generates an HTML report for the probabilistic analysis.

### Table 2.131 Probabilistic Database

The following PDS commands provide access to the probabilistic database.

**PDCLR** Clears the probabilistic design database.  
**PDRESU** Reads the probabilistic model data and stores it in the database.  
**PDSAVE** Writes the probabilistic model data to a file.

### Table 2.132 Response Surface

The following PDS commands allow the evaluation, visualization, and use of the response surfaces.

**RSFIT** Fit a response surface for an output parameter in a solution set.  
**RSLOT** Plot a response surface.  
**RSPRNT** Print a response surface.  
**RSSIMS** Performs Monte Carlo simulations on response surface(s).

### Table 2.133 Auxiliary Commands and Information

The following commands are helpful for using the PDS.

**/PDS** Enters the probabilistic design system.

## 2.16. DISPLAY Program Commands

These commands are used for the DISPLAY program. The DISPLAY program is a companion program to ANSYS, used for recovering graphics displays produced within ANSYS. The commands are grouped by functionality.

### Table 2.134 Setup

**These DISPLAY commands are used to set up the DISPLAY program.**

<b>/CMAP</b>	Changes an existing or creates a new color mapping table.
<b>/DEVDISP</b>	Controls graphics device options.
<b>FILEDISP</b>	Specifies the file containing the graphics data.
<b>HELPDISP</b>	Displays help information on DISPLAY program commands.
<b>/INPUT</b>	Switches the input file for the commands that follow.
<b>NOCOLOR</b>	Removes color from graphics displays.
<b>/SEG</b>	Allows graphics data to be stored in the local terminal memory.
<b>/SHOWDISP</b>	Defines the display driver name.
<b>TRANS</b>	Reformats File.GRPH for improved performance with plotters.

### Table 2.135 Driver Options

**These DISPLAY commands are used to specify the graphics driver and options for subsequent plots.**

<b>TERM</b>	Specifies various terminal driver options.
-------------	--

### Table 2.136 Action

**These DISPLAY commands are used to produce the plots and exit the program.**

<b>FINISH</b>	Exits normally from a processor.
<b>PLOT</b>	Forms a display.
<b>STAT</b>	Displays the status of database settings.

## 2.17. REDUCED Order Modeling Commands

These commands are used for reduced order modeling analyses. The commands are grouped by functionality.

### Table 2.137 Set Up

**These commands are used to save or resume the ROM database.**

<b>RMRESUME</b>	Resumes ROM data from a file.
<b>RMSAVE</b>	Saves ROM data to file.

### Table 2.138 Preparation

**These commands are used to create the input files for the ROM Generation Pass.**

<b>RMNDISP</b>	Extracts neutral plane displacements from a test load or element load solution for the ROM method.
----------------	--

**These commands are used to create the input files for the ROM Generation Pass.**

<b>RMNEVEC</b>	Extracts neutral plane eigenvectors from a modal analysis for the ROM method.
----------------	---

**Table 2.139 Generation Pass**

**These commands are used to create a reduced order model.**

<b>RMALIST</b>	Lists all defined master nodes for a ROM method.
<b>RMANL</b>	Assigns model database, dimensionality, and operating direction for the ROM method.
<b>RMASTER</b>	Defines master nodes for the ROM method.
<b>RMCAP</b>	Defines lumped capacitance pairs between conductors C1 and C2 for a ROM method.
<b>RMCLIST</b>	Lists all lumped capacitances defined.
<b>RMMLIST</b>	Lists all mode specifications for the ROM method.
<b>RMMRANGE</b>	Defines and edits various modal parameters for the ROM method.
<b>RMMSELECT</b>	Selects modes for the ROM method.
<b>RMPORDER</b>	Defines polynomial orders for ROM functions.
<b>RMRGENERATE</b>	Performs fitting procedure for all ROM functions to generate response surfaces.
<b>RMROPTIONS</b>	Defines options for ROM response surface fitting.
<b>RMRPLOT</b>	Plots response surface of ROM function or its derivatives with respect to the dominant mode(s).
<b>RMRSTATUS</b>	Prints status of response surface for ROM function.
<b>RMSMPLE</b>	Runs finite element solutions and obtains sample points for the ROM method.
<b>RMXPORT</b>	Exports ROM model to external VHDL-AMS simulator.

**Table 2.140 Use Pass**

**These commands use the reduced order model in an analysis.**

<b>DCVSWP</b>	Performs a DC voltage sweep on a ROM element.
<b>RMLVSCALE</b>	Defines element load vector scaling for a ROM use pass.
<b>RMUSE</b>	Activates ROM use pass for ROM elements.

## 2.18. Menu-Inaccessible Commands

These command operations are available via the [command input area](#) or batch file input only and are inaccessible from menus within the ANSYS GUI:

<b>ACCOPTION</b>	Specifies GPU accelerator capability options.
<b>ANSTOAQWA</b>	Creates an AQWA-LINE input file from the current ANSYS model.
<b>ANSTOASAS</b>	Creates an ASAS input file from the current ANSYS model.
<b>AREAS</b>	Specifies "Areas" as the subsequent status topic.
<b>/AUX3</b>	Enters the results file editing processor.

<b>*AXPY</b>	Performs the matrix operation $M2 = v*M1 + w*M2$
<b>/BATCH</b>	Sets the program mode to "batch."
<b>BCSOPTION</b>	Sets memory options for the sparse solver.
<b>C***</b>	Places a comment in the output.
<b>CAMPBELL</b>	Prepares the result file for a subsequent <a href="#">Campbell diagram</a> analysis.
<b>CECHECK</b>	Checks constraint equations and couplings for rigid body motions.
<b>CECYC</b>	Generates the constraint equations for a <a href="#">cyclic symmetry analysis</a> .
<b>CHKMSH</b>	Checks area and volume entities for previous meshes.
<b>CINT</b>	Defines parameters associated with contour integral calculations.
<b>CISOL</b>	Stores J-integral information in a variable.
<b>CLOCAL</b>	Defines a local coordinate system relative to the active coordinate system.
<b>/CLOG</b>	Copies the session log file to a named file.
<b>CLRMSHLN</b>	Clears meshed entities.
<b>CMROTATE</b>	Specifies the rotational velocity of an element component about a user-defined rotational axis
<b>CMWRITE</b>	Writes components and assemblies to a file.
<b>CNKMOD</b>	Modifies contact element key options.
<b>*COMP</b>	Compresses the columns of a matrix using a specified algorithm.
<b>CORIOLIS</b>	Applies the Coriolis effect to a <a href="#">rotating structure</a> .
<b>CZDEL</b>	Edits or clears cohesive zone sections.
<b>CZMESH</b>	Creates and meshes a cohesive zone group of elements.
<b>DDOPTION</b>	Sets domain decomposer option for Distributed ANSYS.
<b>DELETE</b>	Specifies sets in the results file to be deleted before postprocessing.
<b>*DMAT</b>	Creates a dense matrix.
<b>*EIGEN</b>	Performs a modal solution with unsymmetric or damping matrices.
<b>EMATWRITE</b>	Forces the writing of all the element matrices to File.EMAT.
<b>EMORE</b>	Adds more nodes to the just-defined element.
<b>ENERSOL</b>	Specifies the total energies to be stored.
<b>/EOF</b>	Exits the file being read.
<b>*EXPORT</b>	Exports a matrix to a file in the specified format.
<b>EXPROFILE</b>	Exports ANSYS interface loads to a CFX Profile file.
<b>EXUNIT</b>	Specifies units of an interface load for ANSYS to CFX transfer.
<b>FILEAUX3</b>	Specifies the results file to be edited.
<b>FITEM</b>	Identifies items chosen by a picking operation (GUI).
<b>FLDATA27</b>	Controls dependent variable printing.
<b>FLST</b>	Specifies data required for a picking operation (GUI).
<b>/FORMAT</b>	Specifies format controls for tables.
<b>*FREE</b>	Deletes a matrix or a solver object and frees its memory allocation.
<b>/GOLIST</b>	Reactivates the suppressed data input listing.
<b>/HEADER</b>	Sets page and table heading print controls.

<b>HELP</b>	Displays help information on ANSYS commands and element types.
<b>HFADP</b>	Turns a high-frequency adaptive error calculation on or off.
<b>HFDEEM</b>	Calibrates S-parameter phase shift.
<b>HFMODPRT</b>	Species a working frequency for a modal port.
<b>HRCPLX</b>	Computes and stores in the database the time-harmonic solution at a prescribed phase angle.
<b>/IMAGE</b>	Allows graphics data to be captured and saved.
<b>INISTATE</b>	Defines initial state data and parameters.
<b>IRLIST</b>	Prints inertia relief summary table.
<b>*ITENGINE</b>	Performs a solution using an iterative solver.
<b>JSOL</b>	Specifies result items to be stored for the joint element.
<b>LINE</b>	Specifies "Lines" as the subsequent status topic.
<b>LIST</b>	Lists out the sets in the results file.
<b>*LIST</b>	Displays the contents of an external, coded file.
<b>LPRT</b>	Defines impedance and calibration lines for excitation eigenfield.
<b>*LSBAC</b>	Performs the solve (forward/backward substitution) of a factorized linear system.
<b>*LSENGINE</b>	Creates a linear solver engine.
<b>*LSFACTOR</b>	Performs the numerical factorization of a linear solver system.
<b>MEMM</b>	Allows the current session to keep allocated memory.
<b>/MENU</b>	Activates the Graphical User Interface (GUI).
<b>/MKDIR</b>	Creates a directory.
<b>/MREP</b>	Enables you to reissue the graphics command macro "name" during a replot or zoom operation.
<b>/MSTART</b>	Controls the initial GUI components.
<b>MSTOLE</b>	Adds two extra nodes from <b>FLUID116</b> elements to <b>SURF152</b> elements for convection analyses.
<b>*MULT</b>	Performs the matrix multiplication $M3 = M1(T1)*M2(T2)$ .
<b>/NOLIST</b>	Suppresses the data input listing.
<b>NOOFFSET</b>	Prevents the <b>CDREAD</b> command from offsetting specified data items.
<b>*NRM</b>	Computes the norm of the specified matrix or vector.
<b>NSMOOTH</b>	Smooths selected nodes among selected elements.
<b>NSVR</b>	Defines the number of variables for user-programmable element options.
<b>OPERATE</b>	Specifies "Operation data" as the subsequent status topic.
<b>/PAGE</b>	Defines the printout and screen page size.
<b>PAUSE</b>	Temporarily releases (pauses) the currently used product license so that another application can use it.
<b>PCGOPT</b>	Controls PCG solver options.
<b>/PCOPY</b>	Automatically generates hard copies for HP UNIX work stations.
<b>PERI</b>	Specifies periodic boundary conditions in an incompressible flow analysis.

<b>PERTURB</b>	Sets linear perturbation analysis options.
<b>PLCAMP</b>	Plots Campbell diagram data for rotordynamics applications.
<b>PLCINT</b>	Plots the contour integral ( <b>CINT</b> ) result data.
<b>PLFAR</b>	Plots electric far fields and far field parameters.
<b>PLFSS</b>	Plots reflection and transmission parameters of a frequency selective surface solution.
<b>PLNEAR</b>	Prints the electric field in the near zone exterior to the equivalent source surface (flagged with the Maxwell surface flag in the preprocessor).
<b>PLORB</b>	Displays the orbital motion of a rotating structure undergoing vibration.
<b>PLTD</b>	Displays TDR/TDT waveforms, an impedance profile, or a total waveform.
<b>PLTLINE</b>	Plots port transmission line data generated by the HFPCSWP or SPSWP macros.
<b>PRCAMP</b>	Prints Campbell diagram data for rotordynamics applications.
<b>PRCINT</b>	Lists the J-integral result data.
<b>PREENERGY</b>	Prints the total energies of a model.
<b>PRFAR</b>	Prints electric far fields and far field parameters.
<b>*PRINT</b>	Prints the matrix values to a file.
<b>PRISM</b>	Creates a prism volume based on working plane coordinate pairs.
<b>PRJSOL</b>	Prints joint element output.
<b>PRNEAR</b>	Prints the electric field in the near zone exterior to the equivalent source surface (flagged with the Maxwell surface flag in the preprocessor).
<b>PRORB</b>	Prints the orbital motion characteristics of a rotating structure undergoing vibration.
<b>PRRFOR</b>	Used with the <b>FORCE</b> command. Prints the constrained node reaction solution.
<b>PSCONTROL</b>	Activate or deactivate shared-memory parallel operations during solution.
<b>PSMAT</b>	Writes an assembled global matrix to a postscript format that graphically displays nonzero matrix values.
<b>RESVEC</b>	Calculates residual vectors in a modal analysis, or includes residual vectors in a modal superposition transient/harmonic analysis.
<b>RESWRITE</b>	Appends results data from the database to a results file in a <a href="#">top-down substructured</a> analysis.
<b>/RMDIR</b>	Removes (deletes) a directory.
<b>RSOPT</b>	Creates or loads the radiosity mapping data file for <b>SURF251</b> or <b>SURF252</b> element types.
<b>RSPLIT</b>	Creates one or more results files from the current results file based on subsets of elements.
<b>RSTOFF</b>	Offsets node or element IDs in the FE geometry record.
<b>SELTOL</b>	Sets the tolerance for subsequent select operations.



<b>*SMAT</b>	Creates a sparse matrix.
<b>/SMBC</b>	Controls the display of solid model boundary condition symbols and labels.
<b>SNOPTION</b>	Specifies Supernode (SNODE) eigensolver options.
<b>SOURCE</b>	Defines a default location for undefined nodes or keypoints.
<b>SPADP</b>	Automatically refines a <b>HF119</b> tetrahedral element mesh based on S-parameter convergence.
<b>SPFSS</b>	Performs frequency sweep or angle sweep calculations to determine reflection and transmission parameters of a frequency selective surface.
<b>SPICE</b>	Generates a SPICE subcircuit model using S-parameters from a Touchstone file.
<b>SPLOT</b>	Displays the selected areas and a faceted view of their underlying surfaces.
<b>/STITLE</b>	Defines subtitles.
<b>SYNCHRO</b>	Specifies whether the excitation frequency is synchronous or asynchronous with the rotational velocity of the structure.
<b>/SYP</b>	Passes a command string and arguments to the operating system.
<b>/SYS</b>	Passes a command string to the operating system.
<b>TBEO</b>	Sets special options or parameters for material data tables.
<b>TBFIELD</b>	Defines values of field variables for the material data tables.
<b>TERM</b>	Specifies various terminal driver options.
<b>TOEXE</b>	Executes one topological optimization iteration.
<b>/UNITS</b>	Annotates the database with the system of units used.
<b>UNDELETE</b>	Removes results sets from the group of sets selected for editing.
<b>UNPAUSE</b>	Restores use of a temporarily released (paused) product license. (Applicable only <i>after</i> a previously issued <b>PAUSE</b> command.)
<b>USRDOF</b>	Specifies the degrees of freedom for the user-defined element <b>USER300</b> .
<b>USRELEM</b>	Specifies the characteristics of the user-defined element <b>USER300</b> .
<b>*VEC</b>	Creates a vector.
<b>VOLUMES</b>	Specifies "Volumes" as the subsequent status topic.
<b>VTIN</b>	Defines an inertial load as an input variable for DesignXplorer
<b>VTOP</b>	Defines options value for the DesignXplorer.
<b>VTSFE</b>	Defines a surface load as an input variable for the DesignXplorer.
<b>WMID</b>	Specifies reordering options for the WAVES command.



---

## Chapter 3: Command Dictionary

---

This chapter contains a dictionary of the ANSYS commands, listed in alphabetical order. The star (\*) and slash (/) of the star and slash commands are ignored for alphabetization (for example, the **/SHOW** command appears between the **SHELL** and **SHPP** commands). As in a dictionary, keywords are located at the top of each page (in the printed version only) indicating the first and last commands contained on that page.

The following section documents the components of a command description.

### Components of a Command Description

#### **SAMPLECMD, Key**

Specifies whether to use automatic time stepping or load stepping.

SOLUTION: Load Step Options

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### Argument Descriptions

#### **Key**

Automatic time stepping key:

#### **OFF --**

Do not use automatic time stepping

#### **ON --**

Use automatic time stepping.

**Default:** No automatic time stepping.

### Notes

Specifies whether to use automatic time stepping (or load stepping) over this load step. If *Key* = ON, both time step prediction and time step bisection will be used. Used only if *DTIME* (specified on the **DELTIM** command) is less than the time span or conversely, if *NSBSTP* (on the **NSUBST** command) is greater than one.

This command is also valid in PREP7.

### Product Restrictions

In ANSYS Professional, *Key* is automatically set to ON and cannot be changed.

### Menu Paths

**Main Menu > Preprocessor > Loads > Load Step Opts > Time/Frequenc > Freq and Substps**

**Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time - Time Step**

**Main Menu> Solution> Load Step Opts> Time/Frequenc> Freq and Substps**

**Main Menu> Solution> Load Step Opts> Time/Frequenc> Time - Time Step**

## Features Documented in ANSYS Commands

The first line of the command description shows the command name followed by the argument names (if any). The second line summarizes the command function. The summary is *not* intended to be all inclusive:

### **SAMPLECMD,Key**

Specifies whether to use automatic time stepping or load stepping.

Listed on the next line are codes that will help you find other commands in the program with related functionality.

SOLUTION: Load Step Options

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

"SOLUTION: Load Step Options" is the code in the example above. There is at least one group code for each command. The first portion of the code before the colon (:) tells which of the tables in [Command Groupings \(p. 9\)](#) the command can be found in. These are major groupings, such as PREP7 or APDL commands. In the example above, SOLUTION commands are found in [SOLUTION Commands \(p. 40\)](#). Most of these major groupings are processors, and in those cases it is implied that the command can be entered only when in that processor. Any exceptions are noted under "Notes" later in the description.

The second portion of the code, after the colon, is the subtable in [Command Groupings \(p. 9\)](#) containing the command. The subtables list commands that relate to each other in function. In the example, the code "Load Step Options" means that other commands related to load step options can be found in that subtable of [SOLUTION Commands \(p. 40\)](#) ([Table 2.64: Load Step Options \(p. 44\)](#)).

If you are viewing this manual in the ANSYS Help System, just click on the code to follow the link to the corresponding table. Then click on any of the commands in the table, to follow the link to its description.

If you are reading the printed version, simply turn to the appropriate table in [Command Groupings \(p. 9\)](#). The table lists the page number where the command is documented.

The next line displays a series of product codes, which may contain all of the ANSYS product codes:

MP ME ST PR PRN DS DSS FL EM EH DY PP VT EME MFS

or a subset of them:

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

For more information on product codes, see [Product Codes \(p. 2\)](#).

Following this "product code" line is the description of all arguments, if any, of the command:

#### **Key**

Automatic time stepping key:

**OFF --**

Do not use automatic time stepping (default).

**ON --**

Use automatic time stepping.

The argument is described and, where necessary, valid choices for the argument are described. Many arguments list defaults, which are the values assumed for *that* argument if you *enter the command but leave the argument blank*. On the other hand, shown after the argument descriptions is often another default:

## Command Default

No automatic time stepping.

This is the *command default*. This is the specification assumed by the program if you *do not enter the command at all*. Only commands that set specifications (*specification* commands) have defaults listed. Commands that cause some action, such as performing some calculation, are called *action* commands and simply do not perform the action if the command is not entered. Defaults are not listed for action commands.

Following the default listing are any notes about the command. These notes expand on the summary description given up near the command format, and describe any other behavior, restrictions, suggestions, etc. of that command:

## Notes

Specifies whether to use automatic time stepping (or load stepping) over this load step. If  $KEY = ON$ , both time step prediction and time step bisection will be used. Used only if  $DTIME$  (specified on the **DELTIM** command) is less than the time span or conversely, if  $NSBSTP$  (on the **NSUBST** command) is greater than one.

This command is also valid in PREP7.

If the command behaves differently, or has restrictions, in any of the various ANSYS products, those differences are described in the *Product Restrictions* section:

In ANSYS Professional,  $KEY$  is automatically set to ON and cannot be changed.

## Product Restrictions

In ANSYS Professional,  $KEY$  is automatically set to ON and cannot be changed.

## Menu Paths

**Main Menu >Preprocessor> Loads> Load Step Opts> Time/Frequenc> Freq and Substps**

**Main Menu >Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time - Time Step**

**Main Menu> Solution> Load Step Opts> Time/Frequenc >Freq and Substps**

**Main Menu> Solution> Load Step Opts> Time/Frequenc> Time - Time Step**

These menu paths are intended as guides to help you find the commands in the GUI. Be aware, however, that often the paths are valid only if some other command has been previously issued. For example, you won't be able to follow a path to transient analysis options if you have not selected a transient analysis in the first place.

Not all commands are directly accessible through the menu (although some of them may be generated indirectly by some menu function). Such cases are indicated by the following statement in the menu path listing:

## Menu Paths

This command cannot be accessed directly in the menu.

---

## Connection Commands

**~CAT5IN**, *Name, Extension, Path, Entity, FMT, NOCL, NOAN*

**Transfers a .CATPart file into the ANSYS program.**

CAD Import

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **Name**

The name of a valid .CATPart file, created with CATIA Version 5.0. The first character of the file name must be an alphanumeric.

### **Extension**

The extension for the file. The default extension is .CATPart.

### **Path**

The path name of the directory in which the file resides enclosed in single quotes. The default path name is the current working directory.

### **Entity**

Entity to be imported.

#### **SOLIDS**

Solids only, imported as ANSYS volumes (default).

#### **SURFACES**

Surfaces only, imported as ANSYS areas.

#### **ALL**

All entities. Use this option when the file contains different types of entities.

### **FMT**

The format in which ANSYS will store the model.

**0**

Neutral format (default). Defeaturing after import is restricted.

**1**

Solid format; this allows defeaturing after import.

### **NOCL**

Remove tiny objects.

**0**

Remove tiny objects without checking model validity (default).

**1**

Do not remove tiny objects.

### **NOAN**

Perform an analysis of the model.

**0**

Analyze the model (default).

- 1 Do not analyze the model.

## Notes

If defeaturing is specified ( $FMT = 1$ ), this command must be the last line of any file, script, or other interactive input.

More information on importing CATIA Version 5 parts is available in [CATIA V5](#) in the *ANSYS Connection User's Guide*.

## Menu Paths

**File > Import > CATIA5**

---

**~CATIAIN**, *Name, Extension, Path, --, --, BLANK, --*

**Transfers a CATIA model into the ANSYS program.**

CAD Import  
MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **Name**

The name of a valid CATIA model, created with CATIA 4.x or lower. The first character of the file name must be an alphanumeric.

### **Extension**

The extension for the file. The default extension is `.model`.

### **Path**

The path name of the directory in which the file resides, enclosed in single quotes. The default path name is the current working directory.

--  
Unused field.

--  
Unused field.

### **BLANK**

Sets whether to import "blanked" entities.

- 0 Does not import "blanked" (suppressed) CATIA entities (default).

- 1 Imports "blanked" entities. The portions of CATIA data that were suppressed will be included in the import.

--  
Unused field.

## Notes

More information on importing CATIA parts is available in [CATIA V4](#) in the *ANSYS Connection User's Guide*.



## Menu Paths

**File > Import > CATIA**

**~PARAIN**, *Name, Extension, Path, Entity, FMT, Scale*

**Transfers a Parasolid file into the ANSYS program.**

CAD Import

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **Name**

The name of a valid Parasolid file. The first character of the file name must be an alphanumeric.

### **Extension**

The extension for the file. The default extension is `.x_t` on a PC or `.xmt_txt` on a Unix system. Parasolid files are compatible across systems, and do not need to be renamed to be used on another platform.

### **Path**

The path name of the directory in which the file resides, enclosed in single quotes. The default path name is the current working directory.

### **Entity**

Entity to be imported:

#### **SOLIDS**

Solids only, imported as ANSYS volumes (default)

#### **SURFACES**

Surfaces only, imported as ANSYS areas.

#### **WIREFRAME**

Wireframe only, imported as ANSYS lines.

#### **ALL**

All entities. Use this option when the file contains more than one type of entity.

### **FMT**

Sets the format in which ANSYS will store the model

#### **0**

Neutral format (default). Defeaturing after import is restricted. Use this option if you need to scale a model to a specific unit of measure (other than meters).

#### **1**

Solid format; this allows defeaturing after import.

### **Scale**

Allows scaling for the model

#### **0**

Do not rescale the model; retain the default Parasolid setting of meters (default).

#### **1**

Scale the model if warranted by the model size.

Very small models will be scaled by the factor of 10 or 100 to increase the chance of successful import; the scaling factor used is displayed in the output window and in the `.para_log` file. Because scaling

changes the dimensions of the model, you must apply loads and material properties appropriately. If the model cannot be properly scaled, the analysis may fail.

If you need to scale your model to a specific set of measurements, set `FMT = 0`, then use either the **VLSCALE**, **ARSCALE** or **LSSCALE** command to select a different unit of measure.

## Notes

More information on importing Parasolid parts is available in [Parasolid](#) in the *ANSYS Connection User's Guide*.

## Menu Paths

**File > Import > PARA**

---

**~PROEIN**, *Name, Extension, Path, Proecomm, FMT*

**Transfers a Pro/ENGINEER part into the ANSYS program.**

CAD Import

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **Name**

The name of the Pro/ENGINEER part to be imported, which cannot exceed 64 characters in length and must begin with an alphanumeric character. Special characters such as & - and \* and spaces are not permitted in the part name.

### **Extension**

The general Pro/ENGINEER extension format is `prt`. The assembly extension format is `asm`

### **Path**

Full path name to the directory containing the part. The default is the current working directory. Neither the path name nor the file name should contain a space.

### **Proecomm**

The start command for the version of Pro/ENGINEER you are using. `proe1` is the default command. Note that the full path name to the Pro/ENGINEER command need not be used here if the path had been included in the `PATH` variable. The Pro/ENGINEER command name is set by the **PROE\_START\_CMD130** environment variable.

### **FMT**

The format in which ANSYS will store the model.

- 0**  
Neutral format (default). Defeaturing after import is restricted.
- 1**  
Solid format; this allows defeaturing after import.

## Notes

More information on importing Pro/ENGINEER parts is available in [Pro/ENGINEER](#) in the *ANSYS Connection User's Guide*.

## Menu Paths

**File> Import> Pro/E**

**~SATIN**, *Name, Extension, Path, Entity, FMT, NOCL, NOAN*

**Transfers a .SAT file into the ANSYS program.**

CAD Import

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **Name**

The name of a valid .SAT file, created with a supported version of ACIS. The first character of the file name must be an alphanumeric. See [File Names](#) in the *Command Reference* for more information about ANSYS file naming conventions.

### **Extension**

The extension for the file. The default extension is .sat.

### **Path**

The path name of the directory in which the file resides enclosed in single quotes. The default path name is the current working directory.

### **Entity**

Entity to be imported.

#### **SOLIDS**

Solids only, imported as ANSYS volumes (default).

#### **SURFACES**

Surfaces only, imported as ANSYS areas.

#### **WIREFRAME**

Wireframe only, imported as ANSYS lines.

#### **ALL**

All entities. Use this option when the file contains different types of entities.

### **FMT**

The format in which ANSYS will store the model.

**0**

Neutral format (default). Defeaturing after import is restricted.

**1**

Solid format; this allows defeaturing after import.

### **NOCL**

Remove tiny objects.

**0**

Remove tiny objects without checking model validity (default).

**1**

Do not remove tiny objects.

### **NOAN**

Perform an ACIS analysis of the model.

- 0** Analyze the model (default).
  - 1** Do not analyze the model.
- 

### Note

*NOCL* and *NOAN* are not supported in the ANSYS GUI.

## Notes

More information on importing SAT parts is available in [ACIS/SAT](#) in the *ANSYS Connection User's Guide*.

## Menu Paths

**File > Import > SAT**

---

**~UGIN**, *Name, Extension, Path, Entity, LAYER, FMT*

**Transfers a NX part into the ANSYS program.**

CAD Import

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **Name**

The file name of the NX part to be imported, which cannot exceed 64 characters in length. The path name must begin with an alphanumeric character. Special characters such as &, -, and \* are not permitted in the part name.

### **Extension**

The NX part file extension. The default is `.prt`.

### **Path**

The full path name to the directory containing the part, enclosed in single quotes; for example, `'/ug_parts'`. The default is the current working directory.

### **Entity**

Entity to be imported.

#### **0 or Solid**

Solids only, imported as ANSYS volumes (the default).

#### **1 or Surface**

Surfaces only, imported as ANSYS areas.

#### **2 or Wireframe**

Wireframe only, imported as ANSYS lines.

#### **3 or All**

All entities. Use this option when the part contains entities that may not be attached to each other, such as a solid in one location and a surface in another.

**LAYER**

The number(s) assigned to the layer(s) to be imported. You can import one layer or a range of layers (designated by hyphens). Defaults to 1-256 (all layers).

**FMT**

The format in which ANSYS will store the model.

**0**

Neutral format (default). Defeaturing after import is restricted.

**1**

Solid format; this allows defeaturing after import.

**Notes**

More information on importing NX parts is available in [UG/NX](#) in the *ANSYS Connection User's Guide*.

**Menu Paths**

**File> Import> UG**

---



---

## A Commands

**A**, *P1, P2, P3, P4, P5, P6, P7, P8, P9, P10, P11, P12, P13, P14, P15, P16, P17, P18*

**Defines an area by connecting keypoints.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*P1, P2, P3, . . . , P18*

List of keypoints defining the area (18 maximum if using keyboard entry). At least 3 keypoints must be entered. If *P1 = P*, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

### Notes

Keypoints (*P1* through *P18*) must be input in a clockwise or counterclockwise order around the area. This order also determines the positive normal direction of the area according to the right-hand rule. Existing lines between adjacent keypoints will be used; missing lines are generated "straight" in the active coordinate system and assigned the lowest available numbers [[NUMSTR](#)]. If more than one line exists between two keypoints, the shorter one will be chosen. If the area is to be defined with more than four keypoints, the required keypoints and lines must lie on a constant coordinate value in the active coordinate system (such as a plane or a cylinder). Areas may be redefined only if not yet attached to a volume. Solid modeling in a toroidal coordinate system is not recommended.

### Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>Through KPs**

---

**AADD**, *NA1, NA2, NA3, NA4, NA5, NA6, NA7, NA8, NA9*

**Adds separate areas to create a single area.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NA1, NA2, NA3, . . . , NA9*

Numbers of areas to be added. If *NA1 = ALL*, add all selected areas and ignore *NA2* to *NA9*. If *NA1 = P*, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

### Notes

The areas must be coplanar. The original areas (and their corresponding lines and keypoints) will be deleted by default. See the [BOPTN](#) command for the options available to Boolean operations. Element attributes

and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. Concatenated entities are not valid with this command.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Add>Areas**

---

**AATT**, *MAT*, *REAL*, *TYPE*, *ESYS*, *SECN*

**Associates element attributes with the selected, unmeshed areas.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **MAT**

The material number to be associated with selected, unmeshed areas.

### **REAL**

The real constant set number to be associated with selected, unmeshed areas.

### **TYPE**

The type number to be associated with selected, unmeshed areas.

### **ESYS**

The coordinate system number to be associated with selected, unmeshed areas.

### **SECN**

The section number to be associated with selected unmeshed areas.

## Notes

Areas subsequently generated from the areas will also have these attributes. These element attributes will be used when the areas are meshed. If an area does not have attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT**, **REAL**, **TYPE**, **ESYS**, and **SECN** command settings. Reissue the **AATT** command (before areas are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association. If any of the arguments *MAT*, *REAL*, *TYPE*, *ESYS*, or *SECN* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, ANSYS can proceed with an area meshing operation even when no logical element type has been assigned via **AATT,,,TYPE** or **TYPE**. For more information, see the discussion on setting element attributes in [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide*.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh Attributes>All Areas**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked Areas**

---



## ABEXTRACT, *MODE1*, *MODE2*

Extracts the alpha-beta damping multipliers for Rayleigh damping.

[SOLUTION: Analysis Options](#)

MP ME <> <> <> <> <> <> <> <> <> PP <> EME MFS

### *MODE1*

First mode number.

### *MODE2*

Second mode number.

## Notes

**ABEXTRACT** calls the command macro **DMPEXT** to extract the damping ratio of *MODE1* and *MODE2* and then computes the Alpha and Beta damping multipliers for use in a subsequent structural harmonic or transient analysis. See [Damping](#) in the [Structural Analysis Guide](#) for more information on the alpha and beta damping multipliers. The damping multipliers are stored in parameters ALPHADMP and BETADMP and can be applied using the **ALPHAD** and **BETAD** commands. Before calling **ABEXTRACT**, you must issue **RMFLVEC** to extract the modal displacements. In addition, a node component FLUN must exist from all **FLUID136** nodes. See ["Thin Film Analysis"](#) for more information on thin film analyses.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>ThinFilm>RayleighDamp

## \*ABBR, *Abbr*, *String*

Defines an abbreviation.

[APDL: Abbreviations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Abbr*

The abbreviation (up to 8 alphanumeric characters) used to represent the string *String*. If *Abbr* is the same as an existing ANSYS command, the abbreviation overrides. Avoid using an *Abbr* which is the same as an ANSYS command.

### *String*

String of characters (60 maximum) represented by *Abbr*. Cannot include a \$ or any of the commands **C\*\*\***, **/COM**, **/GOPR**, **/NOPR**, **/QUIT**, **/UI**, or **\*END**. Parameter names and commands of the **\*DO** and Use the **\*IF** groups may not be abbreviated. If *String* is blank, the abbreviation is deleted. To abbreviate multiple commands, create an "unknown command" macro or define *String* to execute a macro file [**\*USE**] containing the desired commands.

## Notes

Once the abbreviation *Abbr* is defined, you can issue it at the beginning of a command line and follow it with a blank (or with a comma and appended data), and the program will substitute the string *String* for *Abbr* as the line is executed. Up to 100 abbreviations may exist at any time and are available throughout the program. Abbreviations may be redefined or deleted at any time.

Use **\*STATUS** to display the current list of abbreviations. For abbreviations repeated with **\*REPEAT**, substitution occurs before the repeat increments are applied. There are a number of abbreviations that are pre-defined by the program (these can be deleted by using the blank *String* option described above). Note that *String* will be written to the `File.LOG`.

This command is valid in any processor.

## Menu Paths

**Utility Menu>Macro>Edit Abbreviations**  
**Utility Menu>MenuCtrls>Edit Toolbar**

---

**ABBRES**, *Lab*, *Fname*, *Ext*, --

**Reads abbreviations from a coded file.**

[APDL: Abbreviations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Lab*

Label that specifies the read operation:

#### **NEW --**

Replace current abbreviation set with these abbreviations (default).

#### **CHANGE --**

Extend current abbreviation set with these abbreviations, replacing any of the same name that already exist.

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to `ABBR` if *Fname* is blank.

### --

Unused field.

## Notes

The abbreviation file may have been written with the **ABBSAV** command. Do not issue **ABBRES,NEW** while inside an executing abbreviation. Doing so will cause all data for the executing abbreviation to be deleted.

This command is valid in any processor.

## Menu Paths

**Utility Menu>Macro>Restore Abbr**  
**Utility Menu>MenuCtrls>Restore Toolbar**

---

**ABBSAV**, *Lab*, *Fname*, *Ext*, --

**Writes the current abbreviation set to a coded file.**

[APDL: Abbreviations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Lab*

Label that specifies the write operation:

#### **ALL --**

Write all abbreviations (default).

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to `ABBR` if *Fname* is blank.

### --

Unused field.

## Notes

Existing abbreviations on this file, if any, will be overwritten. The abbreviation file may be read with the **ABBRES** command.

This command is valid in any processor.

## Menu Paths

**Utility Menu>Macro>Save Abbr**  
**Utility Menu>MenuCtrls>Save Toolbar**

**ABS**, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*

**Forms the absolute value of a variable.**

POST26: Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to NV [NUMVAR]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

***IA***

Reference number of the variable to be operated on.

--, --

Unused fields.

***Name***

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --

Unused fields.

***FACTA***

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

## Notes

The new variable is calculated as:

$$IR = | FACTA \times IA |$$

For a complex number ( $a + ib$ ), the absolute value is the magnitude, where the *IA* values are obtained from:

$$\sqrt{a^2 + b^2}$$

See [POST26 - Data Operations](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details.

## Menu Paths

**Main Menu>TimeHist Postpro>Math Operations>Absolute Value**

**ACCAT, NA1, NA2****Concatenates multiple areas in preparation for mapped meshing.**

PREP7: Meshing

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**NA1, NA2**

Areas to be concatenated. If *NA1* = ALL, *NA2* will be ignored and all selected areas [ASEL] will be concatenated. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* is ignored).

**Notes**

Concatenates multiple, adjacent areas (the input areas) into one area (the output area) in preparation for mapped meshing. A volume that contains too many areas for mapped meshing can still be mapped meshed if some of the areas in that volume are first concatenated (see [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide* for details on mapped meshing restrictions).

Because of modeling restrictions that result from its use, **ACCAT** is meant to be used solely for meshing. Specifically, (a) the output area and any volumes that have the output area on their area list [VLIST] cannot be used as input to any other solid modeling operation (not even another **ACCAT** command); and (b) the output area cannot accept solid model boundary conditions [DA, SFA].

The output area (or volumes which contain it) will be meshed [AMESH, VMESH] by meshing the input areas, which themselves must be meshable. The output area from the **ACCAT** operation will be coincident with the input areas and the input areas will be retained. Consider the **AADD** command instead of **ACCAT** if you wish to delete the input areas. When an **ACCAT** command is issued, volume area lists [VLIST] that contain *all* of the input areas will be updated so that the volume area lists refer to the output area instead of the input area. Deletion of the output area [ADELE] effectively reverses the **ACCAT** operation and restores volume area lists to their original condition. **ACCAT** operations on pairs of adjacent four-sided areas automatically concatenate appropriate lines [LCCAT]; in all other situations, line concatenations must be addressed by the user.

You can use the **ASEL** command to select areas that were created by concatenation, and then follow it with an **ADELE,ALL** command to delete them. See [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide* for a discussion on how to easily select and delete concatenated areas in one step.

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>Concatenate>Areas**

**ACCOPTION**, *Activate*, *MinSzThresh*, --, --, *SPkey*

**Specifies GPU accelerator capability options.**

SOLUTION: Analysis Options

MP ME ST PR <> <> <> <> EM <> <> PP <> EME MFS

## Argument Descriptions

### *Activate*

Activates the GPU accelerator capability within the equation solvers.

#### **OFF --**

Do not use GPU accelerator.

#### **ON --**

Use GPU accelerator.

### *MinSzThresh*

Threshold used to control the minimum size of data in the equation solver to be sent to the GPU accelerator (units are in MB). Does not apply to the JCG solver or PCG solver (including PCG Lanczos).

--, --

Unused fields.

### *SPkey*

Single precision key that controls whether computations on the GPU accelerator device use double precision or single precision. Does not apply to the JCG solver or PCG solver (including PCG Lanczos).

#### **OFF --**

Use double precision (default).

#### **ON --**

Use single precision.

## Command Default

*Activate* is set to ON when the GPU acceleration command line option (-acc) is specified upon launching ANSYS; otherwise it is OFF. *MinSzThresh* is computed automatically within the sparse direct solver based on the specific GPU device in use and the number of CPU cores specified (-np command line option). *SPkey* is set to OFF by default so that double precision accuracy is maintained when using the GPU accelerator capability.

## Notes

The GPU accelerator capability requires specific hardware to be installed on the machine. See the appropriate ANSYS, Inc. Installation Guide ([Windows](#) or [UNIX/Linux](#)) for a list of supported GPU hardware. Use of this capability also requires HPC licensing. For more information, see "GPU Accelerator Capability" in the [Advanced Analysis Techniques Guide](#).

The GPU accelerator capability is available for the sparse direct solver and the PCG and JCG iterative solvers. Static, buckling, modal, full harmonic, and full transient analyses are supported. For buckling analyses, only the Block Lanczos eigensolver is supported. For modal analyses, only the Block Lanczos and PCG Lanczos eigensolvers are supported. Activating this capability when using other equation solvers or other analysis types has no effect.

The minimum threshold value (*MinSzThresh*) is only recommended for users who want to tune the GPU capability for their specific hardware. A static decision is currently made for this threshold, whereas the optimal value is a dynamic decision based on the relative performance of the GPU and CPU cores available for the ANSYS simulation. When using older, slower CPU cores (or newer, faster GPUs), this threshold could be decreased to see if the overall solution time also decreases. When using newer, faster CPU cores (or older, slower GPUs), this threshold should typically be increased. Also, using more CPU cores generally leads to higher threshold values. Recommended values are generally in the range of 0.5 MB to 20 MB.

Use of the single precision key (*SPkey*) is generally not recommended. Activating this key should further accelerate the time to solution; however, it will also cause a loss of accuracy which could significantly alter the final results. For linear static analyses with high quality meshes, this key may be used to generate a relatively accurate solution. Also, when using the GPU capability for modal analyses with the Block Lanczos eigensolver, this key may be used to generate relatively accurate eigenvalue solutions.

The GPU accelerator capability is supported only on the Windows 64-bit and Linux 64-bit platforms.

**Distributed ANSYS Restriction** The GPU accelerator capability is not currently supported with Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**ACEL**, *ACEL\_X*, *ACEL\_Y*, *ACEL\_Z*

**Specifies the linear acceleration of the global Cartesian reference frame for the analysis.**

SOLUTION: Inertia

MP ME ST PR PRN DS DSS FL <> <> DY PP <> EME MFS

*ACEL\_X*, *ACEL\_Y*, *ACEL\_Z*

Linear acceleration of the reference frame along global Cartesian X, Y, and Z axes, respectively.

## Notes

In the absence of any other loads or supports, the acceleration of the structure in each of the global Cartesian (X, Y, and Z) axes would be equal in magnitude but opposite in sign to that applied in the **ACEL** command. Thus, to simulate gravity (by using inertial effects), accelerate the reference frame with an **ACEL** command in the direction opposite to gravity.

You can define the acceleration for the following analyses types:

- Static (**ANTYPE**,STATIC)
- Harmonic (**ANTYPE**,HARMIC), *full* or *mode superposition* method
- Transient (**ANTYPE**,TRANS)
- Substructure (**ANTYPE**,SUBSTR).

For all but the reduced transient dynamic (**ANTYPE**,TRANS) analysis, accelerations are combined with the element mass matrices to form a body force load vector term. The element mass matrix may be formed from a mass input constant or from a nonzero density (DENS) property, depending upon the element type. For **ANTYPE**,TRANS (reduced), the acceleration is applied to the reduced mass matrix.

For analysis type **ANTYPE**,HARMIC, the acceleration is assumed to be the real component with a zero imaginary component.

Units of acceleration and mass must be consistent to give a product of force units.

The **ACEL** command supports tabular boundary conditions (%TABNAME\_X%, %TABNAME\_Y%, and %TABNAME\_Z%) for *ACEL\_X*, *ACEL\_Y*, and *ACEL\_Z* input values (**\*DIM**) as a function of both time and frequency for full transient and harmonic analyses.

Related commands for rotational effects are **CMACEL**, **CGLOC**, **CGOMGA**, **DCGOMG**, **DOMEGA**, **OMEGA**, **CMOMEGA**, and **CMDOMEGA**.

This command is also valid in **/PREP7**.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Gravity**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Gravity>Global**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Gravity**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Gravity>Global**  
**Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Gravity**  
**Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Gravity**

---

## ACLEAR, NA1, NA2, NINC

Deletes nodes and area elements associated with selected areas.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NA1, NA2, NINC

Delete mesh for areas *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the mesh for all selected areas [**ASEL**] is deleted. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

## Notes

Deletes *all* nodes and area elements associated with selected areas (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed areas and nodes associated with non-area elements will not be deleted. Attributes assigned as a result of **AATT** are maintained. In the program's response to the command, if an area, line, or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

This command is also valid for [rezoning](#). When issued during rezoning (after the **REMESH**,START command and before the **REMESH**,FINISH command), **ACLEAR** clears only the area generated by the **AREMESH** command.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Clear>Areas**

---



**ADAMS**, *NMODES*, *KSTRESS*, *KSHELL*

**Performs solutions and writes flexible body information to a modal neutral file (Jobname.MNF) for use in an ADAMS analysis.**

[SOLUTION: Analysis Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***NMODES***

Number of normal modes to be written to Jobname.MNF file (no default).

***KSTRESS***

Specifies whether to write stress or strain results:

- 0**  
Do not write stress or strain results (default).
- 1**  
Write stress results.
- 2**  
Write strain results.
- 3**  
Write both stress and strain results.

***KSHELL***

Shell element output location. This option is valid only for shell elements.

- 0, 1**  
Shell top surface (default).
- 2**  
Shell middle surface.
- 3**  
Shell bottom surface.

**Notes**

**ADAMS** invokes a predefined ANSYS macro that solves a series of analyses and then writes the modal neutral file, Jobname.MNF. This file can be imported into the ADAMS program in order to perform a rigid body dynamics simulation. For detailed information on how to use the **ADAMS** command macro to create a modal neutral file, see [Rigid Body Dynamics and the ANSYS-ADAMS Interface](#) in the *Advanced Analysis Techniques Guide*.

Before running the **ADAMS** command macro, you must specify the units with the **/UNITS** command. The interface points should be the only selected nodes when the command macro is initiated. (Interface points are nodes where constraints may be applied in ADAMS.) Only selected elements will be considered in the calculations.

By default, stress and strain data is transferred to the ADAMS program for all nodes, as specified by the *KSTRESS* value. If you want to transfer stress/strain data for only a subset of nodes, select the desired subset and create a node component named "STRESS" before running the **ADAMS** command macro. For example, you may want to select exterior nodes for the purpose of visualization in the ADAMS program.

The default filename for the modal neutral file is `Jobname.MNF`. In interactive (GUI) mode, you can specify a filename other than `Jobname.MNF`. In batch mode, there is no option to change the filename, and the modal neutral file is always written to `Jobname.MNF`.

## Menu Paths

**Main Menu>Solution>ADAMS Connection>Export to ADAMS**

---

**ADAPT**, *NSOLN*, *STARGET*, *TTARGET*, *FACMN*, *FACMX*, *KYKPS*, *KYMAC*

**Adaptively meshes and solves a model.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> FL <> <> <> PP <> EME MFS

### **NSOLN**

Number of solutions allowed (1 or more) (defaults to 5).

### **STARGET**

Target percentage for structural percent error in energy norm (SEPC) (defaults to 5). If -1, no target value is used.

### **TTARGET**

Target percentage for thermal percent error in energy norm (TEPC) (defaults to 1). If -1, no target value is used.

### **FACMN**

Minimum factor for the keypoint element size changes (defaults to 0.25).

### **FACMX**

Maximum factor for the keypoint element size changes (defaults to 2.0).

### **KYKPS**

Specifies whether element size is to be modified at selected keypoints:

**0**

Modify element size regardless of selected keypoint set (default).

**1**

Modify element size only at selected keypoints.

### **KYMAC**

Specifies which user-written auxiliary macro files are to be used:

**0**

Ignore user-written auxiliary macro files, if any (default).

**1**

Use user-written auxiliary macro files (if they exist) as follows: Use `ADAPTMSH.MAC` instead of the default meshing command sequence. Use `ADAPTSOL.MAC` instead of the default solution command sequence (**/SOLU ... SOLVE ... FINISH**).

## Notes

**ADAPT** invokes a predefined ANSYS macro for adaptive meshing and solution. The macro causes repeated runs of the PREP7, SOLUTION, and POST1 phases of the ANSYS program with mesh density refinements

based upon the percentage error in energy norm. See the [Advanced Analysis Techniques Guide](#) for additional details. After the adaptive meshing process is complete, the **ADAPT** macro automatically turns element shape checking on (**SHPP,ON**).

A copy of the macro, called `UADAPT.MAC`, is available on the ANSYS distribution medium (system dependent), and may be copied and modified by the user to suit a particular need. The modified file should be given a suitable name (`cmd.MAC`) and run as described above with the **ADAPT** command name replaced by your "`cmd`" name.

This command is also valid at the Begin level.

## Menu Paths

**Main Menu>Solution>Solve>Adaptive Mesh**

**ADD**, *IR, IA, IB, IC, Name, --, --, FACTA, FACTB, FACTC*

**Adds (sums) variables.**

POST26:Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**IR**

Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

**IA, IB, IC**

Reference numbers of the three variables to be operated on. If only two variables, leave *IC* blank. If only one, leave *IB* and *IC* blank.

**Name**

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

**--, --**

Unused fields.

**FACTA, FACTB, FACTC**

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

## Notes

Adds variables (up to three at once) according to the operation:

$$IR = (FACTA \times IA) + (FACTB \times IB) + (FACTC \times IC)$$

## Menu Paths

**Main Menu>Drop Test>Time History>Graph Variables**

**Main Menu>Drop Test>Time History>List Variables**

**Main Menu>TimeHist Postpro>Math Operations>Add**

**ADDAM**, *AF, AA, AB, AC, AD, AMIN*

**Specifies the acceleration spectrum computation constants for the analysis of shock resistance of shipboard structures.**

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**AF**

Direction-dependent acceleration coefficient for elastic or elastic-plastic analysis option (default = 0).

**AA, AB, AC, AD**

Coefficients for the DDAM acceleration spectrum equations. Default for these coefficients is zero.

**AMIN**

The minimum acceleration value in inch/sec<sup>2</sup>. It defaults to 2316 inch/sec<sup>2</sup> which equals 6g, where *g* is acceleration due to gravity (*g* = 386 inch/sec<sup>2</sup>).

## Notes

This command specifies acceleration coefficients to analyze shock resistance of shipboard equipment. These coefficients are used to compute mode coefficients according to the equations given in [Dynamic Design Analysis Method](#) in the [Theory Reference for the Mechanical APDL and Mechanical Applications](#). The form of these equations is based on the Naval NRL Dynamic Design Analysis Method. This command, along with the **VDDAM** and **SED** commands, is used with the spectrum (**ANTYPE**,SPECTR) analysis as a special purpose alternative to the **SV**, **FREQ**, and **SVTYP** commands. The mass and length units of the model must be in pounds and inches, respectively.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>DDAM Options**

**Main Menu>Solution>Load Step Opts>Spectrum>DDAM Options**

**ADELE**, *NA1, NA2, NINC, KSWP*

**Deletes unmeshed areas.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NA1, NA2, NINC**

Delete areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and all selected areas [**ASEL**] are deleted. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

**KSWP**

Specifies whether keypoints and lines are also to be deleted:

**0**

Delete areas only (default).

1

Delete areas, as well as keypoints and lines attached to specified areas but not shared by other areas.

## Notes

An area attached to a volume cannot be deleted unless the volume is first deleted.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Delete>Area and Below**

**Main Menu>Preprocessor>Modeling>Delete>Areas Only**

## ADGL, NA1, NA2, NINC

**Lists keypoints of an area that lie on a parametric degeneracy.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NA1, NA2, NINC

List keypoints that lie on a parametric degeneracy on areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL (default), *NA2* and *NINC* will be ignored and keypoints on all selected areas [**ASEL**] will be listed. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may be substituted in *NA1* (*NA2* and *NINC* will be ignored).

## Notes

See the [Modeling and Meshing Guide](#) for details on parametric degeneracies.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>List Degen Areas**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>List Degen Areas**

## ADRAG, NL1, NL2, NL3, NL4, NL5, NL6, NLP1, NLP2, NLP3, NLP4, NLP5, NLP6

**Generates areas by dragging a line pattern along a path.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NL1, NL2, NL3, . . . , NL6

List of lines in the pattern to be dragged (6 maximum if using keyboard entry). Lines should form a continuous pattern (no more than two lines connected to any one keypoint). If *NL1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *NL1* = ALL, all selected lines (except those that define the drag path) will be swept along the path. A component name may also be substituted for *NL1*.

***NLP1, NLP2, NLP3, . . . , NLP6***

List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines.

**Notes**

Generates areas (and their corresponding keypoints and lines) by sweeping a given line pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which the path lines are input (*NLP1, NLP2*, etc.). If the drag path is a single line (*NLP1*), the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given line pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves.

Keypoint, line, and area numbers are automatically assigned (beginning with the lowest available values **[NUMSTR]**). Adjacent lines use a common keypoint. Adjacent areas use a common line. For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Operate>Extrude>Lines>Along Lines**

---

**AESIZE, ANUM, SIZE,**

**Specifies the element size to be meshed onto areas.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***ANUM***

Area number of the area to which this element size specification applies. If *ANUM* = ALL, size applies to all selected areas. If *ANUM* = P, graphical picking is enabled. A component name may also be substituted for *ANUM*.

***SIZE***

Desired element size.

**Notes**

**AESIZE** allows control over the element sizing inside any area or on the face(s) of a volume.

*SIZE* controls element size on the interior of the area. For any line on the area not having its own size assignment and not controlled by keypoint size assignments, it specifies the element size along the line as well, so long as no adjacent area has a smaller size, which would take precedence. If the **AESIZE** governs the boundary and SmartSizing is on, the boundary size can be refined for curvature or proximity.

This command is also valid for [rezoning](#).

## Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Areas>All Areas

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Areas>Clr Size

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Areas>Picked Areas

## AFILLT, NA1, NA2, RAD

Generates a fillet at the intersection of two areas.

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NA1

Number of the first intersecting area. If  $NA1 = P$ , graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

### NA2

Number of the second intersecting area.

### RAD

Radius of fillet to be generated.

## Notes

Generates an area of constant fillet radius at the intersection of two areas using a series of Boolean operations. Corresponding lines and keypoints are also generated. See [BOPTN](#) command for an explanation of the options available to Boolean operations. If areas do not initially intersect at a common line, use the [AINA](#) command.

## Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Area Fillet

## AFLIST

Lists the current data in the database.

[PREP7: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Lists the current data and specifications in the database. If batch, lists all appropriate data. If interactive, lists only summaries.

## Menu Paths

Utility Menu>List>Other>Database Summary

**AFSURF**, *SAREA*, *TLINE*

**Generates surface elements overlaid on the surface of existing solid elements and assigns the extra node as the closest fluid element node.**

[PREP7: Elements](#)

MP ME <> PR PRN <> <> <> <> <> PP <> EME MFS

***SAREA***

Component name for the surface areas of the meshed solid volumes.

***TLINE***

Component name for the target lines meshed with fluid elements.

**Notes**

This command macro is used to generate surface effect elements overlaid on the surface of existing solid elements and, based on proximity, to determine and assign the extra node for each surface element. The underlying volumes of the solid region and the fluid lines must be meshed prior to calling this command macro. The active element type must be [SURF152](#) with appropriate settings for KEYOPT(4), KEYOPT(5), KEYOPT(6), and KEYOPT(8).

The surface areas of the solid and the target lines of the fluid are grouped into components and named using the **CM** command. The names must be enclosed in single quotes (e.g., '*SAREA*') when the **AFSURF** command is manually typed in.

When using the GUI method, node and element components are created through the picking dialog boxes associated with this command.

The macro is applicable for the [SURF152](#) and [FLUID116](#) element types.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Area to Fluid**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Line to Fluid**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Node to Fluid**

**\*AFUN**, *Lab*

**Specifies units for angular functions in parameter expressions.**

[APDL: Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Argument Descriptions*****Lab***

Specifies the units to be used:



**RAD --**

Use radians for input and output of parameter angular functions (default).

**DEG --**

Use degrees for input and output of parameter angular functions.

**STAT --**

Show current setting (DEG or RAD) for this command.

## Command Default

Use radians for input or output of parameter angular functions.

## Notes

Only the SIN, COS, TAN, ASIN, ACOS, ATAN, ATAN2, ANGLEK, and ANGLN functions [**\*SET**, **\*VFUN**] are affected by this command.

## Menu Paths

Utility Menu>Parameters>Angular Units

**AGEN**, *ITIME*, *NA1*, *NA2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

**Generates additional areas from a pattern of areas.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***ITIME***

Do this generation operation a total of *ITIMEs*, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be more than 1 for generation to occur.

***NA1, NA2, NINC***

Generate areas from the pattern of areas *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is all selected areas [**ASEL**]. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

***DX, DY, DZ***

Keypoint location increments in the active coordinate system (--, D  $\theta$ , DZ for cylindrical; --, D  $\theta$ , -- for spherical).

***KINC***

Keypoint number increment between generated sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

***NOELEM***

Specifies if elements and nodes are also to be generated:

**0**

Generate nodes and elements associated with the original areas, if they exist.

**1**

Do not generate nodes and elements.

**IMOVE**

Specifies whether to redefine the existing areas:

**0**

Generate new areas as requested with the *ITIME* argument.

**1**

Move original areas to new position, retaining the same keypoint numbers (*ITIME*, *KINC*, and *NOELEM* are ignored). If the original areas are needed in the original position (e.g., they may be attached to a volume), they are not moved, and new areas are generated instead. Meshed items corresponding to moved areas are also moved if not needed at their original position.

## Notes

Generates additional areas (and their corresponding keypoints, lines and mesh) from a given area pattern. The *MAT*, *TYPE*, *REAL*, *ESYS*, and *SECNUM* attributes of the new areas are based upon the areas in the pattern and not upon the current settings of the pointers. End slopes of the generated lines remain the same (in the active coordinate system) as those of the given pattern. For example, radial slopes remain radial. Generations which produce areas of a size or shape different from the pattern (i.e., radial generations in cylindrical systems, radial and phi generations in spherical systems, and theta generations in elliptical systems) are not allowed. Solid modeling in a toroidal coordinate system is not recommended. Area and line numbers are automatically assigned, beginning with the lowest available values [**NUMSTR**].

## Menu Paths

**Main Menu>Preprocessor>Modeling>Copy>Areas**

**Main Menu>Preprocessor>Modeling>Move / Modify>Areas>Areas**

---

## **AGLUE**, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

**Generates new areas by "gluing" areas.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **NA1**, **NA2**, **NA3**, . . . , **NA9**

Numbers of the areas to be glued. If *NA1* = ALL, all selected areas will be glued (*NA2* to *NA9* will be ignored). If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

## Notes

Use of the **AGLUE** command generates new areas by "gluing" input areas. The glue operation redefines the input areas so that they share lines along their common boundaries. The new areas encompass the same geometry as the original areas. This operation is only valid if the intersection of the input areas are lines along the boundaries of those areas. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to new entities generated.

The **AGLUE** command results in the merging of lines and keypoints at the common area boundaries. The lines and keypoints of the lower numbered area will be kept. This means one must be aware of area numbering when multiple **AGLUE** commands are applied to avoid any "ungluing" of geometry.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Glue>Areas

---

**AINA**, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

**Finds the intersection of areas.**

PREP7: Booleans

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NA1*, *NA2*, *NA3*, . . . , *NA9***

Numbers of areas to be intersected. If *NA1* = ALL, *NA2* to *NA9* are ignored and the intersection of all selected areas is found. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

## Notes

Finds the common (not pairwise) intersection of areas. The common intersection is defined as the regions shared (in common) by **all** areas listed on this command. New areas will be generated where the original areas intersect. If the regions of intersection are only lines, new lines will be generated instead. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Common>Areas

---

**AINP**, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

**Finds the pairwise intersection of areas.**

PREP7: Booleans

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NA1*, *NA2*, *NA3*, . . . , *NA9***

Numbers of areas to be intersected pairwise. If *NA1* = ALL, *NA2* to *NA9* are ignored and the pairwise intersection of all selected areas is found. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may be substituted for *NA1*.

## Notes

Finds the pairwise intersection of areas. The pairwise intersection is defined as all regions shared by any two or more areas listed on this command. New areas will be generated where the original areas intersect pairwise. If the regions of pairwise intersection are only lines, new lines will be generated. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Pairwise>Areas

---

**AINV**, *NA*, *NV*

**Finds the intersection of an area with a volume.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NA**

Number of area to be intersected. If P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

**NV**

Number of volume to be intersected.

## Notes

New areas will be generated where the areas intersect the volumes. If the regions of intersection are only lines, new lines will be generated instead. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Area with Volume

---

**AL**, *L1*, *L2*, *L3*, *L4*, *L5*, *L6*, *L7*, *L8*, *L9*, *L10*

**Generates an area bounded by previously defined lines.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**L1**, **L2**, **L3**, ..., **L10**

List of lines defining area. The minimum number of lines is 3. The positive normal of the area is controlled by the direction of *L1* using the right-hand rule. A negative value of *L1* reverses the normal direction. If *L1* = ALL, use all selected lines with *L2* defining the normal (*L3* to *L10* are ignored and *L2* defaults to the lowest numbered selected line). If *L1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *L1*.

## Notes

Lines may be input (once each) in any order and must form a simply connected closed curve. If the area is defined with more than four lines, the lines must also lie in the same plane or on a constant coordinate value in the active coordinate system (such as a plane or a cylinder).

---

## Note

Solid modeling in a toroidal coordinate system is not recommended. Areas may be redefined only if not yet attached to a volume.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Lines**

---

## ALIST, NA1, NA2, NINC, Lab

**Lists the defined areas.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NA1, NA2, NINC

List areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL (default), *NA2* and *NINC* are ignored and all selected areas [**ASEL**] are listed. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

### Lab

Determines what type of listing is used (one of the following):

#### (blank)

Prints information about all areas in the specified range.

#### HPT

Prints information about only those areas that contain hard points.

## Notes

An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **AATT** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer [**TYPE**, **MAT**, **REAL**, or **ESYS**] that was active during meshing (and will be reset to zero if the mesh is cleared). A "-1" in the "nodes" column indicates that the area has been meshed but there are no interior nodes. The area size is listed only if an **ASUM** command has been performed on the area.

## Menu Paths

**Utility Menu>List>Areas**

---

**ALLSEL**, *LabT*, *Entity*

**Selects all entities with a single command.**

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**LabT**

Type of selection to be made:

**ALL**

Selects all items of the specified entity type and all items of lower entity types (default).

**BELOW**

Selects all items directly associated with and below the selected items of the specified entity type.

**Entity**

Entity type on which selection is based:

**ALL**

All entity types (default).

**VOLU**

Volumes.

**AREA**

Areas.

**LINE**

Lines.

**KP**

Keypoints.

**ELEM**

Elements.

**NODE**

Nodes.

## Notes

**ALLSEL** is a convenience command that allows the user to select all items of a specified entity type or to select items associated with the selected items of a higher entity.

An entity hierarchy is used to decide what entities will be available in the selection process. This hierarchy from top to bottom is as follows: volumes, areas, lines, keypoints, elements, and nodes. The hierarchy may also be divided into two branches: the solid model and the finite element model. The label **ALL** selects items based on one branch only, while **BELOW** uses the entire entity hierarchy. For example, **ALLSEL,ALL,VOLU** selects all volumes, areas, lines, and keypoints in the data base. **ALLSEL,BELOW,AREA** selects all lines belonging to the selected areas; all keypoints belonging to those lines; all elements belonging to those areas, lines, and keypoints; and all nodes belonging to those elements.

The \$ character should not be used after the **ALLSEL** command.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking**  
**Utility Menu>Select>Everything**  
**Utility Menu>Select>Everything Below>Selected Areas**  
**Utility Menu>Select>Everything Below>Selected Elements**  
**Utility Menu>Select>Everything Below>Selected Keypoints**  
**Utility Menu>Select>Everything Below>Selected Lines**  
**Utility Menu>Select>Everything Below>Selected Volumes**

## ALPHAD, VALUE

Defines the mass matrix multiplier for damping.

[SOLUTION: Dynamic Options](#)

MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS

### VALUE

Mass matrix multiplier for damping.

## Notes

Defines the mass matrix multiplier,  $\alpha$ , for damping. One form of the viscous damping matrix [C] is given by  $\alpha[M] + \beta[K]$ , where [M] is the mass matrix and [K] is the stiffness matrix. Damping is not used in the static (**ANTYPE**,STATIC) or buckling (**ANTYPE**,BUCKLE) analyses.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Transient**  
**Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping**

## AMAP, AREA, KP1, KP2, KP3, KP4

Generates a 2-D mapped mesh based on specified area corners.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### AREA

Area number of area to be meshed. If  $AREA = P$ , graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

### KP1, KP2, KP3, KP4

Keypoints defining corners of the mapped mesh. Three or four corners may be specified, and may be input in any order.

## Notes

Only one area at a time can be meshed with this command. The program *internally* concatenates all lines between the specified keypoints, then meshes the area with all quadrilateral elements. If line divisions are set, the mesh will follow the rules for mapped meshing (see [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide*).

If the area being meshed has concatenated lines, the program will ask if those concatenations should be removed (in batch, the concatenations will automatically be removed). Nodes required for the generated elements are created and assigned the lowest available node numbers. If a mapped mesh is not possible due to mismatched line divisions or poor element shapes, the meshing operation is aborted.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>By Corners**

---

### **AMESH**, *NA1*, *NA2*, *NINC*

**Generates nodes and area elements within areas.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NA1*, *NA2*, *NINC*

Mesh areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and all selected areas [**ASEL**] are meshed. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

## Notes

Any undefined nodes required for the generated elements are created and assigned the lowest available numbers.

This command is also valid for [rezoning](#).

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh>Areas>Free**

**Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>3 or 4 sided**

**Main Menu>Preprocessor>Meshing>Mesh>Areas>Target Surf**

---



**/AN3D**, *Kywrđ*, *KEY***Specifies 3-D annotation functions**

GRAPHICS: Annotation

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

If *Kywrđ* = ANUM, the type of annotation is defined, along with its location. The command format is **/AN3D**,ANUM,NUM,TYPE,XHOT,YHOT,ZHOT.

**NUM**

Unique number assigned as each annotation is applied to a model. These numbers are applied sequentially, although when an annotation entity is deleted, its number is reassigned.

**TYPE**

Annotation internal type number (101 = text, 102 = line, 103 = point, 104 = area, 105 = arrow, 106 = symbol, 108 = bitmap).

**XHOT, YHOT, ZHOT**

X, Y, Z coordinates for hot spot location.

If *Kywrđ* = BITM, the annotation is a bitmap. The command format is **/AN3D**,BITM,TYPE,X,Y,Z.

**TYPE**

An integer value between 1 and 99, indicating a texture or bitmap. Numbers 1 through 40 correspond to existing ANSYS textures (see **/TXTRE**). Numbers 51 through 99 correspond to textures defined using the File option of the **/TXTRE** command. You can use this capability to override the predefined logo, clamp and arrow files available from the GUI dialog box (numbers 51 through 57). Numbers 41 through 50 are reserved.

**X, Y, Z**

X, Y, Z coordinates for the lower left corner of the bitmap.

If *Kywrđ* = TEXT, an annotation text string is created. The command format is **/AN3D**,TEXT,X,Y,Z,Text\_String.

**X, Y, Z**

X, Y, Z coordinate location for text string.

**Text\_String**

Text string to be applied as annotation.

If *Kywrđ* = LINE, an annotation line is created. The command format is **/AN3D**,LINE,X1,Y1,Z1,X2,Y2,Z2.

**X1, Y1, Z1**

X, Y, Z coordinates for beginning of line.

**X2, Y2, Z2**

X, Y, Z coordinates for end of line.

If *Kywrđ* = POINT, a dot will be drawn in space (this option is not available from the GUI). The command format is **/AN3D**,POINT,X,Y,Z.

**X, Y, Z**

X, Y, Z coordinates for point.

If *Kywrđ* = AREA, a polygonal area with *n* vertices will be drawn. The command format is **/AN3D,AREA,NVERT,Xn,Yn,Zn**.

**NVERT**

The number of vertices (*n*) for the polygon. Your Polygon can have between 3 and 9 vertices.

**Xn, Yn, Zn**

X, Y, Z coordinate location for vertex *n*.

If *Kywrđ* = ARROW, an annotation arrow is created. The command format is **/AN3D,ARROW,SIZE,X1,Y1,Z1,X2,Y2,Z2**.

**SIZE**

Symbol size multiplier (0.1 to 20.0, default = 1.0) for the head of the arrow.

**X1, Y1, Z1**

X, Y, Z coordinates for the location of the tail.

**X2, Y2, Z2**

X, Y, Z coordinate for the location of the tip.

If *Kywrđ* = SYMBOL, an annotation symbol is created. The command format is **/AN3D,SYMBOL,TYPE,X,Y,Z,SIZE**.

**TYPE**

The symbol type (1 = CAP, 2 = TEE, 3 = CIRCLE, 4 = TRIANGLE, 5 = STAR).

**X, Y, Z**

X, Y, Z coordinate location for the symbol.

**SIZE**

Size multiplier for the symbol (0.1 to 20.0, default = 1.0)

## Notes

Because 3-D annotation is applied in relation to the XYZ coordinates of the anchor, you can transform your model, and the annotation will maintain the spatial relationship with the model. This works within reason, and there are instances where changing the perspective or the size of the model will change the apparent relationship between the annotation and the model.

The overall 3-D dimensions of your model are defined by a bounding box. If portions of your model's bounding box lie outside of the visible area of your graphics window (if you are zoomed in on a specific area of your model), it can affect the placement of your 3-D annotations. Zooming out will usually overcome this problem.

3-D annotation is valid for the Cartesian (**CSYS,0**) coordinate system only. If you want to annotate a model you created in another coordinate system, use 2-D annotation (note that 2-D annotations do not remain anchored for dynamic rotations or transformations).

When you apply user defined bitmaps, the size of the annotation can vary. Use the options menu of the 3-D annotation widget to adjust the size and placement of your bitmaps.

You cannot use the "!" and "\$" characters in ANSYS text annotation.

The GUI generates this command during 3-D annotation operations and inserts the command into the log file (Jobname.LOG). You should NOT type this command directly during an ANSYS session (although the command can be included in an input file for batch input or for use with the **/INPUT** command).

## Menu Paths

Utility Menu>PlotCtrls>Annotate>Create 3D Annotation

---

**ANCNTR**, *NFRAM*, *DELAY*, *NCYCL*

Produces an animated sequence of a contoured deformed shape.

POST1: Animation

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NFRAM*

Number of frames captures (defaults to 5).

### *DELAY*

Time delay during animation (defaults to 0.1 seconds).

### *NCYCL*

Number of animation cycles (defaults to 5). Available in non-UI mode only.

## Notes

**ANCNTR** involves an ANSYS macro which produces an animation of a contoured deformed shape of the last plot action command. This command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANCNTR**, you can replay the animated sequence by issuing the **ANIM** command.

The command functions only in the postprocessor.

## Menu Paths

Utility Menu>PlotCtrls>Animate>Deformed Results

---

**ANCUT**, *NFRAM*, *DELAY*, *NCYCL*, *QOFF*, *KTOP*, *TOPOFF*, *NODE1*, *NODE2*, *NODE3*

Produces an animated sequence of Q-slices.

POST1: Animation

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NFRAM*

Number of frames captures (defaults to 5).

### *DELAY*

Time delay during animation (defaults to 0.1 seconds).

### *NCYCL*

Number of animation cycles (defaults to 5). Available in non-UI mode only.

**QOFF**

Q-slice working plane increment (defaults to .1 half screens).

**KTOP**

Topological effect on or off (YES or NO; default is NO).

**TOPOFF**

Topological offset (default is .1 half screens).

**NODE1**

Node 1 for start of the Q-slice.

**NODE2**

Node 2 for direction of the Q-slice.

**NODE3**

Node 3 for plane of the Q-slice.

## Notes

**ANCUT** involves an ANSYS macro which produces an animation of Q-slices of the last plot action command. This command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANCUT**, you can replay the animated sequence by issuing the **ANIM** command.

The command functions only in the postprocessor.

## Menu Paths

**Utility Menu>PlotCtrls>Animate>Q-Slice Contours**  
**Utility Menu>PlotCtrls>Animate>Q-Slice Vectors**

---

## **ANCYC**, *NUMFRAMES*, *KCYCL*, *DELAY*

**Applies a traveling wave animation to graphics data in a modal cyclic symmetry analysis.**

[POST1: Animation](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**NUMFRAMES**

The number of plot frames for the animation. Valid values range from 5 through 36. The default is 18. A low value (because it specifies fewer graphical frames) produces a rougher animation but loads faster. A high value produces a smoother animation but requires more time to load.

**KCYCL**

The animation mode:

**0**

Discontinuous animation cycle (forward-reset-forward). This option is the default.

**1**

Continuous animation cycle (forward-reverse-forward).

**DELAY**

The time delay (in seconds) between animation frames. Valid values range from 0.1 through 1.0. The default is 0.1 seconds, which produces a seemingly real-time animation. A higher value produces a slower animation.

## Command Default

The default **ANCYC** command (issuing the command with no arguments) specifies these implicit argument values: **ANCYC**, 18, 0, 0.1

## Notes

The **ANCYC** command is valid in a [modal cyclic symmetry analysis](#) only.

The command animates the cyclic symmetry mode shape plot in the General Post Processor (**/POST1**). When you issue a nodal- or element-results plot command (for example, **PLNSOL**, **PLESOL**, or **PLDISP**) and then issue the **ANCYC** command, ANSYS applies a traveling wave animation to the mode shape plot.

Each frame of the animation is created by expanding the cyclic symmetry mode shape at increasing phase angles (via the **/CYCEXPAND** command) starting at zero in equal increments over 360°. The phase-angle increment is  $360 / \text{NUMFRAMES}$ .

The animation display shows the traveling wave of the result quantity being plotted. The traveling wave animation is applicable only to nodal diameters (harmonic indices) greater than 0 and less than  $N / 2$  (where  $N$  is the number of cyclic sectors in the model).

For more information, see [Applying a Traveling Wave Animation to the Cyclic Model](#) in the *Advanced Analysis Techniques Guide*.

## Menu Paths

**Utility Menu > PlotCtrls > Animate > Cyc Traveling Wave**

---

**ANDATA**, *DELAY*, *NCYCL*, *RSLTDAT*, *MIN*, *MAX*, *INCR*, *FRCLST*, *AUTOCONT*, *--*, *AUTOCNTR*

**Produces a sequential contour animation over a range of results data.**

[POST1: Animation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **DELAY**

Time delay during animation (defaults to 0.5 seconds).

### **NCYCL**

Number of animation cycles (defaults to 5). Available in non-UI mode only.

### **RSLTDAT**

The type of results data to be used for the animation sequence. This can be:

- 0**  
Current load step data (default).
- 1**  
Range of load step data.
- 2**  
Range of results data.

### **MIN**

The range minimum value. If left blank or 0, defaults to the first data point.

**MAX**

The range maximum value. If left blank or 0, defaults to the last data point.

**INCR**

The increment between result data (defaults to 1).

**FRCLST**

Key to force the last sub step in a selected load step to be included in the animation (defaults to 0).

**AUTOCONT**

A value of 1 enables automatic scaling of contour values based on the overall subset range of values. The default value is 0 (no automatic scaling).

--

Unused field.

**AUTOCNTR**

A value of 1 disables automatic centering of displaced plots. The default value is 0 (allow automatic centering).

## Notes

The **ANDATA** command operates only on graphic display platforms supporting the **/SEG** command. It uses an ANSYS macro to produce an animation based on the last plot action command (for example, **PLDISP**).

The results file must have more than one set of results.

The **ANDATA** command implicitly issues **/DSCALE**, 1 for default displacement scaling. Large displacements may not give good results.

This command functions only in the postprocessor.

## Menu Paths

**Main Menu>Drop Test>Animate Results**  
**Utility Menu>PlotCtrls>Animate>Animate Over Results**

---

## **ANDSCL**, *NFRAM*, *DELAY*, *NCYCL*

**Produces an animated sequence of a deformed shape.**

[POST1: Animation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NFRAM**

Number of frames captured (defaults to 5).

**DELAY**

Time delay during animation (defaults to 0.1 seconds).

**NCYCL**

Number of animation cycles (defaults to 5). Available in non-UI mode only.

## Notes

**ANDSCL** involves an ANSYS macro which produces an animation of displacement of the last plot action command (for example, **PLDISP**). This command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANDSCL**, you can replay the animated sequence by issuing the **ANIM** command.

The command functions only in the postprocessor.

## Menu Paths

Utility Menu>PlotCtrls>Animate>Deformed Shape

---

**ANDYNA**, *DELAY*, *NCYCL*, *START*, *END*, *INC*, *AUTOCONTOURKEY*

**Produces an animated sequence of contour values through substeps.**

POST1: Animation

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *DELAY*

Time delay during animation (defaults to 0.1 seconds).

### *NCYCL*

Number of animation cycles (defaults to 5). Available in non-UI mode only.

### *START*

Number of the starting substep (defaults to 1).

### *END*

Number of the ending substep (defaults to the maximum substep).

### *INC*

Increment between substeps (defaults to 1).

### *AUTOCONTOURKEY*

Auto-scales contour values, based on the overall subset range of values (defaults to 0, no auto-scaling).

## Notes

**ANDYNA** involves an ANSYS macro which produces an animation of contour values through all the substeps of the last plot action command. This command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANDYNA**, you can replay the animated sequence by issuing the **ANIM** command.

The command functions only in the postprocessor.

## Menu Paths

Utility Menu>PlotCtrls>Animate>Dynamic Results

---

**/ANFILE**, *LAB*, *Fname*, *Ext*, --**Saves or resumes an animation sequence to or from a file.**

POST1: Animation

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***LAB***

Label type.

**SAVE**

Save the current animation to a file.

**RESUME**

Resume an animation from a file.

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to ANIM if *Fname* is blank.

--

Unused field.

## Notes

This command saves an animation to a file from local terminal segments or resumes an animation from a file to local terminal segments. See the **/SEG** command for details on segment storage. See the **ANCNTR** macro for a convenient method of storing graphics frames in terminal memory segments. This command is device dependent and is valid in any processor.

## Menu Paths

Utility Menu&gt;PlotCtrls&gt;Animate&gt;Restore Animation

Utility Menu&gt;PlotCtrls&gt;Animate&gt;Save Animation

---

**ANFLOW**, *NFRAM*, *DELAY*, *NCYCL*, *TIME*, *SPACING*, *SIZE*, *LENGTH***Produces an animated sequence of particle flow in a flowing fluid or a charged particle traveling in an electric or magnetic field.**

POST1: Animation

MP &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; FL EM &lt;&gt; &lt;&gt; PP &lt;&gt; EME &lt;&gt;

***NFRAM***

Number of frames captured (defaults to 5).



**DELAY**

Time delay during animation (defaults to 0.1 seconds).

**NCYCL**

Number of animation cycles (defaults to 5). Non-UI mode only.

**TIME**

Total Trace Time (seconds) (defaults to 0, which is the full flow trace).

**SPACING**

Particle spacing in seconds (defaults to 0).

**SIZE**

Particle size (defaults to 0, which is a line).

**LENGTH**

Particle length fraction (defaults to .1).

## Notes

**ANFLOW** invokes an ANSYS macro which produces an animation of particle flow in a flowing fluid or charged particle motion in an electric or magnetic field by the last plot action command (i.e., **PLTRAC**). This command is only operational on graphic display platforms supporting the **/SEG** command. After executing **ANFLOW**, you can replay the animated sequence by issuing the **ANIM** command. This command is functional only in the Postprocessor.

The **TIME** option lets you set the time interval of forward travel for the trace. The **SPACING** option is used to define the particle spacing in seconds from adjacent particles in the stream line. The **SIZE** variable sets the radius of the particle. The **LENGTH** variable is used to define the particle length fraction. By default, the **LENGTH** is set to .1, which means the particle occupies 10% of the flow region and the other 90% is a color-code line. The **SPACING** and **LENGTH** variables only make sense when the **SIZE** variable is nonzero (i.e., the particle is bigger than the line).

## Menu Paths

Utility Menu>PlotCtrls>Animate>Particle Flow

**/ANGLE**, *WN*, *THETA*, *Axis*, *KINCR*

**Rotates the display about an axis.**

[GRAPHICS: Views](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WN**

Window number (or ALL) to which command applies (defaults to 1).

**THETA**

Angle (degrees) for changing display orientation (positive, counterclockwise about specified axis).

**Axis**

Rotation axis: XS, YS, or ZS (default) for the screen axes; XM, YM, or ZM for the global Cartesian model axes. ZS is normal to the screen; all axes pass through the focus point.

**KINCR**

Cumulative rotation key:

**0**

Do not use cumulative successive rotations.

**1**

Use cumulative rotations. Rotations are relative to the previous rotation. View settings (**/VIEW**) are recalculated.

## Notes

Default orientation is YS vertical. When the **/XFRM** command is set for rotation about two points, or for entities, the **/ANGLE** command is functional only for *Axis* = ZS or ZM and *KINCR* = 1.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Utility Menu>PlotCtrls>View Settings>Angle of Rotation

---

## **ANHARM**, *NFRAM*, *DELAY*, *NCYCL*, *NPERIOD*

**Produces a time-transient animated sequence of time-harmonic results or complex mode shapes.**

[POST1: Animation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NFRAM**

Number of frame captures per cycle. Defaults to 12.

**DELAY**

Time delay during animation. Defaults to 0.1 seconds.

**NCYCL**

Number of animation cycles. Defaults to 5. Not available in the GUI.

**NDECAY**

Controls the style of complex mode animation. Complex modal solutions can exhibit oscillation decay or amplification. Defaults to 1, animating the decay or growth of only the first period. Issue -1 to cause the animation to be performed without decay or growth. Animation is displayed as a typical modal result.

## Notes

**ANHARM** invokes an ANSYS macro which produces a time-transient animation of:

- Time-harmonic results in the case of a harmonic analysis (**ANTYPE,HARMIC**)
- Complex mode shapes in the case of a modal analysis (**ANTYPE,MODAL**).

In both cases, the results are those of the last plot action (for example, **PLNSOL,B,SUM**).

The animation converts the complex solution variables (real and imaginary sets) into time varying results over one period. For example, if  $NFRAM = 12$ , then the frame captures are in increments of 30 degree phase angles.

A second set of  $NFRAM$  frames will be generated for damped eigenmodes from complex eigensolvers to visualize any exponential decay or growth of the oscillations. The second set generated will display frames from the period number specified by  $NPERIOD$ .

## Menu Paths

**Utility Menu>PlotCtrls>Animate>Animate Over Time**  
**Utility Menu>PlotCtrls>Animate>Time-harmonic**

**ANIM**,  $NCYCL$ ,  $KCYCL$ ,  $DELAY$

**Displays graphics data in animated form.**

POST1: Animation

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NCYCL***

Number of cycles associated with the animation (defaults to 5 in non-GUI mode only)

### ***KCYCL***

Animation mode:

**0**

Continuous animation cycle (forward-reverse-forward-etc.) (default).

**1**

Discontinuous animation cycle (forward-reset-forward-etc.).

### ***DELAY***

Time delay (seconds) between animation frames (defaults to 0.1 seconds).

## Notes

Displays graphics data stored in local terminal segments in animated form. See the **/SEG** command for details on segment storage. See the **ANCNTR** macro for a convenient method of storing graphics frames in terminal memory segments. This command is device-dependent. You should not resize the graphic while animation is in progress; doing so can result in distorted plots.

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Animate>Replay Animation**  
**Utility Menu>PlotCtrls>Animate>Restore Animation**

**ANISOS**, *NFRAM*, *DELAY*, *NCYCL*

**Produces an animated sequence of an isosurface.**

[POST1: Animation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NFRAM***

Number of frames captures (defaults to 9).

***DELAY***

Time delay during animation (defaults to 0.1 seconds).

***NCYCL***

Number of animation cycles (defaults to 5). Available in non-UI mode only.

## Notes

**ANISOS** involves an ANSYS macro which produces an animation of an isosurface of the last plot action command (for example, **PLNSOL,S,EQV**). The **ANISOS** command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANISOS**, you can replay the animated sequence by issuing the **ANIM** command.

This command functions only in the postprocessor.

## Menu Paths

**Utility Menu>PlotCtrls>Animate>Isosurfaces**

---

**ANMODE**, *NFRAM*, *DELAY*, *NCYCL*, *KACCEL*

**Produces an animated sequence of a mode shape.**

[POST1: Animation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NFRAM***

Number of frames captures (defaults to 5).

***DELAY***

Time delay during animation (defaults to 0.1 seconds).

***NCYCL***

Number of animation cycles (defaults to 5). Available in non-UI mode only.

***KACCEL***

Acceleration type:

**0**  
Linear acceleration.

**1**  
Sinusoidal acceleration.

## Notes

**ANMODE** involves an ANSYS macro which produces an animation of mode shape of the last plot action command (for example, **PLDISP**). The **ANMODE** command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANMODE**, you can replay the animated sequence by issuing the **ANIM** command.

This command functions only in the postprocessor.

## Menu Paths

Utility Menu>PlotCtrls>Animate>Mode Shape

**ANMRES**, *DELAY*, *MIN*, *MAX*, *INC*, *AUTOCTRKY*, *FREQ*, *EXT*

**Performs animation of results over multiple results files in an explicit dynamic structural analysis or fluid flow analysis with remeshing.**

POST1: Animation

MP ME ST <> <> <> <> FL <> <> DY PP <> EME MFS

### *DELAY*

Time delay during animation (default = 0.5 seconds).

### *MIN*

Minimum results file number to animate. Default = 1 (for Jobname .RS01).

### *MAX*

Maximum results file number to animate. Defaults to the highest numbered results file, Jobname .RSnn.

### *INC*

Increment between results file numbers. Default = 1.

### *AUTOCTRKY*

Automatic contour scaling option.

**0**

No auto-scaling (default).

**1**

Auto-scaling on.

If you activate automatic contour scaling, ANSYS considers only the minimum and maximum value of the result item from the first results file.

### *FREQ*

Results frequency key.

**0 or 1**

Animate every results set in each Jobname .EXT file (default).

**2**

Animate every other results set in each Jobname .EXT file.

**n**

Animate every *n*th results set in each Jobname .EXT file.

**EXT**

Extension of result files

**'rfl'**

Animate Jobname.rflnn

**'rs'**

Animate Jobname.rsnn. Default = 'rs'.

## Notes

**ANMRES** invokes an ANSYS macro that performs animation across multiple results files (Jobname.EXT, Jobname.EXT, etc.) produced by an explicit dynamic structural analysis or fluid flow analysis with remeshing. Multiple results files typically occur when adaptive meshing is used in an explicit dynamic structural analysis or fluid flow analysis with remeshing. Each results file must have more than one set of results. **ANMRES** cannot be used for multiple results files that are caused by file splitting.

**ANMRES** animates results from files having the currently specified jobname (Jobname.EXT - Jobname.EXT). To change the current jobname, use the **/FILENAME** command. The animation is based on the last plot command (e.g., **PLDISP**).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Drop Test>Animate Results**

**Utility Menu>PlotCtrls>Animate>Animate Over Results**

---

**/ANNOT**, Lab, VAL1, VAL2

**Activates graphics for annotating displays (GUI).**

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Lab**

Annotation control key:

**OFF**

Turns off annotation for each subsequent display (default).

**ON**

Turns on annotation for each subsequent display.

**DELE**

Deletes all annotation.

**SAVE**

Saves annotation on a file. Use VAL1 for file name (defaults to Jobname) and VAL2 for the extension (defaults to ANO).

**SCALE**

Sets annotation scale factor (direct input only). Use VAL1 for value (0.1 to 10.0) (defaults to 1.0).

**XORIG**

Sets the annotation x origin (direct input only). Use VAL1 for value (-3.0 to 3.0).

**YORIG**

Sets annotation y origin (direct input only). Use *VAL1* for value (-3.0 to 3.0).

**SNAP**

Sets annotation snap (menu button input only). Use *VAL1* for value (0.002 to 0.2) (defaults to 0.002).

**STAT**

Displays current annotation status.

**DEFA**

Sets annotation specifications to the default values.

**REFR**

Redisplays annotation graphics.

**TMOD**

Sets the annotation text mode. If *VAL1* = 1, annotation text will be drawn in scalable bitmap fonts (default). If *VAL1* = 0, annotation text will be drawn with stroke text.

**VAL1**

Value (or file name) as noted with label above.

**VAL2**

Value (or file name extension) as noted with label above.

**Notes**

This is a command generated by the GUI and will appear in the log file (*Jobname.LOG*) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

You cannot use the “!” and “\$” characters in ANSYS text annotation.

**/ANNOT** activates annotation graphics for adding annotation to displays. Commands representing the annotation instructions are automatically created by the annotation functions in the GUI and written to *Jobname.LOG*. The annotation commands are **/ANNOT**, **/ANUM**, **/TLABEL**, **/LINE**, **/LARC**, **/LSYMBOL**, **/POLYGON**, **/PMORE**, **/PCIRCLE**, **/PWEDGE**, **/TSPEC**, **/LSPEC**, and **/PSPEC**. Annotation graphics are relative to the full Graphics Window and are not affected by ANSYS window-specific commands (**/WINDOW**, **/VIEW**, etc.).

This command is valid in any processor.

**Menu Paths**

**Utility Menu>PlotCtrls>Annotation>Create Annotation**

**ANORM**, *ANUM*, *NOEFLIP*

**Reorients area normals.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**ANUM**

Area number having the normal direction that the reoriented areas are to match.

**NOEFLIP**

Indicates whether you want to change the normal direction of the existing elements on the reoriented area(s) so that they are consistent with each area's new normal direction.

**0**

Make the normal direction of existing elements on the reoriented area(s) consistent with each area's new normal direction (default).

**1**

Do not change the normal direction of existing elements on the reoriented area(s).

## Notes

Reorients areas so that their normals are consistent with that of a specified area.

If any of the areas have inner loops, the **ANORM** command will consider the inner loops when it reorients the area normals.

You cannot use the **ANORM** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

See [Revising Your Model](#) of the *Modeling and Meshing Guide* for more information.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Areas>Area Normals**

---

**ANSOL**, *NVAR*, *NODE*, *Item*, *Comp*, *Name*, *Mat*, *Real*, *Ename*

**Specifies averaged nodal data to be stored from the results file in the solution coordinate system.**

[POST26: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NVAR***

Arbitrary reference number assigned to this variable (2 to *NV* [**NUMVAR**]). Overwrites any existing results for this variable.

***NODE***

Node number for which data are to be stored.

***Item***

Label identifying the item. General item labels are shown in [Table 141: ANSOL - General Item and Component Labels](#) (p. 134) below. Some items also require a component label.

***Comp***

Component of the item (if required). General component labels are shown in [Table 141: ANSOL - General Item and Component Labels](#) (p. 134) below.



**Name**

Thirty-two character name for identifying the item on the printout and displays. Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

**Mat**

The material number. Average will be computed based on the subset of elements with the specified material number. DEFAULT: Use all elements in the active set unless *Real* and/or *Ename* is specified.

**Real**

The real number. Average will be computed based on the subset of elements with the specified real number. DEFAULT: Use all elements in the active set unless *Mat* and/or *Ename* is specified.

**Ename**

The element type name. Average will be computed based on the subset of elements with the specified element type name. DEFAULT: Use all elements in the active set unless *Mat* and/or *Real* is specified.

**Notes**

Valid item and component labels for averaged nodal results are listed in [Table 141: ANSOL - General Item and Component Labels](#) (p. 134), below.

All element nodal quantities are obtained in **RSYS**, Solu and then averaged.

The **ANSOL** command defines averaged nodal results data to be stored from a results file [**FILE**]. Not all items are valid for all nodes. See the input and output summary tables of the [Element Reference](#) of each element that is attached to the node for the available items.

**COORDINATE SYSTEMS:** All element nodal results used by **ANSOL** for averaging are in the element coordinate system, except for layered elements. Layered element results are in the layer coordinate system. You can further specify the element nodal results, for some elements, with the **SHELL**, **LAYERP26**, and **FORCE** commands.

**ANSOL** does not transform results from **RSYS**, SOLU to other coordinate systems. Verify that all elements attached to the subject node have the same coordinate system before using **ANSOL**.

**SHELL ELEMENTS:** The default shell element coordinate system is based on node ordering. For shell elements the adjacent elements could have a different **RSYS**, SOLU, making the resultant averaged data inconsistent. A note to this effect is issued when **ANSOL** is used in models containing shell elements. Ensure that consistent coordinate systems are active for all associated elements used by the **ANSOL** command.

**DERIVED QUANTITIES:** Some of the result items supported by **ANSOL** (see [Table 141: ANSOL - General Item and Component Labels](#) (p. 134)) are derived from the component quantities. Use **AVPRIN** to specify the principal and vector sum quantity averaging methods.

**DEFAULT:** If *Mat*, *Real* , and *Ename* are not specified, all of the elements attached to the node will be considered. When a material ID, real constant ID, or element type discontinuity is detected at a node, a note

is issued. For example, in a FSI analysis, a **FLUID30** element at the structure interface would be considered. But since it contains no *SX* result, it will not be used during **STORE** operations.

**Table 141 ANSOL - General Item and Component Labels**

**General Item and Component Labels ANSOL,**  
*NVAR, ELEM, NODE, Item, Comp, Name, Mat, Real, Ename*

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1,2,3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
CONT	STAT1	Contact status.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress
"	STOT	Contact total stress (pressure plus friction)
"	SLIDE	Contact sliding distance

**General Item and Component Labels ANSOL,***NVAR, ELEM, NODE, Item, Comp, Name, Mat, Real, Ename*

Item	Comp	Description
"	GAP	Contact gap distance
"	FLUX	Total heat flux at contact surface
"	CNOS	Total number of contact status changes during substep.
"	FPRS	Fluid penetration pressure
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).

- For more information on the meaning of contact status and its possible values, see [Reviewing Results in POST1](#) in the *Contact Technology Guide*.

**Menu Paths****Main Menu>TimeHist Postpro>Variable Viewer****ANSTOAQWA, Fname, VertAxis, Gc, Rho, HWL, DiffKey, SymxKey, SymyKey****Creates an AQWA-LINE input file from the current ANSYS model.**

SESSION: Files

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

**Fname**

AQWA file name. Defaults to Jobname.

**VertAxis**

Axis in the vertical direction:

**Y (or 2)**

Global Y axis.

**Z (or 3)**

Global Z axis (default).

**Gc**

Gravitational acceleration. Defaults to 9.81.

**Rho**

Density of water. Defaults to 1025.0.

**HWL**

Waterline height in model coordinates. Defaults to 0.0.

**DiffKey**

Diffraction model key:

- 0**  
Create a non-diffracting AQWA model.
- 1**  
Create a diffracting AQWA model (default).

**SymxKey**

Key indicating if model is symmetric about the global XZ plane:

- 0**  
No symmetry about XZ plane (default).
- 1**  
Use symmetry about XZ plane. Only include (or select) half the model.

**SymyKey**

Key indicating if model is symmetric about the global YZ plane:

- 0**  
No symmetry about YZ plane (default).
- 1**  
Use symmetry about YZ plane. Only include (or select) half the model.

## Notes

This command creates the input file *Fname*.aqwa for the ANSYS AQWA Multi-Body Hydrodynamics System for diffraction analysis in AQWA-LINE from the model currently in the database, based on the currently selected set of elements. The selected set must only include the hull envelope; no internal structure should be selected.

There should be a line of nodes defined at the waterline. Only those elements that are entirely below the waterline will be specified as diffracting. If there are no waterline nodes, there will be no diffracting elements at the waterline, which will severely reduce the accuracy of the diffraction analysis.

The translator maps PLANE42, SHELL41, SHELL63, and SHELL181 elements to PANELs, and maps PIPE16 and PIPE59 elements to TUBEs. It does not recognize any other element types. Any material or geometric properties can be used for the shell elements, as AQWA does not need any properties at all and the command does not use them. All the shell elements below the water must have their normals pointing outward.

TUBE elements in AQWA have material density, outside diameter, wall thickness, added mass, and drag coefficients, so appropriate properties should be used in the ANSYS model. PIPE59 elements can have added mass and damping coefficients; these will be written to the file. The ANSYS program uses the inertia coefficient  $C_M$ , whereas AQWA uses the added mass coefficient  $C_A$ , where  $C_M = (1 + C_A)$ . This correction is made automatically.

Documentation for legacy elements PIPE16, PLANE42, PIPE59, and SHELL63 appears in the <a href="#">Feature Archive</a> .
---

In AQWA the vertical axis is always the Z-axis. The command can convert a model built with either the Y or Z-axis vertical, but the X-axis must be horizontal and should preferably be along the fore/aft axis of the vessel. If the structure is symmetric and you wish to use the symmetry options, you must only select one

half or one quarter of the model, as appropriate. If you model a complete vessel and specify X symmetry, the AQWA model will contain two sets of coincident elements.

If you are working from a model created for a structural analysis, it will probably be necessary to remesh the model as the structural mesh is most likely finer than needed for a diffraction analysis.

If you enter this command interactively (with the GUI active) and no data is provided for the command options, you will be prompted for their values.

You must verify the completeness and accuracy of the data written.

## AQWA-LINE Notes

The file will specify restart stages 1-2 only. It has no options except REST, so AQWA may fail if any of the elements are badly shaped.

The total mass is obtained by integrating over the wetted surface area and adding the TUBE masses, so it should be reasonably accurate. However, the integration used is not as accurate as that in AQWA, so there may be a small difference between the weight and buoyancy, particularly if tubes represent a large portion of the model.

The position of the CG is unknown. A point mass is placed at the water-line above the CB, but you should change this to the correct position.

The moments of inertia are estimated based on the overall dimensions of the model and using standard formulae for a ship. You should change these to the correct values.

The maximum frequency is calculated from the maximum side length of the underwater elements. The range of frequencies runs from 0.1 rad/s to the calculated maximum, in steps of 0.1 rad/s.

The directions are in steps of 15° over a range that is determined by the symmetry you have specified, in accordance with the requirements of AQWA.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## ANSTOASAS, *Fname*, *KEY*

**Creates an ASAS input file from the current ANSYS model.**

SESSION: Files

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Fname*

ASAS file name. Defaults to Jobname.

### *KEY*

Key indicating type of file to produce:

**0**

ASAS file for use by ANSYS AQWA (no loads written). Creates the file *Fname*.asas.

- 1 ASAS file (all data written, including loads). Creates the file *Fname.asas*.
- 2 ASAS(NL) file. Creates the file *Fname.asnl*.

## Notes

This command creates an input file for the ANSYS ASAS Finite Element Analysis System from the model and loads currently in the database, based on the currently selected set of elements. Most common structural element types are written, as well as sections (or real constants), materials, boundary conditions and loads, and solution and load step options.

### Data Written

The following data is written:

- [Solution control options](#)
- Nodes
- [Elements](#)
- [Material data](#)
- [Geometry data](#)
- [Section data](#)
- ANSYS element components (ASAS sets)
- [Boundary conditions](#)
- [Loads](#)
- Added mass (via [MASS21](#) element)

Details are provided in the following sections.

Not all data is written. You must verify the completeness and accuracy of the data. Only loading at the current step is transferred; hence, no load step history is captured.

### Solution Control Options

The ASAS project name is defined as "ANSYS".

The solution control options are converted as follows:

ANSYS Analysis Type	ASAS(L) / AQWA-WAVE Option	ASAS(NL) Option
Static (0)	JOB NEW LINE	JOB STAT
Buckling (1)	Not applicable	JOB STAT SPIT KGEOM
Modal (2)	JOB NEW FREQ	JOB STAT SPIT KGEOM

ANSYS Analysis Type	ASAS(L) / AQWA-WAVE Option	ASAS(NL) Option
Transient (4)	Not applicable	JOB TRAN
Remaining analysis types	JOB NEW LINE	JOB STAT

For conversion to ASAS(NL), the large displacement option is set based on **NLGEOM**, final load solution time is set based on **TIME**, and sub-step times are set based on **DELTIM** or **NSUBST** (assuming constant step size).

### Element Data

If you intend to use the data only with AQWA-WAVE, only the elements that form the wetted surface are required. Selecting these elements before invoking the **ANSTOASAS** command will improve performance. In order for AQWA-WAVE to identify the direction of the wave loading, all elements must be defined by nodes in a clockwise direction. For further information, refer to the AQWA-WAVE manual.

The element types are converted as follows:

Element Type	Supported Facilities	Notes
<b>COMBIN14</b> - Spring-Damper	SPR1 SPR2 if rotational spring FLA2 (ASAS(L) only) if nodes are not coincident and longitudinal spring	ASAS(L) does not support spring elements with non-coincident nodes. In this case, <b>COMBIN14</b> is converted to FLA2.
<b>MASS21</b> - Structural Mass	N/A	In ASAS, additional mass is not added as an element. Hence, if this element is included, equivalent ASAS lumped added mass information is written.
<b>PLANE42</b> - 2-D Structural Solid [1]	QUM4 TRM3 - if Triangular	-
<b>SOLID45</b> - 3-D Structural Solid [1]	BRK8 TET4 - if Tetrahedral BRK6 - if Prism	TET4 elements are only available in ASAS(L). Element is not converted if Pyramidal.
<b>SHELL61</b> - Axisymmetric-Harmonic Structural Shell	ASH2	ASH2 elements are only available in ASAS(L).
<b>SHELL63</b> - Elastic Shell [1]	QUS4 TBC3 - if Triangular	TBC3 elements are only available in ASAS(L).
<b>PLANE82</b> - 2-D 8-Node Structural Solid [1]	QUM8 TRM6 - if Triangular	-
<b>SOLID92</b> - 3-D 10-Node Tetrahedral Structural Solid [1]	TE10	-

Element Type	Supported Facilities	Notes
SOLID95 - 3-D 20-Node Structural Solid [1]	BR20 TE10 - if Tetrahedral BR15 - if Prism	TE10 elements are only available in ASAS(L). Element is not converted if Pyramidal.
LINK180 - 3-D Finite Strain Spar (or Truss)	FLA2	-
SHELL181 - 4-Node Finite Strain Shell	QUS4 TBC3 - if Triangular	TBC3 elements are only available in ASAS(L).
PLANE182 - 2-D 4-Node Structural Solid	QUM4 TRM3 - if Triangular	-
PLANE183 - 2-D 8-Node or 6-Node Structural Solid	QUM8 TRM6 - if Triangular	-
SOLID185 - 3-D 8-Node Structural Solid or Layered Solid	BRK8 TET4 - if Tetrahedral BRK6 - if Prism	TET4 elements are only available in ASAS(L). Element is not converted if Pyramidal.
SOLID186 - 3-D 20-Node Structural Solid or Layered Solid	BR20 TE10 - if Tetrahedral BR15 - if Prism	TE10 elements are only available in ASAS(L). Element is not converted if Pyramidal.
SOLID187 - 3-D 10-Node Tetrahedral Structural Solid	TE10	TE10 elements are only available in ASAS(L).
BEAM188 - 3-D Linear Finite Strain Beam	BM3D	-
BEAM189 - 3-D Quadratic Finite Strain Beam	TCBM - if ASAS(L) STF4 - if ASAS(NL)	Refer to geometry details for limitations for TCMB elements. An orientation node is needed for STF4 and must be specified.
SHELL208 - 2-Node Finite Strain Axisymmetric Shell	ASH2	ASH2 elements are only available in ASAS(L).
SHELL281 - 8-Node Finite Strain Shell	TCS8	-
PIPE288 - 3-D Linear Finite Strain Pipe	TUBE	-
PIPE289 - 3-D Quadratic Finite Strain Pipe	CURB	-
ELBOW290 - 3-D Quadratic Finite Strain Elbow	CURB - if ASAS(L) STF4 - if ASAS(NL)	-

1. Documentation for this legacy element type appears in the [Feature Archive](#).

## Material Data



Linear isotropic material conversion is supported for ASAS and ASAS(NL).

### Geometry Data

The following ASAS element geometry data is supported:

ASAS Element Type	Support Facilities	Notes
BM3D	Sections, orientation by 3rd node position.	Sections are always defined separately.
TUBE	Thickness and diameter defined, orientation by 3rd node or default local axes.	Using the default local axes will result in BETA being set to 90° (to ensure that the AS-AS local axes are the same as those in ANSYS).
BM2D	A, IZ & AY properties defined.	
TCBM	Uniform section properties supported.	TCBM does not support general local axis orientations. Hence, elements will only be correct if they lie in the global XY plane.
STF4	CTUB, RECT, and HREC ANSYS subtypes supported.	STF4 elements are only supported in ASAS(NL). Local y and z are 90° to the ANSYS definition.
FLA2	Uniform cross sectional area.	
SPR1 / SPR2	Stiffness (and also linear damping if ASAS(NL)) included.	
All non-beam elements	Constant element thickness.	

### Section Data

No user sections are generated if AQWA-WAVE data is selected.

The following sections are converted for ASAS and ASAS(NL):

ANSYS Section Type	ASAS Section Type	Notes
CTUB	TUB	Tubular section
I	FBI	Fabricated I beam
HREC	BOX	Fabricated box
All others	PRI	Prismatic section, only flexural properties defined.

### Boundary Conditions

The following boundary conditions are converted for ASAS and ASAS(NL):

ANSYS Boundary Condition	ASAS Boundary Condition	Notes
Nodal U* and ROT* constraints	SUPPRESSED freedoms: X, Y, Z, RX, RY, RZ	Skewed systems are not supported.
Nodal U* and ROT* imposed non-zero values	DISPLACED freedoms: X, Y, Z, RX, RY, RZ	Skewed systems are not supported.
CP and CE constraint equations	CONSTRaINT equation data	Skewed systems are not supported.

**Loads**

No user loading is generated if AQWA-WAVE data is selected. However, a load case (number 1000) is automatically defined to identify the wetted surface of the elements for use by AQWA-WAVE based on the normal surface loads applied to the solid or shell elements.

Pressure loads from SURF154 elements are converted to equivalent nodal loads for ASAS. For AQWA-WAVE, the SURF154 pressures are used to identify the wetted surface of the underlying elements. The following loads are converted for ASAS:

ANSYS Load Type	ASAS Load Type	Notes
SFE (PRES)	PRESSURE (no sub-types)	Element families supported: Solids, shells (excluding edge pressures), planes (edge pressures only), and axisymmetric shells.
SFE (PRES)	DISTRIBUted loading Shells - ML2 Beams - BL1 / BL2 Tubes - GL1 Curved beams - CB1	Element families supported: Shells (edge pressures only), Beams (includes tubes and curved beams).
F (F* and M*)	NODAL Load, in X, Y, Z, RX, RY, RZ	Skewed systems are not supported.
D (U* and ROT*)	PRESCRIBed displacements	
ACEL	BODY FORce	

**Menu Paths**

This command cannot be accessed from a menu.

**ANTIME**, *NFRAM*, *DELAY*, *NCYCL*, *AUTOCTRKY*, *RSLTDAT*, *MIN*, *MAX*

Produces a sequential contour animation over a range of time.

POST1: Animation

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NFRAM**

Number of frame captures (defaults to 5).

**DELAY**

Time delay during animation (defaults to 0.1 seconds).

**NCYCL**

Number of animation cycles (defaults to 5). Available in non-UI mode only.

**AUTOCNTRKY**

Auto-scales contour values, based on the overall subset range of values. The auto-scaling option defaults to 0, no auto-scaling.

**RSLTDAT**

The results data to be used for the animation sequence. This can be:

**0**

Current load step data (default).

**1**

Range of load step data.

**2**

Range of time data.

**MIN**

The range minimum value. If left blank defaults to the first data point.

**MAX**

The range maximum value. If left blank defaults to the last data point.

## Notes

The **ANTIME** command operates only on graphic display platforms supporting the **/SEG** command. It uses an ANSYS macro to produce an animation of contour values for the last plot action command (for example, **PLDISP**). After executing **ANTIME**, the **ANIM** command will replay the animated sequence.

This command functions only in the postprocessor.

## Menu Paths

**Utility Menu>PlotCtrls>Animate>Animate Over Time**  
**Utility Menu>PlotCtrls>Animate>Time-harmonic**

---

**ANTYPE**, *Antype*, *Status*, *LDSTEP*, *SUBSTEP*, *Action*

**Specifies the analysis type and restart status.**

[SOLUTION: Analysis Options](#)

[MP](#) [ME](#) [ST](#) [PR](#) [PRN](#) [DS](#) [DSS](#) [FL](#) [EM](#) [EH](#) [<>](#) [PP](#) [<>](#) [EME](#) [MFS](#)

[Product Restrictions](#)

**Antype**

Analysis type (defaults to the previously specified analysis type, or to STATIC if none specified):

**STATIC or 0**

Perform a static analysis. Valid for all degrees of freedom.

**BUCKLE or 1**

Perform a buckling analysis. Implies that a previous static solution was performed with prestress effects calculated [**PSTRES,ON**]. Valid for structural degrees of freedom only.

**MODAL or 2**

Perform a modal analysis. Valid for structural and fluid degrees of freedom.

**HARMIC or 3**

Perform a harmonic analysis. Valid for structural, fluid, magnetic, and electrical degrees of freedom.

**TRANS or 4**

Perform a transient analysis. Valid for all degrees of freedom.

**SUBSTR or 7**

Perform a substructure analysis. Valid for all degrees of freedom.

**SPECTR or 8**

Perform a spectrum analysis. Implies that a previous modal analysis was performed. Valid for structural degrees of freedom only.

**Status**

Specifies the status of the analysis (new or restart):

**NEW**

Specifies a new analysis (default). If NEW, the remaining fields on this command are ignored.

**RESTART**

Specifies a restart of a previous analysis. Valid only for structural static, structural transient (full or mode-superposition methods), and thermal analyses. For modal and substructure (backsubstitution method only) analyses, reuse the previous .MODE and .SUB files, respectively.

This option resumes the .rdb file created at the start of solution. If boundary conditions are deleted in solution (for example, after being used to create an initial velocity or to establish initial contact), they will need to be deleted again after issuing this command.

For modal analysis, reuses the existing modes extracted from the previous modal analysis. Valid only with *Antype* = MODAL with Block Lanczos, PCG Lanczos, or Supernode modal solver. The load vectors, residual vectors, and enforced motion vectors will be recalculated and stored.

**VTREST**

Specifies the restart of a previous VT Accelerator analysis. Valid only with *Antype* = STATIC, HARMIC, or TRANS.

**LDSTEP**

Specifies the load step at which a multiframe restart will begin.

For full transient and nonlinear static structural or thermal analyses, the default is the highest load step number found in the *Jobname.Rnnn* files for the current jobname in the current directory.

For mode-superposition transient analyses, the default is none.

**SUBSTEP**

Specifies the substep at which a multiframe restart will begin.

For full transient and nonlinear static structural or thermal analyses, the default is the highest substep number found for the specified *LDSTEP* in the *Jobname.Rnnn* files in the current directory.

For mode-superposition transient analyses, the default is none.

**Action**

Specifies the manner of a multiframe restart. Not used for traditional restarts.

**CONTINUE**

ANSYS will continue the analysis based on the specified *LDSTEP* and *SUBSTEP* (default). The current load step will be continued. If the end of the load step is encountered in the *.Rnnn* file, a new load step will be started. ANSYS will delete all *.Rnnn* files, or *.Mnnn* files for mode-superposition transient analyses, beyond the point of restart and will update the *.LDHI* file if a new load step is encountered.

**ENDSTEP**

At restart, force the specified load step (*LDSTEP*) to end at the specified substep (*SUBSTEP*), even though the end of the current load step has not been reached. At the end of the specified substep, all loadings will be scaled to the level of the current ending and stored in the *.LDHI* file. A run following this **ENDSTEP** will start a new load step. This feature allows you to change the load level in the middle of a load step. ANSYS will update the *.LDHI* file and delete all *.Rnnn* files, or *.Mnnn* files for mode-superposition transient analyses, beyond the point of **ENDSTEP**. The *.Rnnn* or *.Mnnn* file at the point of **ENDSTEP** will be rewritten to record the rescaled load level.

**RSTCREATE**

At restart, retrieve information to be written to the results file for the specified load step (*LDSTEP*) and substep (*SUBSTEP*). Be sure to use **OUTRES** to write the results to the results file. This action does not affect the *.LDHI* or *.Rnnn* files. Previous items stored in the results file at and beyond the point of **RSTCREATE** will be deleted. This option is not available for restart of a mode-superposition transient analysis.

**PERTURB**

At restart, a linear perturbation analysis (MODAL) will be performed for the specified load step (*LDSTEP*) and substep (*SUBSTEP*). This action does not affect the *.LDHI*, *.Rnnn*, or *.RST* files.

**Note**

For a linear perturbation analysis, you must set *Action* = **PERTURB**; otherwise, the existing restart files, such as the *.LDHI*, *.Rnnn*, or *.RST* file, may be modified by the linear perturbation analysis. Use the **PERTURB** command to indicate the desired analysis type (MODAL or BUCKLING).

**Command Default**

New static analysis.

**Notes**

The analysis type *Antype* cannot be changed if it is a restart run. Always save parameters before doing a restart. You can perform a multiframe restart only for structural static, structural transient (full or mode-superposition methods), and thermal analyses.

If you use **ANTYPE** to change your analysis type in the same SOLVE session, ANSYS issues the following note: "Some analysis options have been reset to their defaults. Please verify current settings or respecify as required." Typically, ANSYS resets commands such as **NLGEOM**, **SSTIF**, and **EQSLV** to their default values.

This command is also valid in PREP7.

## Product Restrictions

Command Option An-type	Available Products
STATIC	MP ME ST PR PRN DS DSS FL EM <> <> PP <> EME MFS
BUCKLE	MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS
MODAL	MP ME ST PR PRN DS DSS <> <> EH <> PP <> EME MFS
HARMONIC	MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS
TRANS	MP ME ST PR <> <> <> FL EM <> <> PP <> EME MFS
SUBSTR	MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS
SPECTR	MP ME ST PR PRN <> <> <> <> <> <> <> PP <> EME MFS

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>New Analysis**  
**Main Menu>Preprocessor>Loads>Analysis Type>Restart**  
**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic**  
**Main Menu>Solution>Analysis Type>New Analysis**  
**Main Menu>Solution>Analysis Type>Restart**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Basic**

**/ANUM**, *NUM*, *TYPE*, *XHOT*, *YHOT*

Specifies the annotation number, type, and hot spot (GUI).

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **NUM**

Annotation number. ANSYS automatically assigns the lowest available number. You cannot assign a higher number if a lower number is available; ANSYS will substitute the lowest available number in place of any user-specified higher number.

### **TYPE**

Annotation internal type number. If *TYPE* = DELE, delete annotation *NUM*.

- 1** Text
- 2** Block text (not available in GUI)
- 3** Dimensions
- 4** Lines
- 5** Rectangles

- 6 Circles
- 7 Polygons
- 8 Arcs
- 9 Wedges, pies
- 11 Symbols
- 12 Arrows
- 13 Bitmap

**XHOT**

X hot spot ( $-1.0 < X < 2.0$ ). Used for menu button item delete.

**YHOT**

Y hot spot ( $-1.0 < Y < 1.0$ ). Used for menu button item delete.

**Command Default**

Number, type, and hot spot are automatically determined.

**Notes**

This is a command generated by the GUI and will appear in the log file (Jobname.LOG) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

Type 13 (bitmap) annotation applies user defined bitmaps defined using the FILE option of the **/TXTRE** command.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>PlotCtrls>Annotation>Create Annotation**

**AOFFST, NAREA, DIST, KINC**

**Generates an area, offset from a given area.**

PREP7: Areas

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NAREA**

Area from which generated area is to be offset. If *NAREA* = ALL, offset from all selected areas [**ASEL**]. If *NAREA* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

***DIST***

Distance normal to given area at which keypoints for generated area are to be located. Positive normal is determined from the right-hand-rule keypoint order.

***KINC***

Keypoint increment between areas. If zero, the lowest available keypoint numbers are assigned **[NUMSTR]**.

## Notes

Generates an area (and its corresponding keypoints and lines) offset from a given area. The direction of the offset varies with the given area normal. End slopes of the generated lines remain the same as those of the given pattern. Area and line numbers are automatically assigned, beginning with the lowest available values **[NUMSTR]**.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Offset**

---

**AOVLAP**, *NA1, NA2, NA3, NA4, NA5, NA6, NA7, NA8, NA9*

**Overlaps areas.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NA1, NA2, NA3, . . . , NA9***

Numbers of areas to be operated on. If *NA1* = ALL, use all selected areas and ignore *NA2* to *NA9*. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

## Notes

Generates new areas which encompass the geometry of all the input areas. The new areas are defined by the regions of intersection of the input areas, and by the complementary (non-intersecting) regions. See [Solid Modeling](#) in the [Modeling and Meshing Guide](#) for an illustration. This operation is only valid when the region of intersection is an area. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Overlap>Areas**

---



**APLOT**, *NA1*, *NA2*, *NINC*, *DEGEN*, *SCALE*

**Displays the selected areas.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NA1***, ***NA2***, ***NINC***

Displays areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL (default), *NA2* and *NINC* are ignored and all selected areas [**ASEL**] are displayed.

***DEGEN***

Degeneracy marker:

**(blank)**

No degeneracy marker is used (default).

**DEGE**

A red star is placed on keypoints at degeneracies (see the [Modeling and Meshing Guide](#) ). Not available if **/FACET**,**WIRE** is set.

***SCALE***

Scale factor for the size of the degeneracy-marker star. The scale is the size in window space (-1 to 1 in both directions) (defaults to .075).

**Notes**

This command is valid in any processor. The degree of tessellation used to plot the selected areas is set through the **/FACET** command.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>Plot Degen Areas**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>Plot Degen Areas**

**Utility Menu>Plot>Areas**

**Utility Menu>Plot>Specified Entities>Areas**

**APPEND**, *LSTEP*, *SBSTEP*, *FACT*, *KIMG*, *TIME*, *ANGLE*, *NSET*

**Reads data from the results file and appends it to the database.**

[POST1: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***LSTEP***

Load step number of the data set to be read. Defaults to 1. If FIRST, ignore *SBSTEP* and *TIME* and read the first data set. If LAST, ignore *SBSTEP* and *TIME* and read the last data set. If NEXT, ignore *SBSTEP* and *TIME* and read the next data set. If already at the last data set, the next set is the first data set. If NEAR, ignore *SBSTEP* and read the data set nearest to *TIME*. If *TIME* is blank, read the first data set. If LIST, scan the results file to produce a summary of each load step (*FACT*, *KIMG*, *TIME* and *ANGLE* are ignored).

**SBSTEP**

Substep number (within *LSTEP*) (defaults to last substep of load step). For the Buckling (**ANTYPE**,BUCKLE) or Modal (**ANTYPE**,MODAL) analysis, the substep corresponds to the mode number (defaults to first mode). If *LSTEP* = LIST, *SBSTEP* = 0 or 1 will list the basic load step information; *SBSTEP* = 2 will also list the load step title, and label the imaginary data sets if they exist.

**FACT**

Scale factor applied to data read from the file. If zero (or blank), a value of 1.0 is used. Harmonic velocities or accelerations may be calculated from the displacement results from a Modal or Harmonic Response (**ANTYPE**,HARMIC) analyses. If *FACT* = VELO, the harmonic velocities (*v*) are calculated from the displacements (*d*) at a particular frequency (*f*) according to the relationship  $v = 2 \pi f d$ . Similarly, if *FACT* = ACEL, the harmonic accelerations (*a*) are calculated as  $a = (2 \pi f)^2 d$ .

**KIMG**

Used only with results from complex analyses:

**0**

Store real part of complex solution.

**1**

Store imaginary part.

**TIME**

Time-point identifying the data set to be read. For the harmonic response analyses, time corresponds to the frequency. For the buckling analysis, time corresponds to the load factor. Used only in the following cases: If *LSTEP* is NEAR, read the data set nearest to *TIME*. If both *LSTEP* and *SBSTEP* are zero (or blank), read data set at time = *TIME*. If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. Solution items not written to the results file [**OUTRES**] for either data set will result in a null item after data set interpolation. If *TIME* is beyond the last time point on the file, the last time point is used.

**ANGLE**

Circumferential location (0° to 360°). Defines the circumferential location for the harmonic calculations used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the harmonic elements (**PLANE25**, **PLANE75**, **PLANE78**, **FLUID81**, **PLANE83**, and **SHELL61**) of the load case. See the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details. Note that factored values of applied constraints and loads will overwrite any values existing in the database.

**NSET**

Data set number of the data set to be read. If a positive value for *NSET* is entered, *LSTEP*, *SBSTEP*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **APPEND**,LIST. To determine if data sets are real or imaginary, issue **APPEND**,LIST,2 which labels imaginary data sets.

**Notes**

Reads a data set from the results file and appends it to the existing data in the database for the selected model only. The existing database is not cleared (or overwritten in total), allowing the requested results data to be merged into the database. Various operations may also be performed during the read operation. The database must have the model geometry available (or used the **RESUME** command before the **APPEND** command to restore the geometry from File.DB).

**Menu Paths**

**Main Menu>General Postproc>Read Results>By Load Step**

Main Menu>General Postproc>Read Results>By Set Number  
Main Menu>General Postproc>Read Results>By Time/Freq

**APTN**, *NA1, NA2, NA3, NA4, NA5, NA6, NA7, NA8, NA9*

Partitions areas.

PREP7: Booleans

MP ME ST PRN <> <> FL EM EH DY PP <> EME MFS

**NA1, NA2, NA3, . . . , NA9**

Numbers of areas to be operated on. If *NA1* = ALL, *NA2* to *NA9* are ignored and all selected areas are used. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may be substituted for *NA1*.

## Notes

Partitions areas that intersect. This command is similar to the combined functionality of the **ASBA** and **AOVLAP** commands. If the intersection of two or more areas is an area (i.e., planar), new areas will be created with boundaries that conform to the area of intersection and to the boundaries of the non-intersecting portions of the input areas [**AOVLAP**]. If the intersection is a line (i.e., not planar), the areas will be subtracted, or divided, along the line(s) of intersection [**ASBA**]. Both types of intersection can occur during a single **APTN** operation. Areas that do not intersect will not be modified. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Partition>Areas

**ARCLLEN**, *Key, MAXARC, MINARC*

Activates the arc-length method.

SOLUTION: Nonlinear Options

MP ME ST <> PRN <> <> <> <> <> PP <> EME MFS

**Key**

Arc-length key:

**OFF**

Do not use the arc-length method (default).

**ON**

Use the arc-length method.

**MAXARC**

Maximum multiplier of the reference arc-length radius (default = 25).

**MINARC**

Minimum multiplier of the reference arc-length radius (default = 1/1000).

## Notes

Activates the arc-length method and sets the minimum and maximum multipliers for the arc-length radius. The reference arc-length radius is calculated from the load or displacement increment of the first iteration of the first substep. This increment is determined by the following formula:

$$\text{Reference Arc-Length Radius} = \text{Total Load (or Displacement)} / \text{NSBSTP}$$

where *NSBSTP* is the number of substeps specified on the **NSUBST** command.

The factors *MAXARC* and *MINARC* are used to define the limits of the arc-length radius by using the following formulas:

$$\text{lower limit} = \text{MINARC} * (\text{Reference Arc-Length Radius})$$

$$\text{upper limit} = \text{MAXARC} * (\text{Reference Arc-Length Radius})$$

In each subsequent substep, a new arc-length radius is first calculated based on the arc-length radius of the previous substep and the solution behavior. Next, the newly calculated arc-length radius is further modified so that it falls between the range of the upper limit and lower limit. If the solution does not converge even when using the lower limit of the arc-length radius, the solution will terminate.

These values, together with the reference arc-length radius, define the limit for the new arc-length radius.

**ARCLEN** must be turned OFF for any load step without an applied load or displacement.

You cannot use the arc-length method with the following controls: automatic time stepping [**AUTOTS**], line search [**LNSRCH**], and the DOF solution predictor [**PRED**]. If you activate the arc-length method after you set any of these controls, a warning message appears. If you choose to proceed with the arc-length method activation, ANSYS disables your automatic time stepping, line search, and DOF predictor settings.

You cannot use the arc-length method with tabular loads.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Arc-Length Opts**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL**  
**Main Menu>Solution>Load Step Opts>Nonlinear>Arc-Length Opts**

**ARCTRM**, *Lab*, *VAL*, *NODE*, *DOF*

**Controls termination of the solution when the arc-length method is used.**

[SOLUTION: Nonlinear Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

*Lab*

Specifies the basis of solution termination:

**OFF**

Does not use **ARCTRM** to terminate analysis (default).

**L**

Terminates the analysis if the first limit point has been reached. The first limit point is that point in the response history when the tangent stiffness matrix becomes singular (i.e., the point at which the structure becomes unstable). If *Lab* = L, arguments *VAL*, *NODE*, *DOF* are ignored.

**U**

Terminates the analysis when the displacement first equals or exceeds the maximum desired value.

**VAL**

Maximum desired displacement (absolute value). Valid only if *Lab* = U. The analysis terminates whenever the calculated displacement first equals or exceeds this value. For rotational degrees of freedom, *VAL* must be in radians (not degrees).

**NODE**

Node number corresponding to displacement used to compare with displacement specified by *VAL*. If blank, the maximum displacement will be used. Valid only if *Lab* = U.

**DOF**

Valid degree of freedom label for nodal displacement specified by *NODE*. Valid labels are UX, UY, UZ, ROTX, ROTY, ROTZ. Valid only if *NODE*>0 and *Lab* = U.

## Notes

The **ARCTRM** command is valid only when the arc-length method (**ARCLEN,ON**) is used.

It can be convenient to use this command to terminate the analysis when the first limit point is reached. In addition, the **NCNV** command should be used to limit the maximum number of iterations. If the **ARCTRM** command is not used, and the applied load is so large that the solution path can never reach that load, the arc-length solution will continue to run until a CPU time limit or a "maximum number of iterations" is reached.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Arc-Length Opts**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL**  
**Main Menu>Solution>Load Step Opts>Nonlinear>Arc-Length Opts**

## AREAS

Specifies "Areas" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**This command cannot be accessed from a menu.**

---

### **AREFINE**, *NA1*, *NA2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

**Refines the mesh around specified areas.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ***NA1*, *NA2*, *NINC***

Areas (*NA1* to *NA2* in increments of *NINC*) around which the mesh is to be refined. *NA2* defaults to *NA1*, and *NINC* defaults to 1. If *NA1* = ALL, *NA2* and *NINC* are ignored and all selected areas are used for refinement. If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

#### ***LEVEL***

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

#### ***DEPTH***

Depth of mesh refinement in terms of the number of elements outward from the indicated areas (defaults to 1).

#### ***POST***

Type of postprocessing to be done after element splitting, in order to improve element quality:

##### **OFF**

No postprocessing will be done.

##### **SMOOTH**

Smoothing will be done. Node locations may change.

##### **CLEAN**

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

#### ***RETAIN***

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

##### **ON**

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

##### **OFF**

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

## Notes

**AREFINE** performs local mesh refinement around the specified areas. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

**AREFINE** refines all area elements and tetrahedral volume elements that are adjacent to the specified areas. Any volume elements that are adjacent to the specified areas, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. See [Revising Your Model](#) in the *Modeling and Meshing Guide* for additional restrictions on mesh refinement.

This command is also valid for [rezoning](#).

## Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Areas

## AREMESH, LCOMB, ANGLE

Generates an area in which to create a new mesh for rezoning.

SOLUTION: Rezoning

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### LCOMB

Specifies how to combine adjacent line segments:

0

Line segments combined by connecting ends to ends. This value is the default.

-1

No line segments combined.

### ANGLE

The maximum angle (in degrees) allowed for connecting two line segments together. The default value is 30. This value is valid only when *LCOMB* = 0.

## Notes

Issue the **AREMESH** command *after* issuing a **REMESH,START** command and *before* issuing a **REMESH,FINISH** command.

The **AREMESH** command cannot account for an open area (or “hole”) inside a completely enclosed region. Instead, try meshing around an open area by selecting two adjoining regions; for more information, see [Hints for Remeshing Multiple Regions](#).

## Menu Paths

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Create Rezone Area

**AREVERSE**, *ANUM*, *NOEFLIP*

**Reverses the normal of an area, regardless of its connectivity or mesh status.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***ANUM***

Area number of the area whose normal is to be reversed. If *ANUM* = ALL, the normals of all selected areas will be reversed. If *ANUM* = P, graphical picking is enabled. A component name may also be substituted for *ANUM*.

***NOEFLIP***

Indicates whether you want to change the normal direction of the existing elements on the reversed area(s) so that they are consistent with each area's new normal direction.

**0**

Make the normal direction of existing elements on the reversed area(s) consistent with each area's new normal direction (default).

**1**

Do not change the normal direction of existing elements on the reversed area(s).

## Notes

You cannot use the **AREVERSE** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable. Also, you cannot use this command to change the normal direction for areas attached to volumes because IGES data is unchanged by reversal. Reversed areas that are attached to volumes need to be reversed again when imported.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

See [Revising Your Model](#) in the *Modeling and Meshing Guide* for more information.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Reverse Normals>of Areas**

---

**AROTAT**, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *PAX1*, *PAX2*, *ARC*, *NSEG*

**Generates cylindrical areas by rotating a line pattern about an axis.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NL1*, *NL2*, *NL3*, . . . , *NL6***

List of lines in the pattern to be rotated (6 maximum if using keyboard entry of *NL1* to *NL6*). The lines must lie in the plane of the axis of rotation. If *NL1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *NL1* = ALL, all selected lines will define the pattern to be rotated. A component name may also be substituted for *NL1*.



**PAX1, PAX2**

Keypoints defining the axis about which the line pattern is to be rotated.

**ARC**

Arc length (in degrees). Positive follows right-hand rule about *PAX1-PAX2* vector. Defaults to 360°.

**NSEG**

Number of areas (8 maximum) around circumference. Defaults to minimum number required for 90° - maximum arcs, i.e., 4 for 360°, 3 for 270°, etc.

**Notes**

Generates cylindrical areas (and their corresponding keypoints and lines) by rotating a line pattern (and its associated keypoint pattern) about an axis. Keypoint patterns are generated at regular angular locations, based on a maximum spacing of 90°. Line patterns are generated at the keypoint patterns. Arc lines are also generated to connect the keypoints circumferentially. Keypoint, line, and area numbers are automatically assigned, beginning with the lowest available values [NUMSTR]. Adjacent lines use a common keypoint. Adjacent areas use a common line.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Operate>Extrude>Lines>About Axis**

**ARSCALE, NA1, NA2, NINC, RX, RY, RZ, KINC, NOELEM, IMOVE**

**Generates a scaled set of areas from a pattern of areas.**

PREP7: Areas

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NA1, NA2, NINC**

Set of areas, *NA1* to *NA2* in steps of *NINC*, that defines the pattern to be scaled. *NA2* defaults to *NA1*, *NINC* defaults to 1. If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is defined by all selected areas. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

**RX, RY, RZ**

Scale factors to be applied to the X, Y, and Z keypoint coordinates in the active coordinate system. (*RR*, *R θ*, *RZ* for cylindrical; *RR*, *R θ*, *R ϕ* for spherical). Note that the *R θ* and *R ϕ* scale factors are interpreted as angular offsets. For example, if CSYS = 1, *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

**KINC**

Increment to be applied to keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned [NUMSTR].

**NOELEM**

Specifies whether nodes and elements are also to be generated:

**0**

Nodes and elements associated with the original areas will be generated (scaled) if they exist.

- 1**  
Nodes and elements will *not* be generated.

**IMOVE**

Specifies whether areas will be moved or newly defined:

- 0**  
Additional areas will be generated.

- 1**  
Original areas will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old areas are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

**Notes**

Generates a scaled set of areas (and their corresponding keypoints, lines, and mesh) from a pattern of areas. The MAT, TYPE, REAL, and ESYS attributes are based on the areas in the pattern and not the current settings. Scaling is done in the active coordinate system. Areas in the pattern could have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Operate>Scale>Areas**

**ARSYM**, *Ncomp*, *NA1*, *NA2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

**Generates areas from an area pattern by symmetry reflection.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Ncomp***

Symmetry key:

- X**  
X symmetry (default).

- Y**  
Y symmetry.

- Z**  
Z symmetry.

***NA1*, *NA2*, *NINC***

Reflect areas from pattern beginning with *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is all selected areas [**ASEL**]. If *Ncomp* = P, use graphical picking to specify areas and ignore *NA2* and *NINC*. A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

***KINC***

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

***NOELEM***

Specifies whether nodes and elements are also to be generated:

- 0**  
Generate nodes and elements associated with the original areas, if they exist.
- 1**  
Do not generate nodes and elements.

**IMOVE**

Specifies whether areas will be moved or newly defined:

- 0**  
Generate additional areas.
- 1**  
Move original areas to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old areas are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

**Notes**

Generates a reflected set of areas (and their corresponding keypoints, lines and mesh) from a given area pattern by a symmetry reflection (see analogous node symmetry command, **NSYM**). The *MAT*, *TYPE*, *REAL*, *ESYS*, and *SECNUM* attributes are based upon the areas in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. The active coordinate system must be a Cartesian system. Areas in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Areas are generated as described in the **AGEN** command.

See the **ESYM** command for additional information about symmetry elements.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Reflect>Areas**

**ASBA**, *NA1*, *NA2*, *SEPO*, *KEEP1*, *KEEP2*

**Subtracts areas from areas.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NA1**

Area (or areas, if picking is used) to be subtracted from. If *ALL*, use all selected areas. Areas specified in this argument are not available for use in the *NA2* argument. If *P*, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NA1*.

**NA2**

Area (or areas, if picking is used) to subtract. If *ALL*, use all selected areas (except those included in the *NA1* argument). A component name may also be substituted for *NA2*.

**SEPO**

Behavior if the intersection of the *NA1* areas and the *NA2* areas is a line or lines:

**(blank)**

The resulting areas will share line(s) where they touch.

**SEPO**

The resulting areas will have separate, but coincident line(s) where they touch.

**KEEP1**

Specifies whether *NA1* areas are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NA1* areas after **ASBA** operation (override **BOPTN** command settings).

**KEEP**

Keep *NA1* areas after **ASBA** operation (override **BOPTN** command settings).

**KEEP2**

Specifies whether *NA2* areas are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NA2* areas after **ASBA** operation (override **BOPTN** command settings).

**KEEP**

Keep *NA2* areas after **ASBA** operation (override **BOPTN** command settings).

## Notes

Generates new areas by subtracting the regions common to both *NA1* and *NA2* areas (the intersection) from the *NA1* areas. The intersection can be an area(s) or line(s). If the intersection is a line and *SEPO* is blank, the *NA1* area is divided at the line and the resulting areas will be connected, sharing a common line where they touch. If *SEPO* is set to SEPO, *NA1* is divided into two unconnected areas with separate lines where they touch. See [Solid Modeling](#) in the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **ASBA,ALL,ALL** will have no effect since all the areas (in *NA1*) will be unavailable as *NA2* areas.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by Area**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by Area**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Areas**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Areas**

---

**ASBL**, *NA*, *NL*, *--*, *KEEPA*, *KEEPL***Subtracts lines from areas.**

PREP7: Booleans

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**NA**

Area (or areas, if picking is used) to be subtracted from. If ALL, use all selected areas. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NA*.

**NL**

Line (or lines, if picking is used) to subtract. If ALL, use all selected lines. A component name may also be substituted for *NL*.

**--**

Unused field.

**KEEPA**

Specifies whether *NA* areas are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NA* areas after **ASBL** operation (override **BOPTN** command settings).

**KEEP**

Keep *NA* areas after **ASBL** operation (override **BOPTN** command settings).

**KEEPL**

Specifies whether *NL* lines are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NL* lines after **ASBL** operation (override **BOPTN** command settings).

**KEEP**

Keep *NL* lines after **ASBL** operation (override **BOPTN** command settings).

**Notes**

Generates new areas by subtracting the regions common to both the areas and lines (the intersection) from the *NA* areas. The intersection will be a line(s). See [Solid Modeling](#) in the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by Line**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by Line**

**ASBV**, *NA*, *NV*, *SEPO*, *KEEPA*, *KEEPV***Subtracts volumes from areas.**

PREP7: Booleans

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**NA**

Area (or areas, if picking is used) to be subtracted from. If ALL, use all selected areas. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NA*.

**NV**

Volume (or volumes, if picking is used) to subtract. If ALL, use all selected volumes. A component name may also be substituted for *NV*.

**SEPO**

Behavior if the intersection of the areas and the volumes is a line or lines:

**(blank)**

The resulting areas will share line(s) where they touch.

**SEPO**

The resulting areas will have separate, but coincident line(s) where they touch.

**KEEPA**

Specifies whether *NA* areas are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NA* areas after **ASBV** operation (override **BOPTN** command settings).

**KEEP**

Keep *NA* areas after **ASBV** operation (override **BOPTN** command settings).

**KEEPV**

Specifies whether *NV* volumes are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete volumes after **ASBV** operation (override **BOPTN** command settings).

**KEEP**

Keep volumes after **ASBV** operation (override **BOPTN** command settings).

**Notes**

Generates new areas by subtracting the regions common to both *NA* areas and *NV* volumes (the intersection) from the *NA* areas. The intersection can be an area(s) or line(s). If the intersection is a line and *SEPO* is blank, the *NA* area is divided at the line and the resulting areas will be connected, sharing a common line where they touch. If *SEPO* is set to SEPO, *NA* is divided into two unconnected areas with separate lines where they touch. See [Solid Modeling](#) in the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by Volume

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by Volume

Main Menu>Preprocessor>Modeling>Operate>Divide>Area by Volu

### **ASBW**, *NA*, *SEPO*, *KEEP*

**Subtracts the intersection of the working plane from areas (divides areas).**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### **NA**

Area (or areas, if picking is used) to be subtracted from. If *NA* = ALL, use all selected areas. If *NA* = P, graphical picking is enabled (valid only in the GUI). A component name may also be input for *NA*.

#### **SEPO**

Behavior of the created boundary.

#### **(blank)**

The resulting areas will share line(s) where they touch.

#### **SEPO**

The resulting areas will have separate, but coincident line(s).

#### **KEEP**

Specifies whether *NA* areas are to be deleted.

#### **(blank)**

Use the setting of KEEP on the **BOPTN** command.

#### **DELETE**

Delete *NA* areas after **ASBW** operation (override **BOPTN** command settings).

#### **KEEP**

Keep *NA* areas after **ASBW** operation (override **BOPTN** command settings).

## Notes

Generates new areas by subtracting the intersection of the working plane from the *NA* areas. The intersection will be a line(s). The working plane must not be in the same plane as the *NA* areas(s). If *SEPO* is blank, the *NA* area is divided at the line and the resulting areas will be connected, sharing a common line where they touch. If *SEPO* is set to SEPO, *NA* is divided into two unconnected areas with separate lines. The SEPO option may cause unintended consequences if any keypoints exist along the cut plane. See [Solid Modeling](#) in the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Issuing the **ASBW** command under certain conditions may generate a topological degeneracy error. Do not issue the command if:

- A sphere or cylinder has been scaled. (A cylinder must be scaled unevenly in the XY plane.)
- A sphere or cylinder has not been scaled but the work plane has been rotated.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by WrkPlane

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by WrkPlane

Main Menu>Preprocessor>Modeling>Operate>Divide>Area by WrkPlane

**ASEL**, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP*

Selects a subset of areas.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of select:

#### **S**

Select a new set (default)

#### **R**

Reselect a set from the current set.

#### **A**

Additionally select a set and extend the current set.

#### **U**

Unselect a set from the current set.

#### **ALL**

Restore the full set.

#### **NONE**

Unselect the full set.

#### **INVE**

Invert the current set (selected becomes unselected and vice versa).

#### **STAT**

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

### *Item*

Label identifying data. Valid item labels are shown in [Table 142: ASEL - Valid Item and Component Labels \(p. 165\)](#). Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to AREA.

### *Comp*

Component of the item (if required). Valid component labels are shown in [Table 142: ASEL - Valid Item and Component Labels \(p. 165\)](#).

### *VMIN*

Minimum value of item range. Ranges are area numbers, coordinate values, attribute numbers, etc., as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, or ESYS and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **ALIST** command for a discussion of signed attributes.



**VMAX**

Maximum value of item range. *VMAX* defaults to *VMIN*.

**VINC**

Value increment within range. Used only with integer ranges (such as for area numbers). Defaults to 1. *VINC* cannot be negative.

**KSWP**

Specifies whether only areas are to be selected:

**0**

Select areas only.

**1**

Select areas, as well as keypoints, lines, nodes, and elements associated with selected areas. Valid only with *Type* = *S*.

**Command Default**

All areas are selected.

**Notes**

Selects a subset of areas. For example, to select those areas with area numbers 1 through 7, use **ASEL,S,AREA,,1,7**. The selected subset is then used when the ALL label is entered (or implied) on other commands, such as **ALIST,ALL**. Only data identified by area number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

If *Item* = *ACCA*, the command selects only those areas that were created by concatenation. The *KSWP* field is processed, but the *Comp*, *VMIN*, *VMAX*, and *VINC* fields are ignored.

This command is valid in any processor.

For Selects based on non-integer numbers (coordinates, results, etc.), items that are within the range  $VMIN - Toler$  and  $VMAX + Toler$  are selected. The default tolerance *Toler* is based on the relative values of *VMIN* and *VMAX* as follows:

- If  $VMIN = VMAX$ ,  $Toler = 0.005 \times VMIN$ .
- If  $VMIN = VMAX = 0.0$ ,  $Toler = 1.0E-6$ .
- If  $VMAX \neq VMIN$ ,  $Toler = 1.0E-8 \times (VMAX - VMIN)$ .

Use the **SELTOL** command to override this default and specify *Toler* explicitly.

**Table 142 ASEL - Valid Item and Component Labels**

Valid Item and Component Labels <i>ASEL, Type Item, Comp, VMIN, VMAX, VINC, KSWP</i>		
Item	Comp	Description
AREA		Area number.
EXT		Area numbers on exterior of selected volumes (ignore remaining fields).
LOC	X, Y, Z	X, Y, or Z center (picking "hot spot" location in the active coordinate system).

**Valid Item and Component Labels** *ASEL, Type Item, Comp, VMIN, VMAX, VINC, KSWP*

Item	Comp	Description
HPT		Area number (selects only areas with associated hard points).
MAT		Material number associated with the area.
TYPE		Element type number associated with the area.
REAL		Real constant set number associated with the area.
ESYS		Element coordinate system associated with the area.
SECN		Section number associated with the area.
ACCA		Concatenated areas (selects only areas that were created by area concatenation [ <b>ACCAT</b> ]).

## Menu Paths

Utility Menu&gt;Select&gt;Entities

**\*ASK**, *Par, Query, DVAL***Prompts the user to input a parameter value.**[APDL: Parameters](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

## Argument Descriptions

***Par***An alphanumeric name used to identify the scalar parameter. See **\*SET** for name restrictions.***Query***

Text string to be displayed on the next line as the query (32 characters maximum). Characters having special meaning (such as \$ ! ,) should not be included.

***DVAL***

Default value assigned to the parameter if the user issues a blank response. May be a number or character string (up to 8 characters enclosed in single quotes). If a default is not assigned, a blank response will delete the parameter.

## Notes

Intended primarily for use in macros, the command prints the query (after the word `ENTER`) on the next line and waits for a response. The response is read from the keyboard, except in batch mode [**/BATCH**], when the response(s) must be the next-read input line(s). The response may be a number, a character string (up to 8 characters enclosed in single quotes), a parameter (numeric or character) or an expression that evaluates to a number. The scalar parameter is then set to the response value. For example, **\*ASK,NN,PARAMETER NN** will set NN to the value entered on the next line (after the prompt `ENTER PARAMETER NN`).

The **\*ASK** command is not written to `File.LOG`, but the responses are written there as follows: If **\*ASK** is contained in a macro, the response(s) (only) is written to `File.LOG` on the line(s) following the macro

name. If not contained in a macro, the response is written to File.LOG as a parameter assignment (i.e., *Par* = "user-response").

If used within a do-loop that is executed interactively, **\*ASK** should be contained in a macro. If not contained in a macro, **\*ASK** will still query the user as intended, but the resulting log file will *not* reproduce the effects of the original run.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

### ASKIN, NL1, NL2, NL3, NL4, NL5, NL6, NL7, NL8, NL9

**Generates an area by "skinning" a surface through guiding lines.**

PREP7: Areas

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### NL1

The first guiding line forming the skinned area. If *NL1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NL1*. If *NL1* is negative, the line beginnings and ends will be used to direct the skinning of the remaining lines (see "[Changing the ASKIN Algorithm](#)" (p. 167) below).

#### NL2, NL3, NL4, . . . , NL9

The additional guiding lines for the skinned area (up to 9 total lines, including *NL1*, if using keyboard entry). If negative (and *NL1* is negative), the line beginning and end will be temporarily interchanged for the skinning operation (see "[Changing the ASKIN Algorithm](#)" (p. 167) below).

## Notes

Generates an area by "skinning" a surface through specified guiding lines. The lines act as a set of "ribs" over which a surface is "stretched." Two opposite edges of the area are framed by the first (*NL1*) and last (*NLn*) guiding lines specified. The other two edges of the area are framed by splines-fit lines which the program automatically generates through the ends of all guiding lines. The interior of the area is shaped by the interior guiding lines. Once the area has been created, only the four edge lines will be attached to it. In rare cases, it may be necessary to change the default algorithm used by the **ASKIN** command (see "[Changing the ASKIN Algorithm](#)" (p. 167) below).

### Changing the ASKIN Algorithm

When skinning from one guiding line to the next, the program can create the transition area in one of two ways: one more spiraled and one less spiraled ("flatter"). By default, the program attempts to produce the flatter transition, instead of the more spiraled transition. This algorithm can be changed by inputting *NL1* as a negative number, in which case the program connects all the keypoints at the line "beginnings" (/PSYMB, LDIR command) as one edge of the area, and all the line "ends" as the opposite edge, irrespective of the amount of spiraling produced in each transition area.

To further control the geometry of the area (if *NL1* is negative), the beginning and end of any specified line (other than *NL1*) can be temporarily interchanged (for the skinning operation only) by inputting that line number as negative. See [Solid Modeling](#) in the *Modeling and Meshing Guide* for an illustration.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Skinning**

---

**ASLL**, *Type*, *ARKEY*

**Selects those areas containing the selected lines.**

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of area select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

### *ARKEY*

Specifies whether all contained area lines must be selected [**LSEL**]:

- 0**  
Select area if any of its lines are in the selected line set.
- 1**  
Select area only if all of its lines are in the selected line set.

## Notes

This command is valid in any processor.

## Menu Paths

**Utility Menu>Select>Entities**

---

**ASLV**, *Type*

Selects those areas contained in the selected volumes.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of area select:

**S**

Select a new set (default).

**R**

Reselect a set from the current set.

**A**

Additionally select a set and extend the current set.

**U**

Unselect a set from the current set.

**Notes**

This command is valid in any processor.

**Menu Paths**

**Utility Menu>Select>Entities**

**/ASSIGN**, *Ident, Fname, Ext, --*

Reassigns a file name to an ANSYS file identifier.

[SESSION: Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Ident**

ANSYS file name identifier. Valid identifiers are: CMS, EMAT, EROT, ESAV, FULL, LN07, LN09, LN11, LN20, LN21, LN22, LN25, LN31, LN32, MODE, OSAV, RDSP, REDM, RFL, RFRQ, RMG,RST, RSTP, RTH, SELD, SSCR, and TRI. See "[File Management and Files](#)" for file descriptions. If blank, list currently reassigned files.

**Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

**Ext**

Filename extension (8 character maximum).

**--**

Unused field.

## Notes

The reassignment of file names is valid only if it is done before the file is used. All file reassignments are retained (not cleared) even if the database is cleared [/CLEAR] or the Jobname is changed [/FILNAME]. Assigned files may be overwritten. If file name arguments (*Fname*, *Ext*, *--*) are blank, the default ANSYS assignment is restored. Use **SEOPT** for SUB files and **SEEXP** for DSUB files.

This command is valid only at the Begin Level.

This command also checks to ensure that the path/file is valid and can be written by the user. If it is not valid, an error message will be returned. Ensure that the directory exists prior to using /ASSIGN command.

## Menu Paths

Utility Menu>File>ANSYS File Options

---

**ASUB**, *NA1*, *P1*, *P2*, *P3*, *P4*

**Generates an area using the shape of an existing area.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NA1**

Existing area number whose shape is to be used. If *P1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

**P1**

Keypoint defining starting corner of area.

**P2**

Keypoint defining second corner of area.

**P3**

Keypoint defining third corner of area.

**P4**

Keypoint defining fourth corner of area (defaults to *P3*).

## Notes

The new area will overlay the old area. Often used when the area to be subdivided consists of a complex shape that was not generated in a single coordinate system. Keypoints and any corresponding lines must lie on the existing area. Missing lines are generated to lie on the given area. The active coordinate system is ignored.

## Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>Overlaid on Area

---

**ASUM**, *LAB*

**Calculates and prints geometry statistics of the selected areas.**

PREP7: Areas

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**LAB**

Controls the degree of tessellation used in the calculation of area properties. If *LAB* = DEFAULT, area calculations will use the degree of tessellation set through the **/FACET** command. If *LAB* = FINE, area calculations are based on a finer tessellation.

**Notes**

Calculates and prints geometry statistics (area, centroid location, moments of inertia, volume, etc.) associated with the selected areas. **ASUM** should only be used on perfectly flat areas.

Geometry items are reported in the global Cartesian coordinate system. A unit density (and thickness) is assumed unless the areas have a material (and real constant) association via the **AATT** command. The command always uses a unit density and a unit thickness for layered shell elements (such as **SHELL131**, **SHELL163**, **SHELL181**, and **SHELL281**).

Items calculated by **ASUM** and later retrieved via a **\*GET** or **\*VGET** command are valid only if the model is not modified after issuing the **ASUM** command.

Setting a finer degree of tessellation will provide area calculations with greater accuracy, especially for thin, hollow models. However, using a finer degree of tessellation requires longer processing.

For very narrow (sliver) areas, such that the ratio of the minimum to the maximum dimension is less than 0.01, the **ASUM** command can provide erroneous area information. To ensure that such calculations are accurate, make certain that you subdivide such areas so that the ratio of the minimum to the maximum is at least 0.05.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Areas**

**ATAN**, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*

**Forms the arctangent of a complex variable.**

POST26: Operations

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

**IR**

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

**IA**

Reference number of the complex variable to be operated on.

--,--

Unused fields.

**Name**

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--,--

Unused fields.

**FACTA**

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0). Usually *FACTA* should be set to 1. *FACTA* may affect the position of the angle by a multiple of  $\pi$ , resulting in a quadrant change.

**Notes**

Forms the arctangent of a complex variable according to the operation:

$$IR = ATAN(FACTA \times b/a)$$

where *a* and *b* are the real and imaginary parts, respectively, of the complex variable *IA* (which is of the form *a + ib*). The arctangent represents the phase angle (in radians), and is valid only for a harmonic analysis (**ANTYPE**,HARMIC).

Since the scaling factor is applied uniformly to *b/a*, applying any positive or negative scaling factor will not affect the size of the phase angle, with the exception that a negative scaling factor will change the results quadrant by  $\pi$ . The magnitude of a complex number is still obtained through the **ABS** command. See [POST26 - Data Operations](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details.

**Menu Paths**

**Main Menu>TimeHist Postpro>Math Operations>Arctangent**

---

**ATRAN, KCNTO, NA1, NA2, NINC, KINC, NOELEM, IMOVE**

**Transfers a pattern of areas to another coordinate system.**

[PREP7: Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KCNTO**

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system. The coordinate system type and parameters of *KCNTO* must be the same as the active system.

**NA1, NA2, NINC**

Transfer area pattern beginning with *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is all selected areas [**ASEL**]. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

**KINC**

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].



**NOELEM**

Specifies whether elements and nodes are also to be generated:

**0**

Generate nodes and elements associated with the original areas, if they exist.

**1**

Do not generate nodes and elements.

**IMOVE**

Specifies whether to redefine the existing areas:

**0**

Generate additional areas.

**1**

Move original areas to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old areas are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

**Notes**

Transfers a pattern of areas (and their corresponding lines, keypoints and mesh) from one coordinate system to another (see analogous node **TRANSFER** command). The MAT, TYPE, REAL, and ESYS attributes are based upon the areas in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Coordinate and slope values are interpreted in the active coordinate system and are transferred directly. Areas are generated as described in the **AGEN** command.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Areas**

**ATYPE**

Specifies "Analysis types" as the subsequent status topic.

**SOLUTION:Status**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

**Menu Paths**

**Utility Menu>List>Status>Solution>Analysis Type**

## /AUTO, WN

Resets the focus and distance specifications to "automatically calculated."

[GRAPHICS: Views](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### WN

Window number (or ALL) to which command applies (defaults to 1).

## Notes

Focus point and distance will be automatically calculated during next display. Settings may still be changed with the **/FOCUS** and **/DIST** commands after this command has been issued. See also the **/USER** command.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Utility Menu>PlotCtrls>View Settings>Automatic Fit Mode

---

## AUTOTS, Key

Specifies whether to use automatic time stepping or load stepping.

[SOLUTION: Load Step Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### Key

Automatic time stepping key:

#### OFF

Do not use automatic time stepping.

#### ON

Use automatic time stepping.

## Command Default

ANSYS determined time stepping when **SOLCONTROL,ON**. No automatic time stepping when **SOLCONTROL,OFF**.

## Notes

Specifies whether to use automatic time stepping (or load stepping) over this load step. If *Key* = ON, both time step prediction and time step bisection will be used. Used only if *DTIME* (specified on the **DELTIM** command) is less than the time span or conversely, if *NSBSTP* (on the **NSUBST** command) is greater than one.

If you run an analysis with **SOLCONTROL,ON**, but do not issue the **AUTOTS** command, ANSYS will choose whether or not to use automatic time stepping. The program-chosen option will be recorded on the log file as **AUTOTS,-1**.

You cannot use automatic time stepping [**AUTOTS**], line search [**LNSRCH**], or the DOF solution predictor [**PRED**] with the arc-length method [**ARCLN**, **ARCTRM**]. If you activate the arc-length method after you set **AUTOTS**, **LNSRCH**, or **PRED**, a warning message appears. If you choose to proceed with the arc-length method activation, ANSYS disables your automatic time stepping, line search, and DOF predictor settings.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Basic**  
**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step**  
**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substps**

---

## /AUX2

Enters the binary file dumping processor.

AUX2: Binary Files  
SESSION: Processor Entry

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Enters the binary file dumping processor (ANSYS auxiliary processor AUX2). This processor is used to dump the contents of certain ANSYS binary files for visual examination.

This command is valid only at the Begin Level.

## Menu Paths

**Utility Menu>File>List>Binary Files**  
**Utility Menu>List>Files>Binary Files**

---

## /AUX3

Enters the results file editing processor.

AUX3: Results Files  
SESSION: Processor Entry

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Enters the results file editing processor (ANSYS auxiliary processor AUX3). This processor is used to edit ANSYS results files.

This command is valid only at the Begin Level.

## Menu Paths

This command cannot be accessed from a menu.

---

### /AUX12

Enters the radiation processor.

AUX12:General Radiation  
SESSION:Processor Entry

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

## Notes

Enters the radiation processor (ANSYS auxiliary processor AUX12). This processor supports the Radiation Matrix and the Radiosity Solver methods.

This command is valid only at the Begin Level.

## Menu Paths

Main Menu>Radiation Matrix

---

### /AUX15

Enters the IGES file transfer processor.

AUX15:IGES  
POST1:Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Enters the IGES file transfer processor (ANSYS auxiliary processor AUX15), used to read an IGES data file into the ANSYS program.

This command is valid only at the Begin Level.

## Menu Paths

Utility Menu>File>Import

---

**AVPRIN**, *KEY*, *EFFNU*

**Specifies how principal and vector sums are to be calculated.**

POST1:Controls

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

**KEY**

Averaging key:

**0**

Average the component values from the elements at a common node, then calculate the principal or vector sum from the averaged components (default).

**1**

Calculate the principal or vector sum values on a per element basis, then average these values from the elements at a common node.

**EFFNU**

Effective Poisson's ratio used for computing the von Mises equivalent strain (EQV). This command option is intended for use with line elements or in load case operations (**LCOPER**) only; ANSYS automatically selects the most appropriate effective Poisson's ratio, as discussed below.

**Command Default**

Average components at common node before principal or vector sum calculation except for the von Mises equivalent strain (EQV), see below.

**Notes**

Selects the method of combining components for certain derived nodal results when two or more elements connect to a common node. The methods apply to the calculations of derived nodal principal stresses, principal strains, and vector sums for selects, sorts, and output [**NSOL**, **NSORT**, **PRNSOL**, **PLNSOL**, etc.].

This command also defines the effective Poisson's ratio (*EFFNU*) used for equivalent strain calculations. If you use *EFFNU*, the default effective Poisson's ratios shown below will be overridden for all elements by the *EFFNU* value. To return to the default settings, issue the **RESET** command. The default value for *EFFNU* is:

- Poisson's ratio as defined on the **MP** commands for EPEL and EPTH
- 0.5 for EPPL and EPCR
- 0.5 if the referenced material is hyperelastic
- 0.0 for line elements (includes beam, link, and pipe elements, as well as discrete elements), *cyclic symmetry analysis*, and load case operations (**LCOPER**).

For the von Mises equivalent strain (EQV), it is always computed using the average of the equivalent strains from the elements at a common node irrespective of the value of the averaging *KEY*. If *EFFNU* is input, though, the calculation will be performed according to the *KEY* setting.

For a random vibration (PSD) analysis, issuing either **AVPRIN,0** or **AVPRIN,1** calculates the principal stresses using the appropriate averaging method. They are then used to determine SEQV. The output will have non-zero values for the principal stresses.

If **AVPRIN** is not issued, the [Segalman-Fulcher method](#) is used to calculate SEQV. This method does not calculate principal stresses, but directly calculates SEQV from the component stresses; therefore, the output will have zero values for the principal stresses.

This command is also valid in POST26, where applicable.

See [Combined Stresses and Strains](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information.

## Menu Paths

**Main Menu>General Postproc>Element Table>Define Table**  
**Main Menu>General Postproc>List Results>Nodal Solution**  
**Main Menu>General Postproc>Options for Outp**  
**Main Menu>General Postproc>Path Operations>Map onto Path**  
**Main Menu>General Postproc>Path Operations>Map onto Path>FE Results**  
**Main Menu>General Postproc>Plot Results>Contour Plot>Element Solu**  
**Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu**  
**Utility Menu>List>Results>Nodal Solution**  
**Utility Menu>List>Results>Options**  
**Utility Menu>Plot>Results>Contour Plot>Elem Solution**  
**Utility Menu>Plot>Results>Contour Plot>Nodal Solution**  
**Utility Menu>PlotCtrls>Multi-Plot Contrls**

## AVRES, KEY, Opt

Specifies how results data will be averaged when PowerGraphics is enabled.

[POST1:Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### KEY

Averaging key.

- 1  
Average results at all common subgrid locations.
- 2  
Average results at all common subgrid locations except where material type **[MAT]** discontinuities exist. This option is the default.
- 3  
Average results at all common subgrid locations except where real constant **[REAL]** discontinuities exist.
- 4  
Average results at all common subgrid locations except where material type **[MAT]** or real constant **[REAL]** discontinuities exist.

### Opt

Option to determine how results data are averaged.

#### (blank)

Average surface results data using only the exterior element faces (default).

**FULL**

Average surface results data using the exterior face and interior element data.

**Notes**

The **AVRES** command specifies how results data will be averaged at subgrid locations that are common to 2 or more elements. The command is valid only when PowerGraphics is enabled (via the **/GRAPHICS,POWER** command).

With PowerGraphics active (**/GRAPHICS,POWER**), the averaging scheme for surface data with interior element data included (**AVRES,,FULL**) and multiple facets per edge (**/EFACET,2** or **/EFACET,4**) will yield differing minimum and maximum contour values depending on the Z-Buffering options (**/TYPE,,6** or **/TYPE,,7**). When the Section data is not included in the averaging schemes (**/TYPE,,7**), the resulting absolute value for the midside node is significantly smaller.

PowerGraphics does not average your stresses across discontinuous surfaces. The normals for various planes and facets are compared to a tolerance to determine continuity. The **ANGLE** value you specify in the **/EDGE** command is the tolerance for classifying surfaces as continuous or “coplanar.”

The command affects nodal solution contour plots (**PLNSOL**), nodal solution printout (**PRNSOL**), and subgrid solution results accessed through the Query Results function (under General Postprocessing) in the GUI.

The command has no effect on the nodal degree of freedom solution values (UX, UY, UZ, TEMP, etc.).

The command is also available in **/SOLU**.

**Menu Paths**

**Main Menu>General Postproc>Options for Outp  
Utility Menu>List>Results>Options**

**/AXLAB**, *Axis*, *Lab*

**Labels the X and Y axes on graph displays.**

**GRAPHICS: Graphs**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Axis**

Axis specifier:

**X**

Apply label to X axis.

**Y**

Apply label to Y axis.

**Lab**

Axis label (user defined text up to 30 characters long). Leave blank to reestablish the default for *Axis* axis.

## Command Default

Labels are determined by the program.

## Notes

This command is valid in any processor.

## Menu Paths

Main Menu>Drop Test>Time History>Graph Variables  
Main Menu>Drop Test>Time History>List Variables  
Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

---

**\*AXPY**,  $vr$ ,  $vi$ ,  $M1$ ,  $wr$ ,  $wi$ ,  $M2$

Performs the matrix operation  $M2 = v * M1 + w * M2$ .

[APDL: Matrix Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

$vr$ ,  $vi$

The real and imaginary parts of the scalar  $v$ . Default value is 0.

$M1$

Name of matrix  $M1$ . If not specified, the operation  $M2 = w * M2$  will be performed.

$wr$ ,  $wi$

The real and imaginary parts of the scalar  $w$ . Default value is 0.

$M2$

Name of matrix  $M2$ . Must be specified.

## Notes

The matrices  $M1$  and  $M2$  must have the same dimensions and same type (dense or sparse). If  $M2$  is real,  $vi$  and  $wi$  are ignored.

## Menu Paths

**This command cannot be accessed from a menu.**

---



---

## B Commands

**/BATCH**, *Lab*

Sets the program mode to "batch."

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Lab*

Specifies listing mode during a batch run:

**LIST**

The batch output will include a listing of the input file.

**(blank)**

Suppress input data listing.

### Command Default

Interactive mode.

### Notes

Sets the program mode to "batch" when included as the first line on an input file of ANSYS commands. For convenience, this command is automatically recorded on the log file (*Jobname*.LOG) at the beginning of an interactive ANSYS session so that the log file can be reused later for batch input.

---

### Caution

This command should not be entered directly in an interactive ANSYS session since all subsequent keyboard input is simply copied to a file, without further processing or prompts from the program (use the "system break" to exit the ANSYS program if this occurs).

The recommended method for choosing batch mode, rather than using the **/BATCH** command, is to select the **Batch** simulation environment from the **ANSYS Product Launcher** task in the ANSYS launcher, or the batch mode entry option on the ANSYS execution command when entering the program.

This command is valid only at the Begin Level.

### Menu Paths

**This command cannot be accessed from a menu.**

**BCSOPTION**, --, *Memory\_Option*, *Memory\_Size*, --, --, *Solve\_Info***Sets memory option for the sparse solver.**

SOLUTION: Analysis Options

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

--

Unused field

***Memory\_Option*****DEFAULT**

Use the default memory allocation strategy for the sparse solver. The default strategy attempts to run in the INCORE memory mode. If there is not enough available physical memory when the solver starts to run in the INCORE memory mode, the solver will then attempt to run in the OPTIMAL memory mode.

**INCORE**

Use a memory allocation strategy in the sparse solver that will attempt to obtain enough memory to run with the entire factorized matrix in memory. This option uses the most amount of memory and should avoid doing any I/O. By avoiding I/O, this option often achieves the best performance of all the memory modes. However, a significant amount of memory is required to run in this mode, and it is only recommended on machines with a large amount of memory. If the allocation for in-core memory fails, ANSYS will automatically revert to OPTIMAL out-of-core mode.

**OPTIMAL**

Use a memory allocation strategy in the sparse solver that will attempt to allocate enough work space to keep the largest front matrix in memory during factorization. As a result, this memory mode often achieves an optimal balance between I/O and memory usage.

**MINIMUM**

Use a memory allocation strategy in the sparse solver that will attempt to allocate the least amount of memory needed to run through the solver. By using the least amount of memory, this memory mode will often perform excessive amounts of I/O, which results in decreased performance. Consequently, this option is only recommended as a last resort when trying to run a relatively large analysis on a machine with limited memory resources.

**FORCE**

This option, when used in conjunction with the *Memory\_Size* option, allows you to force the sparse solver to run with a specific amount of memory. This option is only recommended for the advanced user who understands sparse solver memory requirements for the problem being solved, understands the physical memory on the system, and wants to control the sparse solver memory usage.

***Memory\_Size***

Sets the initial memory size allocation for the sparse solver in MB. This option allows you to tune the sparse solver memory and is not generally required. Although there is no upper limit for *Memory\_Size*, the *Memory\_Size* setting should always be well within the physical memory available, but not so small as to cause the sparse solver to run out of memory. Warnings and/or errors from the sparse solver will appear if this value is set too low. If the FORCE memory option is used, this value is the amount of memory allocated for the entire duration of the sparse solver solution.

--, --

Unused fields

## *Solve\_Info*

### **OFF**

Turns off additional output printing from the sparse solver (default).

### **PERFORMANCE**

Turns on additional output printing from the sparse solver, including a performance summary and a summary of file I/O for the sparse solver. Information on memory usage during assembly of the global matrix (that is, creation of the `Jobname.FULL` file) is also printed with this option.

## **Command Default**

Automatic memory allocation is used.

## **Notes**

This command controls options related to the sparse solver in all analysis types where the sparse solver can be used. It also controls the Block Lanczos eigensolver in a modal or buckling analysis.

The sparse solver runs from one large work space (that is, one large memory allocation). The amount of memory required for the sparse solver is unknown until the matrix structure is preprocessed, including equation reordering. The amount allocated for the sparse solver is then dynamically adjusted using the ANSYS memory manager.

If you have a very large memory system, you may want to try selecting the INCORE memory mode for larger jobs to improve performance. If you have a very small memory system, you may need to run in the MINIMUM memory mode, in which the program will most likely use the least amount of memory but do the most amount of I/O, often leading to the least efficient performance.

Running with the INCORE memory mode is best for jobs which comfortably fit within the limits of the physical memory on a given system. If the sparse solver work space exceeds physical memory size, the system will be forced to use virtual memory (or the system page/swap file). In this case, it is typically more efficient to run with the OPTIMAL memory mode (**BCSOPTION,,OPTIMAL**). Assuming the job fits comfortably within the limits of the machine, running with the INCORE memory mode is often ideal for jobs where repeated solves are performed for a single matrix factorization. This occurs in a modal or buckling analysis or when doing multiple load steps in a linear, static analysis.

For repeated runs with the sparse solver, you may set the initial sparse solver memory allocation to the amount required for factorization. This strategy reduces the frequency of allocation and reallocation in the run to make the INCORE option fully effective. If you have a very large memory system, you may use the *Memory\_Size* argument to increase the maximum size attempted for in-core runs.

## **Menu Paths**

**This command cannot be accessed from a menu.**

**BETAD**, *VALUE*

Defines the stiffness matrix multiplier for damping.

SOLUTION: Dynamic Options

MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS

*VALUE*

Stiffness matrix multiplier for damping.

**Notes**

This command defines the stiffness matrix multiplier,  $\beta$ , in the damping expression given in the **ALPHAD** command. Values of  $\beta$  may also be input as a material property (use the DAMP label on the **MP** command). If DAMP is included, the DAMP value is added to the BETAD value as appropriate (see [Damping Matrices](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*). Damping is not used in the static (**ANTYPE**,STATIC) or buckling (**ANTYPE**,BUCKLE) analyses.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Transient**  
**Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping**

**BF**, *NODE*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

Defines a nodal body force load.

SOLUTION: FE Body Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

*NODE*

Node to which body load applies. If *NODE* = ALL, apply to all selected nodes [**NSEL**]. A component name may also be substituted for *NODE*.

*Lab*

Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the [Element Reference](#).

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FREQ	frequency (harmonic analyses only)
	FLUE	fluence
Thermal	HGEN	heat generation rate

Discipline	Body Load Label	Label Description
Magnetic	TEMP	temperature
	MVDI	magnetic virtual displacements flags
Electric	TEMP	temperature
	CHRGD	charge density
High-frequency electromagnetic	JS	current density
	H	magnetic field
	EF	electric field
	PORT	number 1-50 for interior waveguide or transmission line port
	CHRGD	inner conductor flag
	IMPD	impedance sheet
	LUMP	shunt RCL lumped circuit
FLOTRAN	HGEN	heat generation rate
	FORC	nodal body force densities in momentum equation

#### **VAL1, VAL2, VAL3, VAL4**

Value associated with the *Lab* item or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent signs (%) (e.g., **BF,NODE,TEMP,%tablename%**). Use the **\*DIM** command to define a table. Use only *VAL1* for TEMP, FLUE, HGEN, MVDI, CHRGD, and PORT. If *Lab* = PORT, *VAL1* is a port number representing an interior waveguide or transmission line port. If *Lab* = CHRGD for a high-frequency inner conductor flag, *VAL1* is the inner conductor number (no default).

If *Lab* = JS:

#### **VAL1**

X-Component.

#### **VAL2**

Y-Component.

#### **VAL3**

Z-Component.

#### **VAL4**

Phase angle in degrees or a negative port number for a driven port.

If *Lab* = H or EF:

#### **VAL1**

X-Component.

**VAL2**

Y-Component.

**VAL3**

Z-Component.

**VAL4**

Phase angle in degrees.

If  $Lab = \text{FORC}$ :

**VAL1**

X-Component.

**VAL2**

Y-Component.

**VAL3**

Z-Component.

**VAL4**

Not used.

If  $Lab = \text{IMPD}$ :

**VAL1**

Resistance in ohms/square

**VAL2**

Reactance in ohms/square

**VAL3**

Not used.

**VAL4**

Not used.

If  $Lab = \text{LUMP}$ :

**VAL1**

Resistance (ohm) of shunt RCL lumped circuit.

**VAL2**

Capacitance (F) of shunt RCL lumped circuit.

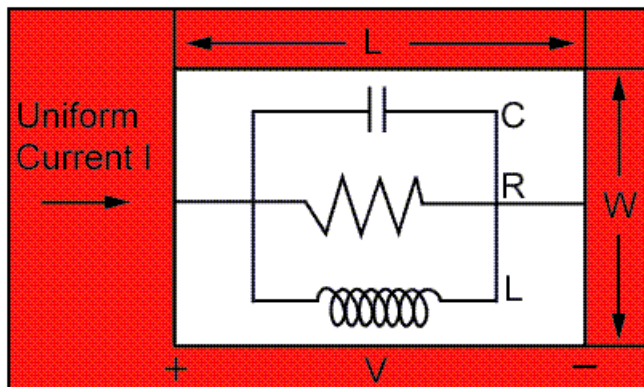
**VAL3**

Inductance (H) of shunt RCL lumped circuit.

**VAL4**

As shown in the following figure, ratio of length (L) to width (W) of a shunt RCL lumped circuit with uniform current (I). *VAL4* defaults to 1.0.

**Figure 1 RCL Lumped Circuit**



## Notes

Defines a nodal body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.). Nodal body loads default to the **BFUNIF** values, if they were previously specified.

You can specify a table name ( $VAL = \%tabname\%$ ) only for temperature (TEMP), heat generation rate (HGEN), and nodal body force density (FORC) body load labels.

The heat generation rate loads specified with the **BF** command are multiplied by the weighted nodal volume of each element adjacent to that node. This yields the total heat generation at that node.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppElecField>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppElecField>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Nodes**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Nodes**

**BFA**, *AREA*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

**Defines a body force load on an area.**

[SOLUTION: Solid Body Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### **AREA**

Area to which body load applies. If ALL, apply to all selected areas [[ASEL](#)]. A component name may also be substituted for *AREA*.

#### **Lab**

Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the [Element Reference](#).

<b>Discipline</b>	<b>Body Load Label</b>	<b>Label Description</b>
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	VLTG	voltage drop
Electric	TEMP	temperature
	CHRGD	charge density
High-frequency electromagnetic	JS	current density
	H	magnetic field
	EF	electric field
	PORT	number 1-50 for interior waveguide or transmission line port
	CHRGD	inner conductor flag
	IMPD	impedance sheet
	LUMP	shunt RCL lumped circuit
FLOTRAN	HGEN	heat generation rate

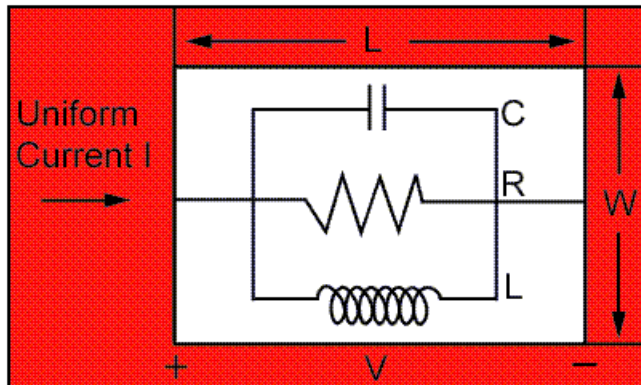


**VAL1, VAL2, VAL3**

Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, CHRGD, and PORT. If *Lab* = PORT, *VAL1* is a port number representing an interior waveguide or transmission line port. If *Lab* = CHRGD for a high-frequency inner conductor flag, *VAL1* is the inner conductor number (no default). Use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS, H, and EF. For *Lab* = VLTG, *VAL1* is the voltage drop and *VAL2* is the phase angle. If *Lab* = IMPD, *VAL1* is the resistance and *VAL2* is the reactance in ohms/square. If *Lab* = LUMP, *VAL1* is the resistance, *VAL2* is the capacitance, and *VAL3* is the inductance of the shunt RCL lumped circuit. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFA,AREA,Lab,%tablename%**. Use the **\*DIM** command to define a table.

**VAL4**

If *Lab* = H or EF, *VAL4* is the phase angle in degrees. If *Lab* = JS, *VAL4* is the phase angle in degrees. If *Lab* = LUMP, *VAL4* is the ratio of length (L) to width (W) of a shunt RCL lumped circuit with uniform current (I).

**Figure 2 RCL Lumped Circuit****Notes**

Defines a body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) on an area. Body loads may be transferred from areas to area elements (or to nodes if area elements do not exist) with the **BFTRAN** or **SBCTRAN** commands. Body loads default to the value specified on the **BFUNIF** command, if it was previously specified.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels.

Body loads specified by the **BFA** command can conflict with other specified body loads. See [Resolution of Conflicting Body Load Specifications](#) in the *Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppElecField>On Areas**

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppElecField>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Areas

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Areas

## BFADELE, AREA, Lab

Deletes body force loads on an area.

SOLUTION: Solid Body Loads

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### AREA

Area at which body load is to be deleted. If ALL, delete for all selected areas [ASEL]. A component name may also be substituted for AREA.

### Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the BFA command for labels.

## Notes

Deletes body force loads (and all corresponding finite element loads) for a specified area and label. Body loads may be defined on an area with the BFA command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelElecField>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelIntPort>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Areas  
 Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelElecField>On Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelIntPort>On Areas  
 Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Areas  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Areas  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Areas  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Areas  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Areas  
 Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Areas  
 Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Areas  
 Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Areas

## BFALIST, AREA, Lab

Lists the body force loads on an area.

SOLUTION: Solid Body Loads

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### AREA

Area at which body load is to be listed. If ALL (or blank), list for all selected areas [ASEL]. If AREA = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for AREA.

### Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the BFA command for labels.

## Notes

Lists the body force loads for the specified area and label. Body loads may be defined on an area with the BFA command.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>Body Loads>On All Areas

Utility Menu>List>Loads>Body Loads>On Picked Areas

**BFCUM**, *Lab*, *Oper*, *FACT*, *TBASE*

**Specifies that nodal body force loads are to be accumulated.**

[SOLUTION: FE Body Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *Lab*

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density

### *Oper*

Accumulation key:

#### **REPL**

Subsequent values replace the previous values (default).

#### **ADD**

Subsequent values are added to the previous values.

#### **IGNO**

Subsequent values are ignored.

### *FACT*

Scale factor for the nodal body load values. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor. The scale factor is not applied to body load phase angles.

### *TBASE*

Used (only with *Lab* = TEMP) to calculate the temperature used in the add or replace operation (see *Oper*) as:

$$\text{Temperature} = TBASE + FACT * (T - TBASE)$$

where  $T$  is the temperature specified on subsequent **BF** commands. *TBASE* defaults to zero.

## Command Default

Replace previous values.

## Notes

Allows repeated nodal body force loads to be replaced, added, or ignored. Nodal body loads are applied with the **BF** command. Issue the **BFLIST** command to list the nodal body loads. The operations occur when the next body loads are defined. For example, issuing the **BF** command with a temperature of 250 after a previous **BF** command with a temperature of 200 causes the new value of the temperature to be 450 with the add operation, 250 with the replace operation, or 200 with the ignore operation. A scale factor is also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a temperature of 700. The scale factor is applied even if no previous values exist. Issue **BFCUM,STAT** to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

---

### Note

FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

**BFCUM** does not work for tabular boundary conditions.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Nodal Body Ld**  
**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Nodal Body Ld**

---

**BFDELE**, *NODE*, *Lab*

**Deletes nodal body force loads.**

[SOLUTION: FE Body Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *NODE*

Node at which body load is to be deleted. If ALL, delete for all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

### *Lab*

Valid body load label. If ALL, use all appropriate labels. See the **BF** command for labels. In an explicit dynamic analysis, the only valid body load label is TEMP.

## Notes

Deletes nodal body force loads for a specified node and label. Nodal body loads may be defined with the **BF** command (except in an explicit dynamic analysis).

The command **BFDELE,TEMP** can be used in an explicit dynamic analysis to delete temperature loads that are read in by the **LDREAD** command. **BFDELE** cannot be used to delete temperature loads defined by the **EDLOAD** command (use **EDLOAD,DELE** to delete this type of load).

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelElecField>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelIntPort>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Body Forces**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Node Components**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Nodes**

**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Delete Temps**

**Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Nodes**

**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelElecField>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelIntPort>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Body Forces**

**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Node Components**

**Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Nodes**

**Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Nodes**

**Main Menu>Solution>Loading Options>Delete Temps**

**BFE**, *ELEM*, *Lab*, *STLOC*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

**Defines an element body force load.**

SOLUTION: FE Body Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

***ELEM***

Element to which body load applies. If ALL, apply to all selected elements (**ESEL**). A component name may also be substituted for *ELEM*.

***Lab***

Valid body load label. Valid labels are also listed for each element type in the *Element Reference* under "Body Loads" in the input table.

Discipline	Body Load Label	Label Description
Structural	TEMP	Temperature
	FREQ	Frequency (harmonic analyses only)
	FLUE	Fluence
	FORC	Body force density in momentum equation
Thermal	HGEN	Heat generation rate
Magnetic	EF	Electric field
	TEMP	Temperature
	JS	Current density
	VLTG	Voltage drop
	MVDI	Magnetic virtual displacements flag
Electric	TEMP	Temperature
	CHRGD	Charge density
High-frequency electromagnetic	JS	Current density
Field volume interface	FVIN	Field volume interface flag

***STLOC***

Starting location for entering *VAL* data, below. For example, if *STLOC* = 1, data input in the *VAL1* field applies to the first element body load item available for the element type, *VAL2* applies to the second element item, etc. If *STLOC* = 5, data input in the *VAL1* field applies to the fifth element item, etc. Defaults to 1.

***VAL1*, *VAL2*, *VAL3*, *VAL4***

For *Lab* = TEMP, FLUE, HGEN, and CHRGD, *VAL1--VAL4* represent body load values at the starting location and subsequent locations (usually nodes) in the element. *VAL1* can also represent a table name for use with tabular boundary conditions. Enter only *VAL1* for a uniform body load across the element.



For nonuniform loads, the values must be input in the same order as shown in the input table for the element type. Values initially default to the **BFUNIF** value (except for CHRGD which defaults to zero). For subsequent specifications, a blank leaves a previously specified value unchanged; if the value was not previously specified, the default value as described in the *Element Reference* is used.

For  $Lab = JS$  and  $STLOC = 1$ ,  $VAL1$ ,  $VAL2$  and  $VAL3$  are the X, Y, and Z components of current density (in the element coordinate system), and  $VAL4$  is the phase angle.

For  $Lab = EF$  and  $STLOC = 1$ ,  $VAL1$ ,  $VAL2$ , and  $VAL3$  are the X, Y, and Z components of electric field (in the global Cartesian coordinate system).

For  $Lab = VLTG$  and  $STLOC = 1$ ,  $VAL1$  is the voltage drop and  $VAL2$  is the phase angle.

If  $Lab = FVIN$  in a Multi-field solver (single or multiple code coupling) analysis,  $VAL1$  is the volume interface number. If  $Lab = FVIN$  in a unidirectional ANSYS to CFX analysis,  $VAL2$  is the volume interface number (not available from within the GUI) and  $VAL1$  is not used unless the ANSYS analysis is performed using the Multi-field solver.  $VAL3$  and  $VAL4$  are not used.

For  $Lab = FORC$  and  $STLOC = 1$ ,  $VAL1$ ,  $VAL2$ , and  $VAL3$  are the X, Y, and Z components of force density (in the global Cartesian coordinate system).

## Notes

Defines an element body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.). Body loads and element specific defaults are described for each element type in the *Element Reference*. If both the **BF** and **BFE** commands are used to apply a body load to an element, the **BFE** command takes precedence.

### Specifying a Table

You can specify a table name ( $VAL1$ ) only when using temperature (TEMP), heat generation rate (HGEN), and current density (JS) body load labels.

Enclose the table name ( $tabname$ ) in percent signs (%), as shown:

```
BFE,ELEM, Lab,STLOC,%tabname%
```

Use the **\*DIM** command to define a table.

For  $Lab = TEMP$ , each table defines  $NTEMP$  temperatures, as follows:

- For layered elements,  $NTEMP$  is the number of layer interface corners that allow temperature input.
- Beam and pipe elements are not supported.
- For non-layered elements,  $NTEMP$  is the number of corner nodes.

The temperatures apply to element items with a starting location of  $(STLOC + n)$ , where  $n$  is the value field location ( $VALn$ ) of the table name input.

For layered elements, a single **BFE** command returns temperatures for one layer interface. Multiple **BFE** commands are necessary for defining all layered temperatures.



When a tabular function load is applied to an element, the load will not vary according to the positioning of the element in space.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Volume Intr>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>Electric Field>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Elements**

**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Elements**

**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Elements**

**Main Menu>Solution>Define Loads>Apply>Field Volume Intr>On Elements**

**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Elements**

**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Elements**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Elements**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Elements**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Elements**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Other>Electric Field>On Elements**

**Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Elements**

**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Elements**

---

**BFECUM**, *Lab, Oper, FACT, TBASE*

**Specifies whether to ignore subsequent element body force loads.**

[SOLUTION: FE Body Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *Lab*

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature

Discipline	Body Load Label	Label Description
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	VLTG	voltage drop
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density

**Oper**

Replace or ignore key:

**REPL**

Subsequent values replace the previous values (default).

**IGNO**

Subsequent values are ignored.

**FACT**

Scale factor for the element body load values. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor. The scale factor is not applied to body load phase angles.

**TBASE**

Used (only with *Lab* = TEMP) to calculate the temperature used in the add or replace operation (see *Oper*) as:

$$\text{Temperature} = \text{TBASE} + \text{FACT} * (T - \text{TBASE})$$

where *T* is the temperature specified on subsequent **BFE** commands. *TBASE* defaults to zero.

**Command Default**

Replace previous values.

**Notes**

Allows repeated element body force loads to be replaced or ignored. Element body loads are applied with the **BFE** command. Issue the **BFELIST** command to list the element body loads. The operations occur when the next body loads are defined. For example, issuing the **BFE** command with a temperature value of 25 after a previous **BFE** command with a temperature value of 20 causes the new value of that temperature to be 25 with the replace operation, or 20 with the ignore operation. A scale factor is also available to multiply the next value before the replace operation. A scale factor of 2.0 with the previous "replace" example results in a temperature of 50. The scale factor is applied even if no previous values exist. Issue **BFECUM,STAT** to show the current label, operation, and scale factors.

**BFECUM** does not work for tabular boundary conditions.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Elem Body Lds**  
**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Elem Body Lds**

## BFEDELE, *ELEM*, *Lab*

Deletes element body force loads.

SOLUTION: FE Body Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *ELEM*

Element at which body load is to be deleted. If ALL, delete for all selected elements [ A component name may also be substituted for *ELEM*.

### *Lab*

Valid body load label. If ALL, use all appropriate labels. See **BFE** command for labels.

## Notes

Deletes element body force loads for a specified element and label. Element body loads may be defined with the **BFE** commands.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Elems**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Element Components**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Elements**

**Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Elems**

**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Elements**

**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Elements**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Elements**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Elements**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Elements**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Elements**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Elements**  
**Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Elements**  
**Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Element Components**  
**Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Elements**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Elements**

## BFELIST, *ELEM, Lab*

Lists the element body force loads.

[SOLUTION: FE Body Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *ELEM*

Element at which body load is to be listed. If ALL (or blank), list for all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*.

### *Lab*

Valid body load label. If ALL (or blank), use all appropriate labels. See **BFE** command for labels.

## Notes

Lists the element body force loads for the specified element and label. Element body loads may be defined with the **BFE** command.

This command is valid in any processor.

## Menu Paths

**Utility Menu>List>Loads>Body Loads>On All Elements**  
**Utility Menu>List>Loads>Body Loads>On Picked Elems**

## BFESCAL, *Lab, FACT, TBASE*

Scales element body force loads.

[SOLUTION: FE Body Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *Lab*

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	VLTG	voltage drop
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density

**FACT**

Scale factor for the element body load values. Zero (or blank) defaults to 1.0. Use a small number for a "zero" scale factor. The scale factor is not applied to body load phase angles.

**TBASE**

Base temperature for temperature difference. Used only with  $Lab = TEMP$ . Scale factor is applied to the temperature difference ( $T - TBASE$ ) and then added to  $TBASE$ .  $T$  is the current temperature.

**Notes**

Scales element body force loads on the selected elements in the database. Issue the **BFELIST** command to list the element body loads. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled. (Note that such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.)

**BFESCAL** does not work for tabular boundary conditions.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Elem Body Lds**

**Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Elem Body Lds**

**BFINT**, *Fname1*, *Ext1*, --, *Fname2*, *Ext2*, --, *KPOS*, *Clab*, *KSHS*, *TOLOUT*, *TOLHGT*

**Activates the body force interpolation operation.**

POST1: Special Purpose

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

***Fname1***

File name and directory path (248 characters maximum, including directory) from which to read data for interpolation. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext1***

Filename extension (8 character maximum).

The extension defaults to `NODE` if *Fname1* is blank.

--

Unused field.

***Fname2***

File name and directory path (248 characters maximum, including directory) to which **BF** commands are written. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext2***

Filename extension (8 character maximum).

The extension defaults to `BFIN` if *Fname2* is blank.

--

Unused field.

***KPOS***

Position on *Fname2* to write block of **BF** commands:

**0**

Beginning of file (overwrite existing file).

**1**

End of file (append to existing file).

***Clab***

Label (8 characters maximum, including the colon) for this block of **BF** commands in *Fname2*. This label is appended to the colon (:). Defaults to `BF $n$` , where  $n$  is the cumulative iteration number for the data set currently in the database.

***KSHS***

Shell-to-solid submodeling key:

**0**

Solid-to-solid or shell-to-shell submodel.

**1**

Shell-to-solid submodel.

**TOLOUT**

Extrapolation tolerance about elements, based on a fraction of the element dimension. Submodel nodes outside the element by more than *TOLOUT* are not accepted as candidates for DOF extrapolation. Defaults to 0.5 (50%).

**TOLHGT**

Height tolerance above or below shell elements, in units of length. Used only for shell-to-shell submodeling (*KSHS* = 0). Submodel nodes off of the element surface by more than *TOLHGT* are not accepted as candidates for DOF interpolation or extrapolation. Defaults to 0.0001 times the maximum element dimension.

**Caution**

Relaxing this tolerance to allow submodel nodes to be “found” could produce poor submodel results.

**Notes**

File *Fname1* should contain a node list for which body forces are to be interpolated [**NWRITE**]. File *Fname2* is created, and contains interpolated body forces written as a block of nodal **BF** commands.

Body forces are interpolated from elements having TEMP as a valid body force or degree of freedom, and only the label TEMP is written on the nodal **BF** commands. Interpolation is performed for all nodes on file *Fname1* using the results data currently in the database. For layered elements, use the **LAYER** command to select the locations of the temperatures to be used for interpolation. Default locations are the bottom of the bottom layer and the top of the top layer.

The block of **BF** commands begins with an identifying colon label command and ends with a **/EOF** command. The colon label command is of the form *:Clab*, where *Clab* is described above. Interpolation from multiple results sets can be performed by looping through the results file in a user-defined macro. Additional blocks can be appended to *Fname2* by using *KPOS* and unique colon labels. A **/INPUT** command, with the appropriate colon label, may be used to read the block of commands.

**Menu Paths**

**Main Menu>General Postproc>Submodeling>Interp Body Forc**

**BFK**, *KPOI*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *PHASE*

**Defines a body force load at a keypoint.**

[SOLUTION: Solid Body Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**KPOI**

Keypoint to which body load applies. If ALL, apply to all selected keypoints [**KSEL**]. A component name may also be substituted for *KPOI*.

**Lab**

Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the [Element Reference](#).

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density
High-frequency electromagnetic	JS	current density

All keypoints on a given area (or volume) must have the same **BFK** table name for the tables to be transferred to interior nodes.

#### **VAL1, VAL2, VAL3**

Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, MVDI and CHRGD. Use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFK,KPOI,Lab,%tablename%**. Use the **\*DIM** command to define a table.

#### **PHASE**

Phase angle in degrees associated with the JS label.

## Notes

Defines a body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) at a keypoint. Body loads may be transferred from keypoints to nodes with the **BFTRAN** or **SBCTRAN** commands. Interpolation will be used to apply loads to the nodes on the lines between keypoints. All keypoints on a given area (or volume) must have the same **BFK** specification, with the same values, for the loads to be transferred to interior nodes in the area (or volume). If only one keypoint on a line has a **BFK** specification, the other keypoint defaults to the value specified on the **BFUNIF** command.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels.

Body loads specified by the **BFK** command can conflict with other specified body loads. See [Resolution of Conflicting Body Load Specifications](#) in the *Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.



## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Key-  
 points  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On  
 Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On  
 Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On  
 Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Keypoints

## BFKDELE, *KPOI*, *Lab*

Deletes body force loads at a keypoint.

SOLUTION: Solid Body Loads

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *KPOI*

Keypoint at which body load is to be deleted. If ALL, delete for all selected keypoints [**KSEL**]. A component name may also be substituted for *KPOI*.

### *Lab*

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFK** command for labels.

## Notes

Deletes body force loads (and all corresponding finite element loads) for a specified keypoint and label. Body loads may be defined at a keypoint with the **BFK** command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All KPs

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Key-  
 points**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On  
 Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On  
 Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On  
 Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All KPs**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Keypoints**

## BFKLIST, *KPOI*, *Lab*

Lists the body force loads at keypoints.

[SOLUTION: Solid Body Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *KPOI*

Keypoint at which body load is to be listed. If ALL (or blank), list for all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*

### *Lab*

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the [Element Reference](#). See the **BFK** command for labels.

## Notes

Lists the body force loads for the specified keypoint and label. Keypoint body loads may be defined with the **BFK** command.

This command is valid in any processor.

## Menu Paths

**Utility Menu>List>Loads>Body Loads>On All Keypoints**  
**Utility Menu>List>Loads>Body Loads>On Picked KPs**

**BFL**, *LINE*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

**Defines a body force load on a line.**

[SOLUTION: Solid Body Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**LINE**

Line to which body load applies. If ALL, apply to all selected lines [**LSEL**]. A component name may also be substituted for *LINE*.

**Lab**

Valid body load label. Load labels are listed under "Body loads" in the input table for each element type in the [Element Reference](#).

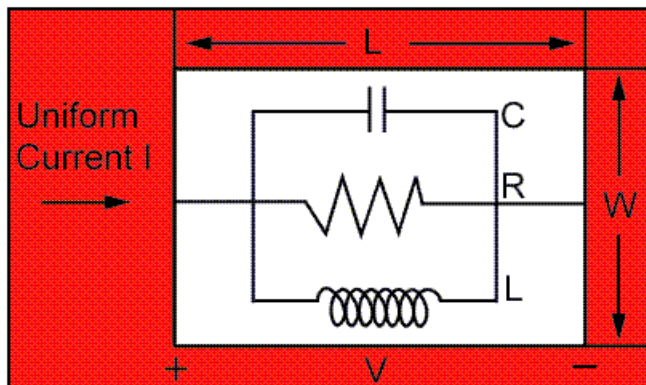
Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	CHRGD	charge density
Electric	TEMP	temperature
	CHRGD	charge density
High-frequency electromagnetic	JS	current density
	EF	electric field
	CHRGD	inner conductor flag
	IMPD	impedance sheet
	LUMP	shunt RCL lumped circuit
FLOTRAN	HGEN	heat generation rate
	FORC	body force density

**VAL1, VAL2, VAL3**

Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, and CHRGD. If *Lab* = CHRGD for a high-frequency inner conductor flag, *VAL1* is the inner conductor number (no default). If *Lab* = IMPD, *VAL1* is the resistance and *VAL2* is the reactance in ohms/square. If *Lab* = LUMP, *VAL1* is the resistance, *VAL2* is the capacitance, and *VAL3* is the inductance of the shunt RCL lumped circuit. Use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS, EF, and FORC. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFL,LINE,Lab,%tablename%**. Use the **\*DIM** command to define a table.

**VAL4**

If *Lab* = EF, *VAL4* is the phase angle in degrees. If *Lab* = JS, *VAL4* is the phase angle in degrees or a negative port number for a driven port. If *Lab* = LUMP, *VAL4* is the length (L) of a shunt RCL lumped circuit with uniform current (I).  $W = 1$ .

**Figure 3 RCL Lumped Circuit**

## Notes

Defines a body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) on a line. Body loads may be transferred from lines to line elements (or to nodes if line elements do not exist) with the **BFTRAN** or **SBCTTRAN** commands.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels.

Body loads specified by the **BFL** command can conflict with other specified body loads. See [Resolution of Conflicting Body Load Specifications](#) in the *Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppElecField>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppElecField>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Lines**

**BFLDELE**, *LINE*, *Lab*

Deletes body force loads on a line.

[SOLUTION: Solid Body Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

***LINE***

Line at which body load is to be deleted. If ALL, delete for all selected lines [**LSEL**]. A component name may also be substituted for *LINE*.

***Lab***

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the [Element Reference](#). See the **BFL** command for labels.

**Notes**

Deletes body force loads (and all corresponding finite element loads) for a specified line and label. Body loads may be defined on a line with the **BFL** command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelElecField>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Lines**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Lines**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelElecField>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Lines**

**BFLIST**, *NODE*, *Lab*

**Lists the body force loads on nodes.**

[SOLUTION: FE Body Loads](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NODE**

Node at which body load is to be listed. If ALL (or blank), list for all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

**Lab**

Valid body load label. If ALL (or blank), use all appropriate labels. See the **BF** command for labels. In an explicit dynamic analysis, the only valid body load label is TEMP.

## Notes

Lists the body force loads for the specified node and label. Nodal body loads may be defined with the **BF** command (except in an explicit dynamic analysis).

The command **BFLIST**,TEMP can be used in an explicit dynamic analysis to list temperature loads that are read in by the **LDREAD** command. **BFLIST** cannot be used to list temperature loads defined by the **EDLOAD** command (use **EDLOAD**,LIST to list this type of load).

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>List Temps**

**Main Menu>Solution>Loading Options>List Temps**

**Utility Menu>List>Loads>Body Loads>On All Nodes**

**Utility Menu>List>Loads>Body Loads>On Picked Nodes**

---

**BFLLIST**, *LINE*, *Lab*

**Lists the body force loads on a line.**

[SOLUTION: Solid Body Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**LINE**

Line at which body load is to be listed. If ALL (or blank), list for all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*.

**Lab**

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFL** command for labels.

## Notes

Lists the body force loads for the specified line and label. Body loads may be defined on a line with the **BFL** command.

This command is valid in any processor.

## Menu Paths

**Utility Menu>List>Loads>Body Loads>On All Lines**  
**Utility Menu>List>Loads>Body Loads>On Picked Lines**

**BFSCALE**, *Lab*, *FACT*, *TBASE*

**Scales body force loads at nodes.**

[SOLUTION: FE Body Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *Lab*

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density

### *FACT*

Scale factor for the nodal body load values. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor. The scale factor is not applied to body load phase angles.

### *TBASE*

Base temperature for temperature difference. Used only with *Lab* = TEMP. Scale factor is applied to the temperature difference ( $T - TBASE$ ) and then added to *TBASE*. *T* is the current temperature.

## Notes

Scales body force loads in the database on the selected nodes. Issue the **BFLIST** command to list the nodal body loads. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

**Note**

Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

**BFSCALE** does not work for tabular boundary conditions.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Nodal Body Ld**  
**Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Nodal Body Ld**

**BFTRAN**

**Transfers solid model body force loads to the finite element model.**

[SOLUTION: Solid Body Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**Notes**

Body loads are transferred from selected keypoints and lines to selected nodes and from selected areas and volumes to selected elements. The **BFTRAN** operation is also done if the **SBCTRAN** command is either explicitly issued or automatically issued upon initiation of the solution calculations [**SOLVE**].

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Body Loads**  
**Main Menu>Solution>Define Loads>Operate>Transfer to FE>Body Loads**

**BFUNIF**, *Lab*, *VALUE*

**Assigns a uniform body force load to all nodes.**

[SOLUTION: FE Body Loads](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Lab***

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence



Discipline	Body Load Label	Label Description
Explicit dynamics	TEMP	temperature
Thermal	HGEN	heat generation rate
	TEMP	initial temperature

**VALUE**

Uniform value associated with *Lab* item, or table name when specifying tabular boundary conditions. To specify a table, enclose the table name in percent signs (%), e.g., **BFUNIF**,*Lab*,%*tablename*%.

**Command Default**

Set TEMP to the reference temperature ([**TREF**] but not **MP**,**REFT**), and FLUE and HGEN to zero.

**Notes**

In a transient or nonlinear thermal analysis, the uniform temperature is used during the first iteration of a solution as follows: (a) as the starting nodal temperature (except where temperatures are explicitly specified [**D**, **DK**]), and (b) to evaluate temperature-dependent material properties. In a structural analysis or explicit dynamic analysis, the uniform temperature is used as the *default* temperature for thermal strain calculations and material property evaluation (except where body load temperatures are specified [**BF**, **BFE**, **BFK**, **LDREAD**]). In other scalar field analyses, the uniform temperature is used for material property evaluation.

When the command **BFUNIF**,TEMP is used in an explicit dynamic analysis, you cannot use the **EDLOAD**,TEMP command to apply temperature loading. Furthermore, any temperature loading defined by **BFUNIF** cannot be listed or deleted by the **EDLOAD** command.

An alternate command, **TUNIF**, may be used to set the uniform temperature instead of **BFUNIF**,TEMP. Since **TUNIF** (or **BFUNIF**,TEMP) is step-applied in the first iteration, you should use **BF**, ALL, TEMP, Value to ramp on a uniform temperature load.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels. When using TEMP, you can define a one-dimensional table that varies with respect to time (TIME) only. When defining this table, enter TIME as the primary variable. No other primary variables are valid. Tabular boundary conditions cannot be used in an explicit dynamic analysis.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>Uniform Heat Gen**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>Uniform Fluen**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>Uniform Heat Gen**

**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>Uniform Heat Gen**

**Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>Uniform Fluen**

**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>Uniform Heat Gen**

**BFV**, *VOLU*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *PHASE*

**Defines a body force load on a volume.**

SOLUTION: Solid Body Loads

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### ***VOLU***

Volume to which body load applies. If ALL, apply to all selected volumes [**VSEL**]. A component name may also be substituted for *VOLU*.

#### ***Lab***

Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the [Element Reference](#).

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	VLTG	voltage drop
Electric	TEMP	temperature
	CHRGD	charge density
High-frequency electromagnetic	JS	current density
FLOTRAN	HGEN	heat generation rate

#### ***VAL1*, *VAL2*, *VAL3***

Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, and CHRGD. Use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS. For *Lab* = VLTG, *VAL1* is the voltage drop and *VAL2* is the phase angle. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFV**,*VOLU*,*Lab*,%*tablename*%. Use the **\*DIM** command to define a table.

#### ***PHASE***

Phase angle in degrees associated with the JS label.

## Notes

Defines a body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) on a volume. Body loads may be transferred from volumes to volume elements (or to nodes if volume elements do not exist) with the **BFTRAN** or **SBCTAN** commands. Body loads default to the value specified on the **BFUNIF** command, if it was previously specified.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels.

Body loads specified by the **BFV** command can conflict with other specified body loads. See [Resolution of Conflicting Body Load Specifications](#) in the *Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Volumes**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Volumes**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Volumes**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Volumes**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Volumes**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Volumes**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Volumes**

**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Volumes**

**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Volumes**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Volumes**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Volumes**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Volumes**

**Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Volumes**

**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Volumes**

## BFVDELE, *VOLU*, *Lab*

**Deletes body force loads on a volume.**

**SOLUTION:** Solid Body Loads

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *VOLU*

Volume at which body load is to be deleted. If ALL, delete for all selected volumes [**VSEL**]. A component name may also be substituted for *VOLU*.

### *Lab*

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the [Element Reference](#). See the **BFV** command for labels.

## Notes

Deletes body force loads (and all corresponding finite element loads) for a specified volume and label. Body loads may be defined on a volume with the **BFV** command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Volms  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Volumes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Volumes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Volumes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Volumes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Volumes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Volumes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Volumes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Volumes  
 Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Volms  
 Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Volumes  
 Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Volumes  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Volumes  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Volumes  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Volumes  
 Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Volumes  
 Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Volumes  
 Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Volumes

## BFVLIST, *VOLU*, *Lab*

Lists the body force loads on a volume.

[SOLUTION: Solid Body Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *VOLU*

Volume at which body load is to be listed. If ALL (or blank), list for all selected volumes [**VSEL**]. If *VOLU* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *VOLU*.

### *Lab*

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the [Element Reference](#). See the **BFV** command for labels.

## Notes

Lists the body force loads for the specified volume and label. Body loads may be defined on a volume with the **BFV** command.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>Body Loads>On All Volumes  
Utility Menu>List>Loads>Body Loads>On Picked Volumes

## BIOOPT

Specifies "Biot-Savart options" as the subsequent status topic.

SOLUTION: Status

MP <> <> <> <> <> <> <> EM <> DY PP <> EME <>

## Notes

This is a status [STAT] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Solution>Biot Savart Options

## BIOT, Label

Calculates the Biot-Savart source magnetic field intensity.

SOLUTION: Misc Loads

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

### Label

Controls the Biot-Savart calculation:

#### NEW

Calculate the magnetic source field intensity ( $H_s$ ) from the selected set of source elements to the selected set of nodes. Overwrite any existing  $H_s$  field values.

#### SUM

Calculate the  $H_s$  field from the selected set of source elements to the selected set of nodes. Accumulate with any existing  $H_s$  field values.

## Command Default

Calculate the  $H_s$  field upon encountering the first **SOLVE** command to produce a source field.

## Notes

Calculates the Biot-Savart source magnetic field intensity ( $H_s$ ) at the selected nodes from the selected source elements. The calculation is done at the time the **BIOT** command is issued.

Source elements include primitives described by element **SOURC36**, and coupled-field elements **SOLID5**, **LINK68**, and **SOLID98**. Current conduction elements do not have a solved-for current distribution from which to calculate a source field until after the first substep. Inclusion of a current conduction element  $H_s$  field will require a subsequent **BIOT,SUM** command (with **SOURC36** elements unselected) and a **SOLVE** command.

The units of  $H_s$  are as specified by the current **EMUNIT** command setting.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>Biot-Savart**  
**Main Menu>Solution>Load Step Opts>Magnetics>Options Only>Biot-Savart**

## **BLC4**, *XCORNER*, *YCORNER*, *WIDTH*, *HEIGHT*, *DEPTH*

**Creates a rectangular area or block volume by corner points.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *XCORNER*, *YCORNER*

Working plane X and Y coordinates of one corner of the rectangle or block face.

### *WIDTH*

The distance from *XCORNER* on or parallel to the working plane X-axis that, together with *YCORNER*, defines a second corner of the rectangle or block face.

### *HEIGHT*

The distance from *YCORNER* on or parallel to the working plane Y-axis that, together with *XCORNER*, defines a third corner of the rectangle or block face.

### *DEPTH*

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the block. If *DEPTH* = 0 (default), a rectangular area is created on the working plane.

## Notes

Defines a rectangular area anywhere on the working plane or a hexahedral volume with one face anywhere on the working plane. A rectangle will be defined with four keypoints and four lines. A volume will be defined with eight keypoints, twelve lines, and six areas, with the top and bottom faces parallel to the working plane. See the **BLC5**, **RECTNG**, and **BLOCK** commands for alternate ways to create rectangles and blocks.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By 2 Corners**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Block>By 2 Corners & Z**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Block>By 2 Corners & Z**

**BLC5**, *XCENTER*, *YCENTER*, *WIDTH*, *HEIGHT*, *DEPTH*

**Creates a rectangular area or block volume by center and corner points.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***XCENTER*, *YCENTER***

Working plane X and Y coordinates of the center of the rectangle or block face.

***WIDTH***

The total distance on or parallel to the working plane X-axis defining the width of the rectangle or block face.

***HEIGHT***

The total distance on or parallel to the working plane Y-axis defining the height of the rectangle or block face.

***DEPTH***

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the block. If *DEPTH* = 0 (default), a rectangular area is created on the working plane.

### Note

If you are working with a model imported from an IGES file (import option set to DEFAULT), you must supply a value for *DEPTH* or the command is ignored.

## Notes

Defines a rectangular area anywhere on the working plane or a hexahedral volume with one face anywhere on the working plane by specifying the center and corner points. A rectangle will be defined with four keypoints and four lines. A volume will be defined with eight keypoints, twelve lines, and six areas, with the top and bottom faces parallel to the working plane. See the **BLC4**, **RECTNG**, and **BLOCK** commands for alternate ways to create rectangles and blocks.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By Centr & Cornr**

**Main Menu>Preprocessor>Modeling>Create>Primitives>Block**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Block>By Centr,Cornr,Z**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Block>By Centr,Cornr,Z**

**BLOCK**,  $X1, X2, Y1, Y2, Z1, Z2$ 

Creates a block volume based on working plane coordinates.

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

$X1, X2$

Working plane X coordinates of the block.

$Y1, Y2$

Working plane Y coordinates of the block.

$Z1, Z2$

Working plane Z coordinates of the block.

## Notes

Defines a hexahedral volume based on the working plane. The block must have a spatial volume greater than zero (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) The volume will be defined with eight keypoints, twelve lines, and six areas, with the top and bottom faces parallel to the working plane. See the [BLC4](#) and [BLC5](#) commands for alternate ways to create blocks.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Volumes>Block>By Dimensions**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Block>By Dimensions**

---

## BOOL

Specifies "Booleans" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [\[STAT\]](#) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the [STAT](#) command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>Preprocessor>Solid Model**

---



**BOPTN**, *Lab*, *Value*

**Specifies Boolean operation options.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Lab***

Default/status key:

**DEFA**

Resets settings to default values.

**STAT**

Lists status of present settings.

Option to be controlled:

**KEEP**

Delete or keep input entity option.

**NUMB**

Output numbering warning message option.

**NWARN**

No effect warning message option.

**VERSION**

Boolean compatibility option.

***Value***

Option settings if *Lab* = KEEP:

**NO**

Delete entities used as input with a Boolean operation (default). Entities will not be deleted if meshed or if attached to a higher entity.

**YES**

Keep input solid modeling entities.

Option settings if *Lab* = NUMB:

**0**

No warning message will be produced if the output entities of a Boolean operation are numbered based on geometry (default).

**1**

A warning message will be produced if the output entities of a Boolean operation are numbered based on geometry. (With geometric numbering, re-use of the input with altered dimensions (as in design optimization) may not produce the same numbering, and later operations in the input may fail or produce unexpected results.)

Option settings if *Lab* = NWARN:

**0**

A warning message will be produced if a Boolean operation has no effect (default).

**1**

No warning or error messages will be generated if a Boolean operation has no effect.

-1

An error message will be produced if a Boolean operation has no effect.

Option settings if *Lab* = VERSION:

### **RV52**

Activate the Revision 5.2 compatibility option (default). The 5.2 option can produce different numbering of the entities produced by Boolean operations than the 5.1 option. See Notes below.

### **RV51**

Activate the Revision 5.1 compatibility option. The 5.1 option can produce different numbering of the entities produced by Boolean operations than the 5.2 option. See "*Notes*" (p. 222) below.

## **Command Default**

Input entities will be deleted, and operations with no effect (i.e., operations which are valid but which do not cause a change in the input entities, such as adding two non-touching areas) will produce a warning message. The Revision 5.2 Boolean compatibility option will be used.

## **Notes**

Boolean operations at Revision 5.2 may produce a different number of entities than previous revisions of ANSYS. When running input files created at earlier revisions of ANSYS, match the Boolean compatibility option (VERSION) to the revision originally used. For instance, if you are running Revision 5.2 and are reading an input file (**/INPUT**) created at Revision 5.1, it is recommended that you set VERSION to RV51 before reading the input.

See the *Modeling and Meshing Guide* for further details on the functions of the RV51 and RV52 labels.

This command is valid in any processor.

## **Menu Paths**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Settings**

---

**BSAX**, VAL1, VAL2, T

**Specifies the axial strain and axial force relationship for beam sections.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**VAL1**

Axial strain component ( $\epsilon$ ).

**VAL2**

Axial force component ( $N$ ).

**T**

Temperature.

## Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon, T) & & & & & \\ & I_1 E(\kappa_1, T) & & & & \\ & & I_2 E(\kappa_2, T) & & & \\ & & & JG(\chi, T) & & \\ & & & & A_1 G(\gamma_1, T) & \\ & 0 & & & & A_2 G(\gamma_2, T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSAX** command, one of several [nonlinear general beam section commands](#), specifies the relationship of axial strain and axial force for a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified values default to zero.

Related commands are **BSM1**, **BSM2**, **BSTQ**, **BSS1**, **BSS2**, **BSMD**, and **BSTE**.

For complete information, see [Using Nonlinear General Beam Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Beam>NL Generalized**

## BSMD, DENS, T

**Specifies mass density for a nonlinear general beam section.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### DENS

Mass density.

### T

Temperature.

## Notes

The **BSMD** command, one of several [nonlinear general beam section commands](#), specifies the mass density (assuming a unit area) for a beam section. The value specified is associated with the section most recently defined (via the **SECTYPE** command).

Related commands are **BSAX**, **BSM1**, **BSM2**, **BSTQ**, **BSS1**, **BSS2**, and **BSTE**.

For complete information, see [Using Nonlinear General Beam Sections](#).

## Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

**BSM1**, VAL1, VAL2, T

Specifies the bending curvature and moment relationship in plane XZ for beam sections.

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**VAL1**

Curvature component ( $\kappa_1$ ).

**VAL2**

Bending moment component ( $M_1$ ).

**T**

Temperature.

## Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon, T) & & & & & & \\ & I_1 E(\kappa_1, T) & & & & & \\ & & I_2 E(\kappa_2, T) & & & & \\ & & & JG(\chi, T) & & & \\ & & & & A_1 G(\gamma_1, T) & & \\ & 0 & & & & A_2 G(\gamma_2, T) & \\ & & & & & & \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSM1** command, one of several [nonlinear general beam section commands](#), specifies the bending curvature and moment for plane XZ of a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified values default to zero.

Related commands are **BSAX**, **BSM2**, **BSTQ**, **BSS1**, **BSS2**, **BSMD**, and **BSTE**.

For complete information, see [Using Nonlinear General Beam Sections](#).

## Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

**BSM2**, VAL1, VAL2, T

**Specifies the bending curvature and moment relationship in plane XY for beam sections.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**VAL1**

Curvature component ( $\kappa_2$ ).

**VAL2**

Bending moment component ( $M_2$ ).

**T**

Temperature.

**Notes**

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon, T) & & & & & & \\ & I_1 E(\kappa_1, T) & & & & & \\ & & I_2 E(\kappa_2, T) & & & & \\ & & & JG(\chi, T) & & & \\ & & & & A_1 G(\gamma_1, T) & & \\ & & & & & A_2 G(\gamma_2, T) & \\ & & & & & & \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSM2** command, one of several [nonlinear general beam section commands](#), specifies the bending curvature and moment relationship for plane XY of a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified values default to zero.

Related commands are **BSAX**, **BSM1**, **BSTQ**, **BSS1**, **BSS2**, **BSMD**, and **BSTE**.

For complete information, see [Using Nonlinear General Beam Sections](#).

**Menu Paths**

**Main Menu>Preprocessor>Sections>Beam>NL Generalized**

**BSPLIN**,  $P1, P2, P3, P4, P5, P6, XV1, YV1, ZV1, XV6, YV6, ZV6$ 

**Generates a single line from a spline fit to a series of keypoints.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

 **$P1, P2, P3, \dots, P6$** 

Keypoints through which a spline is fit. At least two keypoints must be defined. If  $P1 = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

The following fields are used only if specified end slopes are desired; otherwise, zero curvature end slopes will be automatically calculated. The slope vector is parallel to a vector pointing from the origin of the active coordinate system [**CSYS**] to the position in space that  $XV, YV, ZV$  represents in that system.

 **$XV1, YV1, ZV1$** 

Orientation point of an outward vector tangent to line at  $P1$ . Vector coordinate system has its origin at the keypoint. Coordinate interpretation corresponds to the active coordinate system type, i.e., X is R for cylindrical, etc. Defaults to zero curvature slope.

 **$XV6, YV6, ZV6$** 

Orientation point of an outward vector tangent to a line at  $P6$  (or the last keypoint specified if fewer than six specified). Defaults to zero curvature slope.

## Notes

One line is generated between keypoint  $P1$  and the last keypoint entered. The line will pass through each entered keypoint. Solid modeling in a toroidal coordinate system is not recommended.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Lines>Splines>Spline thru KPs**

**Main Menu>Preprocessor>Modeling>Create>Lines>Splines>Spline thru Locs**

**Main Menu>Preprocessor>Modeling>Create>Lines>Splines>With Options>Spline thru KPs**

**Main Menu>Preprocessor>Modeling>Create>Lines>Splines>With Options>Spline thru Locs**

**BSS1**,  $VAL1, VAL2, T$ 

**Specifies the transverse shear strain and force relationship in plane XZ for beam sections.**

[PREP7:Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

 **$VAL1$** 

Transverse shear strain component ( $\gamma_1$ ).

 **$VAL2$** 

Transverse shear force component ( $S_1$ ).

 **$T$** 

Temperature.

## Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon, T) & & & & & \\ & I_1 E(\kappa_1, T) & & & & \\ & & I_2 E(\kappa_2, T) & & & \\ & & & JG(\chi, T) & & \\ & & & & A_1 G(\gamma_1, T) & \\ & 0 & & & & A_2 G(\gamma_2, T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSS1** command, one of several [nonlinear general beam section commands](#), specifies the transverse shear strain and transverse shear force relationship for plane XZ of a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified values default to zero.

Related commands are **BSAX**, **BSM1**, **BSM2**, **BSTQ**, **BSS2**, **BSMD**, and **BSTE**.

For complete information, see [Using Nonlinear General Beam Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Beam>NL Generalized**

**BSS2**, VAL1, VAL2, T

**Specifies the transverse shear strain and force relationship in plane XY for beam sections.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**VAL1**

Transverse shear strain component ( $\gamma_2$ ).

**VAL2**

Transverse shear force component ( $S_2$ ).

**T**

Temperature.

## Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon, T) & & & & & \\ & I_1 E(\kappa_1, T) & & & & \\ & & I_2 E(\kappa_2, T) & & & \\ & & & JG(\chi, T) & & \\ & & & & A_1 G(\gamma_1, T) & \\ & 0 & & & & A_2 G(\gamma_2, T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSS1** command, one of several [nonlinear general beam section commands](#), specifies the transverse shear strain and transverse shear force relationship for plane XY of a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified values default to zero.

Related commands are **BSAX**, **BSM1**, **BSM2**, **BSTQ**, **BSS1**, **BSMD**, and **BSTE**.

For complete information, see [Using Nonlinear General Beam Sections](#).

## Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

## BSTE, ALPHA, T

Specifies a thermal expansion coefficient for a nonlinear general beam section.

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### ALPHA

Coefficient of thermal expansion for the cross section.

### T

Temperature.

## Notes

The **BSTE** command, one of several [nonlinear general beam section commands](#), specifies a thermal expansion coefficient for a beam section. The value specified is associated with the section most recently defined (via the **SECTYPE** command).

Related commands are **BSAX**, **BSM1**, **BSM2**, **BSTQ**, **BSS1**, **BSS2**, and **BSMD**.

For complete information, see [Using Nonlinear General Beam Sections](#).

## Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized



**BSTQ**, VAL1, VAL2, T**Specifies the cross section twist and torque relationship for beam sections.**

PREP7: Cross Sections

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

**VAL1**Twist component ( $\chi$ ).**VAL2**Torque component ( $\tau$ ).**T**

Temperature.

## Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon, T) & & & & & & \\ & I_1 E(\kappa_1, T) & & & & & \\ & & I_2 E(\kappa_2, T) & & & & \\ & & & JG(\chi, T) & & & \\ & & & & A_1 G(\gamma_1, T) & & \\ & & & & & A_2 G(\gamma_2, T) & \\ & & & & & & \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSTQ** command, one of several [nonlinear general beam section commands](#), specifies the cross section twist and torque relationship for a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified values default to zero.

Related commands are **BSAX**, **BSM1**, **BSM2**, **BSS1**, **BSS2**, **BSMD**, and **BSTE**.

For complete information, see [Using Nonlinear General Beam Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Beam>NL Generalized**

**BTOL**, *PTOL*

**Specifies the Boolean operation tolerances.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***PTOL***

Point coincidence tolerance. Points within this distance to each other will be assumed to be coincident during Boolean operations. Loosening the tolerance will increase the run time and storage requirements, but will allow more Boolean intersections to succeed. Defaults to 0.10E-4.

**Command Default**

*PTOL* = 0.10E-4.

**Notes**

Use **BTOL,DEFA** to reset the setting to its default value. Use **BTOL,STAT** to list the status of the present setting.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Settings**

**BUCOPT**, *Method, NMODE, SHIFT, LDMULTE*

**Specifies buckling analysis options.**

[SOLUTION: Nonlinear Options](#)

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

***Method***

Mode extraction method to be used (no default; you must specify the method). Currently, Block Lanczos is the only available method:

**LANB**

Block Lanczos.

***NMODE***

Number of modes to extract (defaults to 1).

***SHIFT***

Lower end of the load multiplier range of interest. This value also acts as the initial shift point about which eigenvalues are calculated (defaults to 0.0).

When *LDMULTE* = CENTER, the *SHIFT* value will determine the center value about which the modes will be extracted. A blank value yields a center value equal to zero (default).

***LDMULTE***

Upper end of the load multiplier range of interest (defaults to  $+\infty$ ). By default, the program calculates buckling modes from 0.0 to positive infinity.

If *LDMULTE* = CENTER, *NMODE* modes will be extracted about the center value on both the left and right ends.

---

## Notes

This command is also valid in PREP7. If used in SOLUTION, this command is valid only within the first load step. The PCG Lanczos method is NOT available for buckling analyses.

We recommend that you request an additional few modes beyond what is needed in order to enhance the accuracy of the final solution. We also recommend that you input a non zero *SHIFT* value and a reasonable *LDMULTE* value when numerical problems are encountered.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>Analysis Options**

---



---

## C Commands

**C\*\*\***, *Comment*

**Places a comment in the output.**

[SESSION: List Controls](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

*Comment*

Comment string, up to 75 characters.

### Notes

The output from this command consists of two lines -- a blank line followed by a line containing C\*\*\* and the comment. This command is similar to **/COM** except that the comment produced by C\*\*\* is more easily identified in the output.

Another way to include a comment is to precede it with a ! character (on the same line). The ! may be placed anywhere on the line, and any input following it is ignored as a comment. No output is produced by such a comment, but the comment line is included on the log file. This is a convenient way to annotate the log file.

This command is valid anywhere.

### Menu Paths

**This command cannot be accessed from a menu.**

---

## CALC

**Specifies "Calculation settings" as the subsequent status topic.**

[POST1: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>General Postproc>Calculations

---

### **CAMPBELL**, *Key*, *NSOLVE*

Prepares the result file for a subsequent Campbell diagram of a prestressed structure.

[SOLUTION: Load Step Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### **Key**

Campbell key:

#### **OFF**

Do not prepare the result file. This option is the default behavior.

#### **ON**

Prepare the result file for a subsequent [Campbell diagram](#) of a prestressed structure.

#### **NSOLVE**

Number of modal analyses to be performed. The same number of static analyses is expected.

## Notes

For an analysis involving a prestressed structure, the **CAMPBELL** command specifies whether or not to prepare the result file to support a [Campbell diagram](#) analysis (**PRCAMP** or **PLCAMP**).

To prestress a structure, ANSYS performs a static solution before the modal solution. In such an analysis, the default result file is unusable for a subsequent Campbell diagram analysis. To use the result file in this case, issue a **CAMPBELL,ON** command in the first static analysis. The command forces the result file to keep load steps from the modal analyses only (and discards results from the static analyses).

The **CAMPBELL** command requires that modal and static analyses be performed alternately. It works only when the number of static analyses is the same as the number of modal analyses. Any number of analyses can be performed, but the same number of each (static and modal) is expected.

For an example of **PLCAMP** command usage, see [Sample Campbell Diagram Analysis](#).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**CBDOF**, *Fname1*, *Ext1*, --, *Fname2*, *Ext2*, --, *KPOS*, *Clab*, *KSHS*, *TOLOUT*, *TOLHGT*, *TOLTHK*

**Activates cut boundary interpolation (for submodeling).**

POST1: Special Purpose

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

***Fname1***

File name and directory path (248 characters maximum, including directory) from which to read boundary node data. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext1***

Filename extension (8 character maximum).

The extension defaults to `NODE` if *Fname1* is blank.

--

Unused field.

***Fname2***

File name and directory path (248 characters maximum, including directory) to which cut boundary **D** commands are written. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext2***

Filename extension (8 character maximum).

The extension defaults to `CBDO` if *Fname2* is blank.

--

Unused field.

***KPOS***

Position on *Fname2* to write block of **D** commands:

**0**

Beginning of file (overwrite existing file).

**1**

End of file (append to existing file).

***Clab***

Label (8 characters maximum, including the colon) for this block of **D** commands on *Fname2*. This label is appended to the colon (:). Defaults to `CB $n$` , where  $n$  is the cumulative iteration number for the data set currently in the database. For imaginary data (see *KIMG* on the **\*SET** command), *Clab* defaults to `Cl $n$` .

***KSHS***

Shell-to-solid submodeling key:

**0**

Solid-to-solid or shell-to-shell submodel.

**1**

Shell-to-solid submodel.

***TOLOUT***

Extrapolation tolerance about elements, based on a fraction of the element dimension. Submodel nodes outside the element by more than *TOLOUT* are not accepted as candidates for DOF extrapolation. Defaults to 0.5 (50%).

***TOLHGT***

Height tolerance above or below shell elements, in units of length. Used only for shell-to-shell submodeling (*KSHS* = 0). Submodel nodes off of the element surface by more than *TOLHGT* are not accepted as candidates for DOF interpolation or extrapolation. Defaults to 0.0001 times the maximum element dimension.

---

**Caution**

Relaxing this tolerance to allow submodel nodes to be "found" could produce poor submodel results.

***TOLTHK***

Height tolerance above or below shell elements, based on a fraction of the shell element thickness. Used only for shell-to-solid submodeling (*KSHS* = 1). Submodel nodes off of the element surface by more than *TOLTHK* are not accepted as candidates for DOF interpolation or extrapolation. Defaults to 0.1 times the average shell thickness.

---

**Caution**

Relaxing this tolerance to allow submodel nodes to be "found" could produce poor submodel results.

**Notes**

File *Fname1* should contain a node list for which boundary conditions are to be interpolated [**NWRITE**]. File *Fname2* is created which contains interpolated boundary conditions written as a block of **D** commands. Boundary conditions are written for the active degree of freedom set for the element from which interpolation is performed. Interpolation is performed on the selected set of elements. The block of **D** commands begins with an identifying colon label and ends with a **/EOF** command. The colon label is of the form *:Clab*, where *Clab* is described above. Interpolation from multiple results sets can be performed by looping through the results file in a user-defined macro. Additional blocks can be appended to *Fname2* by using *KPOS* and unique colon labels. A **/INPUT** command, with the appropriate colon label, may be used to read the block of commands.

**Menu Paths**

**Main Menu>General Postproc>Submodeling>Interpolate DOF**

---



**CBMD**, *ROW*,  $C_{(R)(R)}$ ,  $C_{(R)(R+1)}$ ,  $C_{(R)(R+2)}$ ,  $C_{(R)(R+3)}$ ,  $C_{(R)(R+4)}$ ,  $C_{(R)(R+5)}$

**Specifies preintegrated section mass matrix for composite-beam sections.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**ROW**

Row number of the matrix.

$C_{(R)(R)}$ , ...,  $C_{(R)(R+5)}$

Upper triangle of the cross-section mass matrix **[C]**.

## Notes

With a unit beam length, the section mass matrix relates the resultant forces and torques to accelerations and angular accelerations as follows (applicable to the local element coordinate system):

$$\begin{bmatrix} N_x \\ N_y \\ N_z \\ M_x \\ M_y \\ M_z \end{bmatrix} = \begin{bmatrix} C_{11}(T) & C_{12}(T) & C_{13}(T) & C_{14}(T) & C_{15}(T) & C_{16}(T) \\ & C_{22}(T) & C_{23}(T) & C_{24}(T) & C_{25}(T) & C_{26}(T) \\ & & C_{33}(T) & C_{34}(T) & C_{35}(T) & C_{36}(T) \\ & & & C_{44}(T) & C_{45}(T) & C_{46}(T) \\ & & & & C_{55}(T) & C_{56}(T) \\ & & & & & C_{66}(T) \end{bmatrix} \begin{bmatrix} A_x \\ A_y \\ A_z \\ R_x \\ R_y \\ R_z \end{bmatrix}$$

The **CBMD** command, one of several [composite beam section commands](#), specifies the section mass matrix (submatrix **[C]** data) for a composite beam section. The section data defined is associated with the section most recently defined (**SECTYPE**) at the specified temperature (**CBTMP**).

Unspecified values default to zero.

Related commands are **CBTMP**, **CBTE**, and **CBMX**.

For complete information, see [Using Preintegrated Composite Beam Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Beam>Composite Sections**

**CBMX**, *ROW*,  $S_{(R)(R)}$ ,  $S_{(R)(R+1)}$ ,  $S_{(R)(R+2)}$ ,  $S_{(R)(R+3)}$ ,  $S_{(R)(R+4)}$ ,  $S_{(R)(R+5)}$ ,  $S_{(R)(R+6)}$

**Specifies preintegrated cross-section stiffness for composite beam sections.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**ROW**

Row number of the matrix.

$S_{(R)(R)}$ , ...,  $S_{(R)(R+6)}$

Upper triangle of the cross-section stiffness matrix **[S]**.

## Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \\ B_m \end{bmatrix} = \begin{bmatrix} S_{11}(T) & S_{12}(T) & S_{13}(T) & S_{14}(T) & S_{15}(T) & S_{16}(T) & S_{17}(T) \\ & S_{22}(T) & S_{23}(T) & S_{24}(T) & S_{25}(T) & S_{26}(T) & S_{27}(T) \\ & & S_{33}(T) & S_{34}(T) & S_{35}(T) & S_{36}(T) & S_{37}(T) \\ & & & S_{44}(T) & S_{45}(T) & S_{46}(T) & S_{47}(T) \\ & & & & S_{55}(T) & S_{56}(T) & S_{57}(T) \\ & & & & & S_{66}(T) & S_{67}(T) \\ & & & & & & S_{77}(T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \\ B_k \end{bmatrix}$$

## Notes

The **CBMX** command, one of several [composite beam section commands](#), specifies the cross-section stiffness matrix (submatrix **[S]** data) for a composite beam section. The section data defined is associated with the section most recently defined (**SECTYPE**) at the specified temperature (**CBTMP**).

Unspecified values default to zero.

Related commands are **CBTMP**, **CBTE**, and **CBMD**.

For complete information, see [Using Preintegrated Composite Beam Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Beam>Composite Sections**

### CBTE, ALPHA

**Specifies a thermal expansion coefficient for a composite beam section.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### ALPHA

Coefficient of thermal expansion for the cross section.

## Notes

The **CBTE** command, one of several [composite beam section commands](#), specifies a thermal expansion coefficient for a beam section. The value specified is associated with the section most recently defined (**SECTYPE**) at the specified temperature (**CBTMP**).

Unspecified values default to zero.

Related commands are **CBTMP**, **CBMX**, and **CBMD**.

For complete information, see [Using Preintegrated Composite Beam Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Beam>Composite Sections**

---

### **CBTMP**, *TEMP*

**Specifies a temperature for composite-beam input.**

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM <> DY PP <> EME MFS

#### *TEMP*

Temperature value.

## Notes

The **CBTMP** command, one of several [composite beam-section commands](#), specifies a temperature to be associated with the data input via subsequent **CBMX** (preintegrated cross-section stiffness), **CBMD** (preintegrated section mass), or **CBTE** (thermal-expansion) commands.

The specified temperature remains active until the next **CBTMP** command is issued.

An unspecified temperature value defaults to zero.

For complete information, see [Using Preintegrated Composite Beam Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Beam>Composite Sections**

---

### **CDOPT**, *Option*

**Specifies format to be used for archiving geometry.**

[PREP7: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Option*

##### **IGES**

Write solid model geometry information using IGES format (default).

##### **ANF**

Write solid model geometry information using ANSYS Neutral File format.

##### **STAT**

Print out the current format setting.

## Notes

This command controls your solid model geometry format for **CDWRITE** operations. The ANF option affects only the COMB and SOLID options of the **CDWRITE** command. All other options remain unaffected.

This option setting is saved in the database.

## Menu Paths

**Main Menu>Preprocessor>Archive Model>Read**

**Main Menu>Preprocessor>Archive Model>Write**

**CDREAD**, *Option*, *Fname*, *Ext*, *--*, *Fnamei*, *Exti*

**Reads a file of solid model and database information into the database.**

[PREP7: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Option*

Selects which data to read:

#### **ALL**

Read all geometry, material property, load, and component data (default). Solid model geometry and loads will be read from the file *Fnamei.Exti*. All other data will be read from the file *Fname.Ext*.

#### **DB**

Read all database information contained in file *Fname.Ext*. This file should contain all information mentioned above except the solid model loads. If reading a .CDB file written with the GEOM option of the **CDWRITE** command, element types **[ET]** compatible with the connectivity of the elements on the file must be defined prior to reading.

#### **SOLID**

Read the solid model geometry and solid model loads from the file *Fnamei.Exti*. This file could have been written by the **CDWRITE** or **IGESOUT** command.

#### **COMB**

Read the combined solid model and database information from the file *Fname.Ext*.

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to CDB if *Fname* is blank.

### *--*

Unused field.

***Fnamei***

Name of the IGES file and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Fname*. Used only if *Option* = ALL or SOLID.

***Exti***

Filename extension (8 character maximum).

Defaults to IGES if *Fnamei* is blank.

## Notes

This command causes coded files of solid model (in IGES format) and database (in command format) information to be read. These files are normally written by the **CDWRITE** or **IGESOUT** command. Note that the active coordinate system in these files has been reset to Cartesian (**CSYS,0**).

If a set of data exists prior to the **CDREAD** operation, that data set is offset upward to allow the new data to fit without overlap. The **NOOFFSET** command allows this offset to be ignored on a set-by-set basis, causing the existing data set to be overwritten with the new data set.

When you write the geometry data using the **CDWRITE,GEOM** option, you use the **CDREAD,DB** option to read the geometry information.

Using the **CDREAD,COMB** option will not write **NUMOFF** commands to offset entity ID numbers if there is no solid model in the database.

Multiple CDB file imports cannot have elements with real constants in one file and section definitions in another. The section attributes will override the real constant attributes. If you use **CDREAD** to import multiple CDB files, define all of the elements using only real constants, or using only section definitions. Combining real constants and section definitions is not recommended.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Archive Model>Read**

---

**CDWRITE**, *Option*, *Fname*, *Ext*, --, *Fnamei*, *Exti*, *Fmat*

**Writes geometry and load database items to a file.**

[PREP7: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS  
[Product Restrictions](#)

***Option***

Selects which data to write:

**ALL**

Write all appropriate geometry, material property, load, and component data (default). Two files will be produced. *Fname* .*Ext* will contain all data items mentioned in "Notes" (p. 243), except the solid

model data. *Fname.i.Ext.i* will contain the solid model geometry and solid model loads data in the form of IGES commands. This option is not valid when **CDOPT**, ANF is active.

**COMB**

Write all data mentioned, but to a single file, *Fname.Ext*. Solid model geometry data will be written in either IGES or ANF format as specified in the **CDOPT** command, followed by the remainder of the data in the form of ANSYS commands. More information on these (IGES/ANF) file formats is provided below.

**DB**

Write all database information except the solid model and solid model loads to *Fname.Ext* in the form of ANSYS commands. This option is not valid when **CDOPT**, ANF is active.

**SOLID**

Write only the solid model geometry and solid model load data. This output will be in IGES or ANF format, as specified in the **CDOPT** command. More information on these (IGES/ANF) file formats is provided below.

**GEOM**

Write only element and nodal geometry data. Neither solid model geometry nor element attribute data will be written. One file, *Fname.Ext*, will be produced. Use **CDREAD,DB** to read in a file written with this option. Element types **[ET]** compatible with the connectivity of the elements on the file must first be defined before reading the file in with **CDREAD,DB**.

**CM**

Write only node and element component and geometry data to *Fname.Ext*.

**MAT**

Write only material property data (both linear and nonlinear) to *Fname.Ext*.

**LOAD**

Write only loads for current load step to *Fname.Ext*.

**SECT**

Write only section data to *Fname.Ext*. Pretension sections are not included.

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

***Ext***

Filename extension (8 character maximum).

The extension defaults to CDB if *Fname* is blank.

--

Unused field.

***Fname.i***

Name of the IGES file and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Fname*. Used only if *Option* = ALL or SOLID. Previous data on this file, if any, is overwritten.

***Ext.i***

Filename extension (8 character maximum).

The extension defaults to IGES in all cases, except when **CDOPT**, ANF is active and your **CDWRITE**, *Option* = SOLID. In this case *Exti* = ANF.

### ***Fmat***

Format of the output file (defaults to BLOCKED).

#### **BLOCKED**

Blocked format. This format allows faster reading of the output file. The time savings is most significant when BLOCKED is used to read .cdb files associated with very large models.

#### **UNBLOCKED**

Unblocked format.

## **Command Default**

When **SOLCONTROL**, ON, does not write default values for most of the relevant solution control commands or options. When **SOLCONTROL**, OFF, writes all default values for solution control commands.

Default output format is determined by your **CDOPT** settings.

## **Notes**

- IGES option to write solid model information:

**CDOPT**, IGS (default)

Before you write your solid model entities, you must select all corresponding lower level entities (use **ALLSEL**, BELOW, ALL).

Section properties assigned to areas, lines and other solid model entities will not be maintained when the model is exported from ANSYS using **CDWRITE** with the IGES option.

If you issue **CDWRITE** after generating a beam mesh with orientation nodes, the database file will contain all of the nodes for every beam element, including the orientation nodes. However, the orientation keypoints that were specified for the line (**LATT**) are no longer associated with the line and won't be written out to the geometry file. All associativity between the line and the orientation keypoints is lost. The **CDWRITE** command with IGES format does not support (for beam meshing) any line operation that relies on solid model associativity. For example, meshing the areas adjacent to the meshed line, plotting the line that contains the orientation nodes, or clearing the mesh from the line that contains orientation nodes may not work as expected. Concatenated lines are not written. In IGES option, the line segments that make up the concatenated lines are written. However, if you encounter an area that contains a concatenated line, the write operation halts (that area can't be recreated during the read operation). If your model has areas that contain concatenated lines, you must first list these and then unconcatenate them before issuing the **CDWRITE** command. Similarly, hardpoint information cannot be written.

- ANF option to write solid model information:

**CDOPT**, ANF

All model information in the database (regardless of select status) is written to the archive file. However, when you restore the database using this archived file, the select status of entities will also be restored. This option restores all line attributes, including orientation keypoints. It also writes out any components (not assemblies) comprised of solid model entities. The ANF option halts **CDWRITE** when a concatenated

line or an area that contains a concatenated line is detected. You must delete the concatenated lines before issuing **CDWRITE**. Similarly, hardpoint information cannot be written.

Load data includes the current load step only. Loads applied to the solid model (if any) are automatically transferred to the finite element model when this command is issued. **CDWRITE** writes out solid model loads for meshed models only. If the model is not meshed, ANSYS cannot save the solid model loads. Component data include component definitions, but not assembly definitions. Appropriate **NUMOFF** commands are included at the beginning of the file; this is to avoid overlap of an existing database when the file is read in.

Element order information (resulting from a **WAVES** command) is not written. The data in the database remain untouched.

**CDWRITE** does not support the **GSBDATA** and **GSGDATA** commands, and these commands will not be written to the file. **CDWRITE** does not support the [FiberSIM-ANSYS interface](#).

The data may be reread (on a different machine, for example) with the **CDREAD** command. Caution: When the file is read in, the **NUMOFF,MAT** command may cause a mismatch between material definitions and material numbers referenced by certain loads and element real constants. See **NUMOFF** for details. Also, be aware that the files created by the **CDWRITE** command explicitly set the active coordinate system to Cartesian (**CSYS,0**).

You should generally use the blocked format ( $Format = \text{BLOCKED}$ ) when writing out model data with **CDWRITE**. This is a compressed data format that greatly reduces the time required to read large models through the **CDREAD** command. The blocked and unblocked formats are described in Chapter 3 of the [Guide to Interfacing with ANSYS](#).

If you use **CDWRITE** in any of the derived products (ANSYS Emag, ANSYS Professional), then before reading the file, you must edit the `Jobname.cdb` file to remove commands that are not available in the respective component product.

The **CDWRITE** command writes the PART information for any ANSYS LS-DYNA input file to the `Jobname.cdb` file using the **EDPREAD** command. (**EDPREAD** is not an ANSYS documented command, it is written only when the **CDWRITE** command is issued.) The PART information can be automatically read into ANSYS with the **CDREAD** command. However, if more than one `Jobname.cdb` file is read, the PART list from the last `Jobname.cdb` file overwrites the existing PART list of the total model. This will affect all PART-related commands contained in the `Jobname.cdb` file. That means the user can join models, but not PART-related inputs, which the user must modify using the newly-created PART numbers. In limited cases, an update of the PART list (**EDWRITE,PUPDATE**) is possible. This requires that no used combination of MAT/TYP/REAL appears more than once in the list. However, partial changes to the PART-related commands may be necessary.

---

### Caution

The **CDWRITE** command does not support (for beam meshing) any line operation that relies on solid model associativity. For example, meshing the areas adjacent to the meshed line, plotting the line that contains the orientation nodes, or clearing the mesh from the line that contains orientation nodes may not work as expected. For more information about beam meshing, see [Meshing Your Solid Model](#) in the [Modeling and Meshing Guide](#).

This command is also valid in SOLUTION.



## Product Restrictions

In ANSYS ED, *Option* = ALL, COMB, or SOLID is not allowed, and the default is *Option* = DB.

## Menu Paths

Main Menu>Preprocessor>Archive Model>Write

**CE**, *NEQN*, *CONST*, *NODE1*, *Lab1*, *C1*, *NODE2*, *Lab2*, *C2*, *NODE3*, *Lab3*, *C3*

**Defines a constraint equation relating degrees of freedom.**

PREP7: Constraint Equations

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### **NEQN**

Set equation reference number:

**n**

Arbitrary set number.

### **HIGH**

The highest defined constraint equation number. This option is especially useful when adding nodes to an existing set.

### **NEXT**

The highest defined constraint equation number plus one. This option automatically numbers coupled sets so that existing sets are not modified.

The default value is HIGH.

### **CONST**

Constant term of equation.

### **NODE1**

Node for first term of equation. If *-NODE1*, this term is deleted from the equation.

### **Lab1**

Degree of freedom label for first term of equation. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations, in radians). Thermal labels: TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature). Electric labels: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials).

### **C1**

Coefficient for first node term of equation. If zero, this term is ignored.

### **NODE2, Lab2, C2**

Node, label, and coefficient for second term.

### **NODE3, Lab3, C3**

Node, label, and coefficient for third term.

## Notes

Repeat the **CE** command to add additional terms to the same equation. To change only the constant term, repeat the command with no node terms specified. Only the constant term can be changed during solution, and only with the **CECMOD** command.

Linear constraint equations may be used to relate the degrees of freedom of selected nodes in a more general manner than described for nodal coupling [CP]. The constraint equation is of the form:

$$\text{Constant} = \sum_{l=1}^N (\text{Coefficient}(l) * U(l))$$

where U(l) is the degree of freedom (displacement, temperature, etc.) of term (l). The following example is a set of two constraint equations, each containing three terms:

$$0.0 = 3.0 * (1 \text{ UX}) + 3.0 * (4 \text{ UX}) + (-2.0) * (4 \text{ ROTY})$$

$$2.0 = 6.0 * (2 \text{ UX}) + 10.0 * (4 \text{ UY}) + 1.0 * (3 \text{ UZ})$$

The first unique degree of freedom in the equation is eliminated in terms of all other degrees of freedom in the equation. A unique degree of freedom is one which is not specified in any other constraint equation, coupled node set, specified displacement set, or master degree of freedom set. It is recommended that the first term of the equation be the degree of freedom to be eliminated. The first term of the equation cannot contain a master degree of freedom, and no term can contain coupled degrees of freedom. The same degree of freedom may be specified in more than one equation but care must be taken to avoid over-specification (over-constraint).

The degrees of freedom specified in the equation (i.e., UX, UY, ROTZ, etc.) must also be included in the model (as determined from the element types [ET]). Also, each node in the equation must be defined on an element (any element type containing that degree of freedom will do).

For buckling and modal analyses, the constant term of the equation will not be taken into account (that is, *CONST* is always zero).

## Menu Paths

**Main Menu>Preprocessor>Coupling / Ceqn>Constraint Eqn**

**CECHECK**, *ItemLab*, *Tolerance*, *DOF*

**Check constraint equations and couplings for rigid body motions.**

[PREP7: Database](#)

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *ItemLab*

Item indicating what is to be checked:

#### **CE**

Check constraint equations only

#### **CP**

Check couplings only

#### **ALL**

Check both CE and CP

**Tolerance**

Allowed amount of out-of-balance for any constraint equation or coupled set. The default value of 1.0e-6 is usually good.

**DOF**

Specifies which DOF is to be checked. Default is RIGID, the usual option. Other choices are individual DOF such as UX, ROTZ, etc. or THERM. The THERM option will check the constraint equations or coupled sets for free thermal expansions, whereas the individual DOFs check under rigid body motions. ALL is RIGID and THERM.

**Notes**

This command imposes a rigid body motion on the nodes attached to the constraint equation or coupled set and makes sure that no internal forces are generated for such rigid body motions. Generation of internal forces by rigid body motions usually indicates an error in the equation specification (possibly due to nodal coordinate rotations). The THERM option does a similar check to see that no internal forces are created by the equations if the body does a free thermal expansion (this check assumes a single isotropic coefficient of expansion).

**Menu Paths**

**This command cannot be accessed from a menu.**

**CECMOD**, *NEQN*, *CONST*

**Modifies the constant term of a constraint equation during solution.**

[SOLUTION: Load Step Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**NEQN**

Reference number of constraint equation.

**CONST**

New value of the constant term of equation.

**Notes**

Other terms of the constraint equation cannot be changed during the solution phase, but must be defined or changed within PREP7 prior to the solution. See the **CE** command for details.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Coupling / Ceqn>Modify ConstrEqn**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Modify ConstrEqn**

**Main Menu>Solution>Load Step Opts>Other>Modify ConstrEqn**

**CECYC**, *Lowname*, *Highname*, *Nsector*, *HIndex*, *Tolerance*, *Kmove*, *Kpairs*

**Generates the constraint equations for a cyclic symmetry analysis**

[PREP7: Constraint Equations](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***Lowname***

Name of a component for the nodes on the low angle edge of the sector. Enclosed in single quotes.

***Highname***

Name of a component for the nodes on the high angle edge of the sector. Enclosed in single quotes.

***Nsector***

Number of sectors in the complete 360 degrees.

***HIndex***

Harmonic index to be represented by this set of constraint equations. If *Hindex* is -1, generate constraint equations for static cyclic symmetry. If *HIndex* is -2, generate constraint equations for static cyclic asymmetry.

***Tolerance***

A positive tolerance is an absolute tolerance (length units), and a negative tolerance is a tolerance relative to the local element size.

***Kmove***

**0**  
Nodes are not moved.

**1**  
HIGHNAME component nodes are moved to match LOWNAME component nodes exactly.

***Kpairs***

**0**  
Do not print paired nodes

**1**  
Print table of paired nodes

## Notes

The analysis can be either modal cyclic symmetry or static cyclic symmetry.

The pair of nodes for which constraint equations are written are rotated into **CSYS,1**.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**CEDELE**, *NEQN1*, *NEQN2*, *NINC*, *Nsel***Deletes constraint equations.**[PREP7: Constraint Equations](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; EM EH DY PP &lt;&gt; EME MFS

***NEQN1*, *NEQN2*, *NINC***

Delete constraint equations from *NEQN1* to *NEQN2* (defaults to *NEQN1*) in steps of *NINC* (defaults to 1). If *NEQN1* = ALL, *NEQN2* and *NINC* will be ignored all constraint equations will be deleted.

***Nsel***

Additional node selection control:

**ANY**Delete equation set if *any* of the selected nodes are in the set (default).**ALL**Delete equation set only if *all* of the selected nodes are in the set.

## Menu Paths

**Main Menu>Preprocessor>Coupling / Ceqn>Del Constr Eqn****CEINTF**, *TOLER*, *DOF1*, *DOF2*, *DOF3*, *DOF4*, *DOF5*, *DOF6*, *MoveTol***Generates constraint equations at an interface.**[PREP7: Constraint Equations](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; EM EH DY PP &lt;&gt; EME MFS

***TOLER***

Tolerance about selected elements, based on a fraction of the element dimension (defaults to 0.25 (25%)). Nodes outside the element by more than the tolerance are not accepted as being on the interface.

***DOF1*, *DOF2*, *DOF3*, . . . , *DOF6***

Degrees of freedom for which constraint equations are written. Defaults to all applicable DOFs. *DOF1* accepts ALL as a valid label, in which case the rest are ignored (all DOFs are applied).

***MoveTol***

The allowed "motion" of a node (see Note below). This distance is in terms of the element coordinates (-1.0 to 1.0). A typical value is 0.05. Defaults to 0 (do not move). *MoveTol* must be less than or equal to *TOLER*.

## Notes

This command can be used to "tie" together two regions with dissimilar mesh patterns by generating constraint equations that connect the selected nodes of one region to the selected elements of the other region. At the interface between regions, nodes should be selected from the more dense mesh region, A, and the elements selected from the less dense mesh region, B. The degrees of freedom of region A nodes are interpolated with the corresponding degrees of freedom of the nodes on the region B elements, using the shape functions of the region B elements. Constraint equations are then written that relate region A and B nodes at the interface.

The *MoveTol* field lets the nodes in the previously mentioned region A change coordinates when slightly inside or outside the elements of region B. The change in coordinates causes the nodes of region A to assume the same surface as the nodes associated with the elements of region B. The constraint equations that relate the nodes at both regions of the interface are then written.

Solid elements with six degrees of freedom should only be interfaced with other six degree-of-freedom elements. The region A nodes should be near the region B elements. A location tolerance based on the smallest region B element length may be input. Stresses across the interface are not necessarily continuous. Nodes in the interface region should not have specified constraints.

Use the **CPINTF** command to connect nodes by coupling instead of constraint equations. Use the **EINTF** command to connect nodes by line elements. See also the **NSEL** and **ESEL** commands for selecting nodes and elements. See the *Theory Reference for the Mechanical APDL and Mechanical Applications* for a description of 3-D space used to determine if a node will be considered by this command.

As an alternative to the **CEINTF** command, you can use contact elements and the internal multipoint constraint (MPC) algorithm to tie together two regions having dissimilar meshes. See *Solid-Solid and Shell-Shell Assemblies* for more information.

## Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Adjacent Regions

---

**CELIST**, *NEQN1*, *NEQN2*, *NINC*, *Option*

**Lists the constraint equations.**

[PREP7: Constraint Equations](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***NEQN1*, *NEQN2*, *NINC***

List constraint equations from *NEQN1* to *NEQN2* (defaults to *NEQN1*) in steps of *NINC* (defaults to 1). If *NEQN1* = ALL (default), *NEQN2* and *NINC* are ignored and all constraint equations are listed.

***Option***

Options for listing constraint equations:

**ANY**

List equation set if *any* of the selected nodes are in the set (default). Only externally-generated constraint equations are listed.

**ALL**

List equation set only if *all* of the selected nodes are in the set. Only externally-generated constraint equations are listed.

**INTE**

List internally-generated constraint equations that are associated with MPC-based contact. Constraint equations are listed only if *all* the nodes in the set are selected.

**CONV**

Convert internal constraint equations to external constraint equations. Internal constraint equations are converted only if *all* of the nodes in the set are selected.

## Notes

This command is valid in any processor. However, the INTE and CONV options are only valid in the Solution processor after a **SOLVE** command has been issued.

## Menu Paths

**Utility Menu>List>Other>Constraint Eqns>All CE nodes selected**  
**Utility Menu>List>Other>Constraint Eqns>Any CE node selected**

## CENTER, NODE, NODE1, NODE2, NODE3, RADIUS

**Defines a node at the center of curvature of 2 or 3 nodes.**

[PREP7: Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NODE

Number to be assigned to the node generated at the center of curvature.

### NODE1, NODE2, NODE3

Three nodes used to calculate the center of curvature, as described under *RADIUS*.

### RADIUS

Used to control the interpretation of *NODE1*, *NODE2* and *NODE3*:

**0**

*NODE1*, *NODE2* and *NODE3* lie on a circular arc. The program will calculate the center of curvature (and radius) (default).

**≠0**

*NODE1* and *NODE2* are the endpoints of an arc, and *RADIUS* is the radius of curvature. The program will locate the center of curvature on the *NODE3* side of the *NODE1-NODE2* line if *RADIUS* > 0, and opposite to *NODE3* if *RADIUS* < 0.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Nodes>At Curvature Ctr**

## CEQN

**Specifies "Constraint equations" as the subsequent status topic.**

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Preprocessor>Constraint Eqns

**CERIG**, *MASTE*, *SLAVE*, *Ldof*, *Ldof2*, *Ldof3*, *Ldof4*, *Ldof5*

**Defines a rigid region.**

[PREP7: Constraint Equations](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### **MASTE**

Retained (or master) node for this rigid region. If *MASTE* = P, then graphical picking of the master and slave nodes is enabled (first node picked will be the master node, and subsequent nodes picked will be slave nodes), and subsequent fields are ignored (valid only in GUI).

### **SLAVE**

Removed (or slave) node for this rigid region. If ALL, slave nodes are all selected nodes.

### **Ldof**

Degrees of freedom associated with equations:

#### **ALL**

All applicable degrees of freedom (default). If 3-D, generate 6 equations based on UX, UY, UZ, ROTX, ROTY, ROTZ; if 2-D, generate 3 equations based on UX, UY, ROTZ.

#### **UXYZ**

Translational degrees of freedom. If 3-D, generate 3 equations based on the slave nodes' UX, UY, and UZ DOFs and the master node's UX, UY, UZ, ROTX, ROTY, and ROTZ DOFs; if 2-D, generate 2 equations based on the slave nodes UX and UY DOFs and the master nodes UX, UY, and ROTZ DOFs. No equations are generated for the rotational coupling.

#### **RXYZ**

Rotational degrees of freedom. If 3-D, generate 3 equations based on ROTX, ROTY, ROTZ; if 2-D, generate 1 equation based on ROTZ. No equations are generated for the translational coupling.

#### **UX**

Slave translational UX degree of freedom only.

#### **UY**

Slave translational UY degree of freedom only.

#### **UZ**

Slave translational UZ degree of freedom only.

#### **ROTX**

Slave rotational ROTX degree of freedom only.

#### **ROTY**

Slave rotational ROTY degree of freedom only.

#### **ROTZ**

Slave rotational ROTZ degree of freedom only.



***Ldof2, Ldof3, Ldof4, Ldof5***

Additional degrees of freedom. Used only if more than one degree of freedom required and *Ldof* is not ALL, UXYZ, or RXYZ.

**Notes**

Defines a rigid region (link, area or volume) by automatically generating constraint equations to relate nodes in the region. Nodes in the rigid region must be assigned a geometric location before this command is used. Also, nodes must be connected to elements having the required degree of freedom set (see *Ldof* above). Generated constraint equations are based on small deflection theory. Generated constraint equations are numbered beginning from the highest previously defined equation number (*NEQN*) plus 1. Equations, once generated, may be listed [**CELIST**] or modified [**CE**] as desired. Repeat **CERIG** command for additional rigid region equations.

This command will generate the constraint equations needed for defining rigid lines in 2-D or 3-D space. Multiple rigid lines relative to a common point are used to define a rigid area or a rigid volume. In 2-D space, with *Ldof* = ALL, three equations are generated for each pair of constrained nodes. These equations define the three rigid body motions in global Cartesian space, i.e., two in-plane translations and one in-plane rotation. These equations assume the X-Y plane to be the active plane with UX, UY, and ROTZ degrees of freedom available at each node. Other types of equations can be generated with the appropriate *Ldof* labels.

Six equations are generated for each pair of constrained nodes in 3-D space (with *Ldof* = ALL). These equations define the six rigid body motions in global Cartesian space. These equations assume that UX, UY, UZ, ROTX, ROTY, and ROTZ degrees of freedom are available at each node.

The UXYZ label allows generating a partial set of rigid region equations. This option is useful for transmitting the bending moment between elements having different degrees of freedom at a node. With this option only two of the three equations are generated for each pair of constrained nodes in 2-D space. In 3-D space, only three of the six equations are generated. In each case the rotational coupling equations are not generated. Similarly, the RXYZ label allows generating a partial set of equations with the translational coupling equations omitted.

Applying this command to a large number of slave nodes may result in constraint equations with a large number of coefficients. This may significantly increase the peak memory required during the process of element assembly. If real memory or virtual memory is not available, consider reducing the number of slave nodes.

As an alternative to the **CERIG** command, you can define a similar type of rigid region using contact elements and the internal multipoint constraint (MPC) algorithm. See [Surface-Based Constraints](#) for more information.

**CERIG** cannot be deleted using **CEDELE,ALL** and then regenerated in the second or higher load steps if the **LSWRITE** and **LSSOLVE** procedure is used. **CERIG** writes constraint equations directly into load step files. Deleting constraint equations (**CEDELE,ALL**) cannot always maintain the consistency among load steps.

**Menu Paths**

**Main Menu>Preprocessor>Coupling / Ceqn>Rigid Region**

**CESGEN**, *ITIME*, *INC*, *NSET1*, *NSET2*, *NINC*

**Generates a set of constraint equations from existing sets.**

[PREP7: Constraint Equations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***ITIME*, *INC***

Do this generation operation a total of *ITIMES*, incrementing all nodes in the existing sets by *INC* each time after the first. *ITIME* must be >1 for generation to occur.

***NSET1*, *NSET2*, *NINC***

Generate sets from sets beginning with *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* is negative, *NSET2* and *NINC* are ignored and the last  $|NSET1|$  sets (in sequence from maximum set number) are used as the sets to be repeated.

**Notes**

Generates additional sets of constraint equations (with same labels) from existing sets. Node numbers between sets may be uniformly incremented.

**Menu Paths**

Main Menu>Preprocessor>Coupling / Ceqn>Gen w/Same DOF

**CFACT**, *RFACTA*, *IFACTA*, *RFACTB*, *IFACTB*, *RFACTC*, *IFACTC*

**Defines complex scaling factors to be used with operations.**

[POST26: Controls](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***RFACTA***

Real portion of the complex scale factor used in place of *FACTA*.

***IFACTA***

Imaginary portion of the complex scale factor used in place of *FACTA*.

***RFACTB***

Real portion of the complex scale factor used in place of *FACTB*.

***IFACTB***

Imaginary portion of the complex scale factor used in place of *FACTB*.

***RFACTC***

Real portion of the complex scale factor used in place of *FACTC*.

***IFACTC***

Imaginary portion of the complex scale factor used in place of *FACTC*.

**Command Default**

Use the real factors as described with the operation command.

## Notes

Defines complex scale factors to be used with the operations [**ADD**, **PROD**, etc.]. If this command is supplied, these complex factors override any real factors (*FACTA*, *FACTB*, *FACTC*) supplied on the operation commands. Factors are typically involved in scaling a specified variable, such as in the term *FACTA* x *IA* of the **ADD** command to scale variable *IA* before the **ADD** operation.

When the **CFACT** command is active, defaults are as follows: 1) if the complex factor is not specified, but the variable upon which it acts (such as *IA*) is specified, the factor defaults to 1.0+i0.0; 2) if the variable upon which the factor operates is not specified, but the factor is specified, the variable defaults to 1.0 so that the term in the operation becomes the complex factor itself; 3) if neither the factor nor the variable number is supplied, the term is omitted from the operation. Once the operation (such as the **ADD** command) has been processed, the **CFACT** command becomes inactive and must be specified again if it is to be used.

## Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Complex ScaleFact

---

### \*CFCLOS

Closes the "command" file.

[APDL: Macro Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This command is valid in any processor.

## Menu Paths

This command cannot be accessed from a menu.

---

### \*CFOPEN, *Fname*, *Ext*, *--*, *Loc*

Opens a "command" file.

[APDL: Macro Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to *Jobname*.

**Ext**

Filename extension (8 character maximum).

The extension defaults to CMD if *Fname* is blank.

--

Unused field.

**Loc**

Determines whether existing file will be overwritten or appended:

**(blank) --**

The existing file will be overwritten.

**APPEND --**

The file will be appended to the existing file.

## Notes

Data processed with the **\*VWRITE** command will also be written to this file if the file is open when the **\*VWRITE** command is issued.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

### **\*CFWRITE**, *Command*

**Writes an ANSYS command (or similar string) to a "command" file.**

APDL: Macro Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

**Command**

Command or string to be written. The standard command form of a label followed by arguments separated by commas is assumed. *Command* may be a parameter assignment (e.g., **\*CFWRITE**, A = 5).

## Notes

Writes an ANSYS command (or similar string) to the file opened with **\*CFOPEN**. The *Command* string is not executed (except that numeric and character parameter substitution and operations (with imbedded \*, /, >, etc. characters) are performed before writing). When used with **\*GET** results and parameter substitution, an ANSYS command can be created from results and then read back into the ANSYS program (or used elsewhere). For example, if the command **\*CFWRITE,BF,NNUM,TEMP,TVAL** is used in a do-loop, where TVAL is a parameter value returned from the **\*GET** operation and NNUM is a specified or returned parameter value, a series of **BF** commands, with numerical values substituted for the two parameters, will be written. To create a file without parameter substitution, use **\*CREATE**.

This command is valid in any processor.

## Menu Paths

This command cannot be accessed from a menu.

**/CFORMAT**, *NFIRST*, *NLAST*

**Controls the graphical display of alphanumeric character strings for parameters, components, assemblies, and tables.**

GRAPHICS:Labeling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NFIRST***

Display the first *n* characters of the parameter, component, assembly, or table name, up to 32. Defaults to 32.

### ***NLAST***

Display the last *n* characters of the parameter, component, assembly, or table name, up to 32. Defaults to 0.

## Notes

Use this command to control the length of the character string that is shown in the graphics window for a parameter, component, assembly, or table name.

The total number of characters (*NFIRST* + *NLAST* +3) cannot exceed 32.

If *NFIRST* is greater than zero and *NLAST* = 0, only the *NFIRST* characters are displayed, followed by an ellipsis.

If *NFIRST* = 0 and *NLAST* is greater than zero, only the *NLAST* characters are displayed, preceded by an ellipsis (...).

If both *NFIRST* and *NLAST* are greater than zero, the name will be shown as *NFIRST*, followed by an ellipsis (...), followed by *NLAST*, up to a maximum of 32 characters.

For example, if *NFIRST* = 6 and *NLAST* = 3, and the character string is LENGTHOFSIDEONE, then it will appear in the graphics window as LENGTH...ONE.

If the actual length of the character string is less than the specified combination of *NFIRST* + *NLAST* +3, then the actual string will be used.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

**CGLOC**, *XLOC*, *YLOC*, *ZLOC*

**Specifies the origin location of the acceleration coordinate system.**

**SOLUTION:** Inertia

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

*XLOC*, *YLOC*, *ZLOC*

Global Cartesian X, Y, and Z coordinates of the acceleration coordinate system origin.

**Notes**

Specifies the origin location of the acceleration coordinate system with respect to the global Cartesian system. The axes of this acceleration coordinate system are parallel to the global Cartesian axes.

A structure may be rotating about the global Cartesian origin [**OMEGA**, **DOMEGA**], which may in turn be rotating about another point (the origin of the acceleration coordinate system), introducing Coriolis effects. The location of this point (relative to the global Cartesian origin) is specified with this **CGLOC** command. For example, if Y is vertical and the global system origin is at the surface of the earth while the acceleration system origin is at the center of the earth, *YLOC* should be -4000 miles (or equivalent) if the rotational effects of the earth are to be included. The rotational velocity of the global Cartesian system about this point is specified with the **CGOMGA** command, and the rotational acceleration is specified with the **DCGOMG** command.

The rotational velocities and accelerations are mainly intended to include mass effects in a static (**ANTYPE**, **STATIC**) analysis. If used in dynamic analyses, no coupling exists between the user input terms and the time history response of the structure. See [Acceleration Effect](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details. Related commands are **ACEL**, **CGOMGA**, **DCGOMG**, **DOMEGA**, and **OMEGA**.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Rotating Coords**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Coriolis Effects**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Coriolis Effects**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Acceleration CS>Delete Accel CS**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Acceleration CS>Set Accel CS**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Coriolis Effects**  
**Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Coriolis Effects**  
**Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Rotating Coords**  
**Main Menu>Solution>Loading Options>Acceleration CS>Delete Accel CS**  
**Main Menu>Solution>Loading Options>Acceleration CS>Set Accel CS**

**CGOMGA**, *CGOMX*, *CGOMY*, *CGOMZ*

Specifies the rotational velocity of the global origin.

SOLUTION: Inertia

MP ME ST PR PRN DS DSS FL <> <> DY PP <> EME MFS

*CGOMX*, *CGOMY*, *CGOMZ*

Rotational velocity of the global origin about the acceleration system X, Y, and Z axes.

**Notes**

Specifies the rotational velocity of the global origin about each of the acceleration coordinate system axes. The location of the acceleration coordinate system is defined with the **CGLOC** command. Rotational velocities may be defined in analysis types **ANTYPE**, **STATIC**, **HARMIC** (full or mode superposition), **TRANS** (full or mode superposition), and **SUBSTR**. See *Acceleration Effect* in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details. Units are radians/time. Related commands are **ACEL**, **CGLOC**, **DCGOMG**, **DOMEGA**, and **OMEGA**.

The **CGOMGA** command supports tabular boundary conditions (%*TABNAME\_X*%, %*TABNAME\_Y*%, and %*TABNAME\_Z*%) for *CGOMGA\_X*, *CGOMGA\_Y*, and *CGOMGA\_Z* input values (**\*DIM**) for full transient and harmonic analyses.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Rotating Coords**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Coriolis Effects**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Coriolis Effects**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Coriolis Effects**  
**Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Coriolis Effects**  
**Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Rotating Coords**

**CHECK**, *Sele*, *Levl*

Checks current database items for completeness.

PREP7: Database

SOLUTION: Analysis Options

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

*Sele*

Specifies which elements are to be checked:

**(blank)**

Check all data.

**ESEL**

Check only elements in the selected set and unselect any elements not producing geometry check messages. The remaining elements (those producing check messages) can then be displayed and corrected. A null set results if no elements produce a message. Issue **ESEL,ALL** to select all elements before proceeding.

**Levl**

Used only with *Sele* = ESEL:

**WARN**

Select elements producing warning and error messages.

**ERR**

Select only elements producing error messages (default).

## Notes

This command will not work if **SHPP,OFF** has been set. A similar, automatic check of all data is done before the solution begins.

If the "Check Elements" option is invoked through the GUI (menu path **Main Menu> Preprocessor> Meshing> Check Elms**), the **CHECK,ESEL** logic is used to highlight elements in the following way: good elements are blue, elements having warnings are yellow, and bad (error) elements are red.

---

**Note**

The currently selected set of elements is not changed by this GUI function.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Check Mesh>Individual Elm>Select Warning/Error Elements**

---

## CHKMSH, *Comp*

Checks area and volume entities for previous meshes.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Comp***

Name of component containing areas or volumes.

## Notes

**CHKMSH** invokes a predefined ANSYS macro that checks areas and volumes to find out if they were previously meshed. This macro name will appear in the log file (*Jobname*.LOG) prior to area and volume meshing operations initiated through the GUI. This command is not intended to be typed in directly in an ANSYS session (although it can be included in an input file for use with the **/INPUT** command).

## Menu Paths

**This command cannot be accessed from a menu.**

---



**CINT**, *Action*, *Par1*, *Par2*, *Par3*, *Par4*, *Par5*, *Par6*, *Par7*

## Defines parameters associated with fracture parameter calculations

PREP7: Data Tables  
SOLUTION: Analysis Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Action*

Specifies action for defining or manipulating initial crack data:

#### **NEW**

Initiate a new calculation and assign an ID.

#### **CTNC**

Define the crack tip node component.

#### **CENC**

Define the crack extension node component, the crack tip node, and the crack extension direction.

#### **TYPE**

Define the type of calculation to perform.

#### **DELE**

Delete the **CINT** object associated with the specified ID.

#### **NCON**

Define the number of contours to be calculated in the contour-integral calculation.

#### **SYMM**

Indicate whether the crack is on a symmetrical line or plane.

#### **NORM**

Define the crack plane normal.

## Command Specification for *Action* = **NEW**

**CINT**, **NEW**, *Par1*

*Par1* --

**CINT** ID number.

## Command Specifications for *Action* = **CTNC**

**CINT**, **CTNC**, *Par1*, *Par2*, *Par3*

*Par1* --

Crack tip node component name.

*Par2* --

Crack-extension direction calculation-assist node. Any node on the open side of the crack.

*Par3* --

Crack front's end-node crack-extension direction override flag:

**0** --

Align the extension direction with the edges attached at the two end nodes of the crack front (default).

1 --

Align the extension direction to be perpendicular to the crack front.

## Command Specifications for *Action = CENC*

***CINT, CENC, Par1, Par2, Par3, Par4, Par5, Par6, Par7***

***Par1*** --

Crack extension node component name (**CM**).

***Par2*** --

Crack tip node. The crack tip node defaults to the first node of the crack extension node component.

***Par3, Par4*** --

Coordinate system number (*Par3*) and the number of the axis that is coincident with the crack direction (*Par4*). When these parameters are defined, *Par5, Par6* and *Par7* are ignored.

***Par5, Par6, Par7*** --

Global x, y, and z components of the crack extension direction vector. (*Par3* and *Par4* must be blank.)

## Command Specifications for *Action = TYPE*

***CINT, TYPE, Par1***

***Par1*** --

Type of calculation to perform:

**JINT** --

Calculate J-integral (default).

**SIFS** --

Calculate stress-intensity factors.

**VCCT** --

Calculate energy-release rate parameters using the VCCT method.

## Command Specifications for *Action = DELE*

***CINT, DELE, Par1***

***Par1*** --

**CINT** ID (default = ALL).

## Command Specifications for *Action = NCON*

***CINT, NCON, Par1***

***Par1*** --

Number of contours to be calculated.

## Command Specifications for *Action = SYMM*

### *CINT, SYMM, Par1*

*Par1* --

**OFF, 0, or NO** --

No symmetry (default).

**ON, 1, or YES** --

Symmetric about the crack line/plane.

## Command Specifications for *Action = NORM*

### *CINT, NORM, Par1, Par2*

*Par1* --

Coordinate system number (default = 0, global Cartesian).

*Par2* --

Axis of coordinate system (default = 2, global Cartesian Y-axis).

## Notes

Initiate a new calculation via the *Action = NEW* parameter. Subsequent **CINT** commands (with parameters other than NEW) define the input required for the fracture parameter calculations.

The simplest method is to define crack information using *Action = CTNC*; however, this method limits you to only one node for a given location along the crack front.

To define crack information at multiple locations along the crack front, use *Action = CENC*. You can issue **CINT,CENC, Par1**, etc. multiple times to define the crack extension node component, the crack tip, and the crack extension directions at multiple locations along the crack front.

Although you can vary the sequence of your definitions, all of the crack tip nodes specified must be at the crack front, and no crack tip node can be omitted

You can define your crack extension direction directly using either *Action = CENC* or *Action = NORM*.

The stress-intensity factors calculation (**CINT,TYPE,SIFS**) applies only to isotropic linear elasticity. Use only one material type for the crack tip elements that are used for the calculations.

For more information about using the **CINT** command, including supported element types and material behavior, see [Numerical Evaluation of Fracture Mechanics Parameters](#) in the *Structural Analysis Guide*.

## Menu Paths

**This command cannot be accessed from a menu.**

**CIRCLE**, *PCENT*, *RAD*, *PAXIS*, *PZERO*, *ARC*, *NSEG*

**Generates circular arc lines.**

PREP7:Lines

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**PCENT**

Keypoint defining the center of the circle (in the plane of the circle). If *PCENT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**RAD**

Radius of the circle. If *RAD* is blank and *PCENT* = P, the radius is the distance from *PCENT* to *PZERO*.

**PAXIS**

Keypoint defining axis of circle (along with *PCENT*). If *PCENT* = P and *PAXIS* is omitted, the axis is normal to the working plane.

**PZERO**

Keypoint defining the plane normal to circle (along with *PCENT* and *PAXIS*) and the zero degree location. Need not be in the plane of the circle. This value is not required if *PAXIS* is defined along the Y axis (that is, a circle in the XZ plane).

**ARC**

Arc length (in degrees). Positive follows right-hand rule about *PCENT*-*PAXIS* vector. Defaults to 360°.

**NSEG**

Number of lines around circumference (defaults to minimum required for 90°-maximum arcs, i.e., 4 for 360°). Number of keypoints generated is *NSEG* for 360° or *NSEG* + 1 for less than 360°.

## Notes

Generates circular arc lines (and their corresponding keypoints). Keypoints are generated at regular angular locations (based on a maximum spacing of 90°). Arc lines are generated connecting the keypoints. Keypoint and line numbers are automatically assigned, beginning with the lowest available values [NUMSTR]. Adjacent lines use a common keypoint. Line shapes are generated as arcs, regardless of the active coordinate system. Line shapes are invariant with coordinate system after they are generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>By Cent & Radius**

**Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>Full Circle**

**CISOL**, *n*, *ID*, *node*, *Cont*, *Dtype*

**Stores fracture parameter information in a variable.**

POST26:Set Up

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**n**

Arbitrary reference number or name assigned to this variable. Number must be >1 but </= NUMVAR.

**ID**

Crack ID number.

**node**

Crack tip node number.

**Cont**

Contour number.

**Dtype**

Data type to output:

**JINT**

J-integral

**IIN1**

Interaction integral 1

**IIN2**

Interaction integral 2

**IIN3**

Interaction integral 3

**K1**

Mode 1 stress-intensity factor

**K2**

Mode 2 stress-intensity factor

**K3**

Mode 3 stress-intensity factor

**G1**

Mode 1 energy release rate

**G2**

Mode 2 energy release rate

**G3**

Mode 3 energy release rate

**GT**

Total energy release rate

## Menu Paths

**This command cannot be accessed from a menu.**

**/CLABEL**, *WN*, *KEY*

**Specifies contour labeling.**

[GRAPHICS: Labeling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WN**

Window number (or ALL) to which command applies (defaults to 1).

**KEY**

Labeling key:

**0 or 1**

Label contours with legend or color (default).

**-1**

No contour labeling.

***N***

Same as 1 except show alphabetic legend only on every *N*th element.

## Command Default

Show contour line labels.

## Notes

Labels contours for identification with alphabetic legend for vector displays and color for raster displays. Number of contours is automatically reduced to 9 (or fewer) for clarity. Use **/CONTOUR** command to increase (24 maximum for alphabetic labeling; no limit for color labeling).

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Style>Contours>Contour Labeling**

---

## **/CLEAR**, *Read*

**Clears the database.**

[DATABASE: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Read*

File read option:

**START**

Reread `start130.ans` file (default).

**NOSTART**

Do not reread `start130.ans` file.

## Notes

Resets the ANSYS database to the conditions at the beginning of the problem. Sets the import and Boolean options back to the ANSYS default. All items are deleted from the database and memory values are set to zero for items derived from database information. All files are left intact. This command is useful between multiple analyses in the same run, or between passes of a multipass analysis (such as between the substructure generation, use, and expansion passes). Should not be used in a do-loop since loop counters will be reset. The `start130.ans` file will be reread (by default) after the database is cleared, unless *Read* is set to NOSTART. Additional commands cannot be stacked (using the \$ separator) on the same line as the **/CLEAR** command.

Use care when placing the **/CLEAR** command within branching constructs (for example, those employing **\*DO** or **\*IF** commands). The command deletes all parameters including the looping parameter for do-loops. (You can preserve your iteration parameter by issuing a **PARSAV** command prior to the **/CLEAR** command, then following the **/CLEAR** command with a **PARRES** command.)

**/CLEAR** resets the jobname to match the currently open session `.LOG` and `.ERR` files. This will return the jobname to its original value, or to the most recent value specified on **/FILENAME** with `KEY = 1`.

This command is valid only at the Begin level.

## Menu Paths

Utility Menu>File>Clear & Start New

---

**CLOCAL**, *KCN*, *KCS*, *XL*, *YL*, *ZL*, *THXY*, *THYZ*, *THZX*, *PAR1*, *PAR2*

**Defines a local coordinate system relative to the active coordinate system.**

DATABASE: Coordinate System

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### **KCN**

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

### **KCS**

Coordinate system type:

#### **0 or CART**

Cartesian

#### **1 or CYLIN**

Cylindrical (circular or elliptical)

#### **2 or SPHE**

Spherical (or spheroidal)

#### **3 or TORO**

Toroidal

### **XL, YL, ZL**

Location (in the active coordinate system) of the origin of the new coordinate system (R,  $\theta$ , Z for cylindrical, R,  $\theta$ ,  $\Phi$  for spherical or toroidal).

### **THXY**

First rotation about local Z (positive X toward Y).

### **THYZ**

Second rotation about local X (positive Y toward Z).

### **THZX**

Third rotation about local Y (positive Z toward X).

### **PAR1**

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

**PAR2**

Used for spheroidal systems. If  $KCS = 2$ ,  $PAR2$  = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

**Notes**

Defines and activates a local coordinate system by origin location and orientation angles relative to the active coordinate system. This local system becomes the active coordinate system, and is automatically aligned with the active system (i.e., x is radial if a cylindrical system is active, etc.). Nonzero rotation angles (degrees) are relative to this automatic rotation. See the **CS**, **CSKP**, **CSWPLA**, and **LOCAL** commands for alternate definitions. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

**Menu Paths**

**This command cannot be accessed from a menu.**

---

**CLOG**, *IR*, *IA*, *--*, *--*, *Name*, *--*, *--*, *FACTA*, *FACTB*

**Forms the common log of a variable**

[POST26: Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**IR**

Arbitrary reference number assigned to the resulting variable (2 to NV **[NUMVAR]**). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

**IA**

Reference number of the variable to be operated on.

*--*, *--*

Unused fields.

**Name**

Thirty-two character name for identifying the variable on printouts and displays. Embedded blanks are compressed for output.

*--*, *--*

Unused fields.

**FACTA**

Scaling factor applied to variable *IA* (defaults to 1.0).

**FACTB**

Scaling factor (positive or negative) applied to the operation (defaults to 1.0).

**Notes**

Forms the common log of a variable according to the operation:

$$IR = FACTB * \text{LOG}(FACTA \times IA)$$



## Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Common Log

**/CLOG**, *Fname*, *Ext*, --

**Copies the session log file to a named file.**

[SESSION: Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path to which the log file is to be copied (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

### *Ext*

Filename extension (8 character maximum).

--

Unused field.

## Notes

This command is valid in any processor, but only during an interactive run.

## Menu Paths

**This command cannot be accessed from a menu.**

## CLRMShLN

**Clears meshed entities.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

When you use the GUI method to set the number of elements on specified lines, and any of those lines is connected to one or more meshed lines, areas, or volumes, ANSYS gives you the option to clear the meshed entities. (This occurs only when you perform this operation via the GUI; ANSYS does not provide such an option when you use the command method [[LESIZE](#)].)

If you activate the mesh clearing option, the program invokes an ANSYS macro, **CLRMShLN**, that clears the meshed entities. This macro name will appear in the log file (*Jobname*.LOG). This macro is for the ANSYS program's internal use only. This command is not intended to be typed in directly in an ANSYS session, although it can be included in an input file for batch input or for use with the [/INPUT](#) command.

## Menu Paths

This command cannot be accessed from a menu.

**CM**, *Cname*, *Entity*

**Groups geometry items into a component.**

DATABASE: Components

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Cname*

An alphanumeric name used to identify this component. *Cname* may be up to 32 characters, beginning with a letter and containing only letters, numbers, and underscores. Component names beginning with an underscore (e.g., `_LOOP`) are reserved for use by ANSYS and should be avoided. Components named "ALL," "STAT," and "DEFA" are not permitted. Overwrites a previously defined name.

### *Entity*

Label identifying the type of geometry items to be grouped:

#### **VOLU**

Volumes.

#### **AREA**

Areas.

#### **LINE**

Lines.

#### **KP**

Keypoints.

#### **ELEM**

Elements.

#### **NODE**

Nodes.

## Notes

Components may be further grouped into assemblies [**CMGRP**]. The selected items of the specified entity type will be stored as the component. Use of this component in the select command [**CMSEL**] causes all these items to be selected at once, for convenience.

A component is a grouping of some geometric entity that can then be conveniently selected or unselected. A component may be redefined by reusing a previous component name. The following entity types may belong to a component: nodes, elements, keypoints, lines, areas, and volumes. A component may contain only 1 entity type, but an individual item of any entity may belong to any number of components. Once defined, the items contained in a component may then be easily selected or unselected [**CMSEL**]. Components may be listed [**CMLIST**], modified [**CMMOD**] and deleted [**CMDELE**]. Components may also be further grouped into assemblies [**CMGRP**]. Other entities associated with the entities in a component (e.g., the lines and keypoints associated with areas) may be selected by the **ALLSEL** command.

An item will be deleted from a component if it has been deleted by another operation (see the **KMODIF** command for an example). Components are automatically updated to reflect deletions of one or more of

their items. Components are automatically deleted and a warning message is issued if all their items are deleted. Assemblies are also automatically updated to reflect deletions of one or more of their components or subassemblies, but are not deleted if all their components and subassemblies are deleted.

This command is valid in any processor.

## Menu Paths

Utility Menu>Select>Comp/Assembly>Create Component

---

**CMACEL**, *CM\_NAME*, *CMACEL\_X*, *CMACEL\_Y*, *CMACEL\_Z*

**Specifies the translational acceleration of an element component**

SOLUTION: Inertia

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

*CM\_NAME*

The name of the element component (8-character maximum).

*CMACEL\_X*, *CMACEL\_Y*, *CMACEL\_Z*

Acceleration of the element component *CM\_NAME* in the global Cartesian X, Y, and Z axis directions, respectively.

## Notes

The **CMACEL** command specifies the translational acceleration of the element component in each of the global Cartesian (X, Y, and Z) axis directions.

You can use the **CMACEL** command to specify translational, acceleration-based loading on up to 100 element components.

Components for which you want to specify acceleration loading must consist of elements only. The elements you use cannot be part of more than one component, and elements that share nodes cannot exist in different element components. You cannot apply the loading to an assembly of element components.

To simulate gravity (by using inertial effects), accelerate the structure in the direction opposite to gravity. For example, apply a positive *CMACELY* to simulate gravity acting in the negative Y direction. Units are length/time<sup>2</sup>.

You can define the acceleration for the following analyses types:

- Static (**ANTYPE**,STATIC)
- Harmonic (**ANTYPE**,HARMIC), full or mode superposition method
- Transient (**ANTYPE**,TRANS), full or mode superposition method
- Substructure (**ANTYPE**,SUBSTR)

Accelerations are combined with the element mass matrices to form a body force load vector term. Units of acceleration and mass must be consistent to give a product of force units.

In a modal harmonic or transient analysis, you must apply the load in the modal portion of the analysis. ANSYS calculates a load vector and writes it to the mode shape file, which you can apply via the **LVSCALE** command.

The **CMACEL** command does not support these elements: **FLUID79**, **FLUID80**, and **FLUID81**.

The **CMACEL** command supports tabular boundary conditions (**%TABNAME\_X%**, **%TABNAME\_Y%**, and **%TABNAME\_Z%**) for **CMACEL\_X**, **CMACEL\_Y**, and **CMACEL\_Z** input values (**\*DIM**) as a function of both time and frequency for full transient and harmonic analyses.

Related commands for inertia loads are **ACEL**, **CGLOC**, **CGOMGA**, **DCGOMG**, **DOMEGA**, **OMEGA**, **CMOMEGA**, and **CMDOMEGA**.

This command is also valid in **/PREP7**.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Gravity>On Components**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Gravity>On Components**

**/CMAP**, *Fname*, *Ext*, *--*, *Kyword*, *NCNTR*

**Changes an existing or creates a new color mapping table.**

[GRAPHICS: Set Up](#)

[DISPLAY: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

If blank, restore color map.

### *Ext*

Filename extension (8 character maximum).

--

Unused field.

### *Kyword*

Keyword indicating the disposition of the color map file.

#### **(blank)**

Loads existing color map file.

#### **CREATE**

Starts CMAP utility and modifies or creates the specified file.

#### **SAVE**

Writes the active color map to the specified file, which can be imported into future ANSYS sessions.

### *NCNTR*

Number of contours to be defined by the **/CMAP** command (max = 128). If no value is specified, CMAP defaults to 9, even if an existing file is being modified.

## Command Default

Use predefined ANSYS color map table.

## Notes

Reads the color map file (RGB index specifications) to change from current specifications. Only one color map may be active at a time.

For 2-D drivers (especially Win32c), modifying the color map can produce anomalies, including legend/contour disagreement.

Changing the color map in ANSYS with the **/CMAP** command will change the meaning of the color labels on the **/COLOR** command. See **/COLOR** for other color controls.

This command is valid anywhere.

## Menu Paths

Utility Menu>PlotCtrls>Redirect Plots>To GRPH File  
 Utility Menu>PlotCtrls>Redirect Plots>To HPGL File  
 Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File  
 Utility Menu>PlotCtrls>Redirect Plots>To PSCR File  
 Utility Menu>PlotCtrls>Style>Colors>Default Color Map

---

## CMATRIX, SYMFAC, Condname, NUMCOND, GRNDKEY, Capname

**Performs electrostatic field solutions and calculates the self and mutual capacitances between multiple conductors.**

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

### **SYMFAC**

Geometric symmetry factor. Capacitance values are scaled by this factor which represents the fraction of the total device modeled. Defaults to 1.

### **Condname**

Alpha-numeric prefix identifier used in defining named conductor components.

### **NUMCOND**

Total Number of Components. If a ground is modeled, it is to be included as a component. If a ground is not modeled, but infinite elements are used to model the far-field ground, a named component for the far-field ground is not required.

### **GRNDKEY**

Ground key:

**0**

Ground is one of the components, which is not at infinity.

**1**

Ground is at infinity (modeled by infinite elements or a Trefftz domain).

**Capname**

Array name for computed capacitance matrix. Defaults to **CMATRIX**.

**Notes**

To invoke the **CMATRIX** macro, the exterior nodes of each conductor must be grouped into individual components using the **CM** command. Each set of independent components is assigned a component name with a common prefix followed by the conductor number. A conductor system with a ground must also include the ground nodes as a component. The ground component is numbered last in the component name sequence.

A Ground Capacitance matrix is a matrix relating charge to a voltage vector. A ground matrix can not be applied to a circuit modeler such as SPICE. The Lumped Capacitance matrix is a matrix formed by a combination of lumped "arrangements" of voltage differences between conductors. You can use the lumped capacitance terms in a circuit modeler to represent capacitances between conductors.

You must enclose all name-strings in single quotes in the **CMATRIX** command line.

See the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) and [HMAGSOLV](#) in the [Low-Frequency Electromagnetic Analysis Guide](#) for details.

**Menu Paths**

**Main Menu>Solution>Solve>Electromagnet>Static Analysis>Capac Matrix**

---

**CMDELE**, *Name*

**Deletes a component or assembly definition.**

[DATABASE: Components](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Name**

Name of the component or assembly whose definition is to be removed.

**Notes**

Entities contained in the component, or the components within the assembly, are unaffected. Only the grouping relationships are deleted. Assemblies are automatically updated to reflect deletion of their components or subassemblies, but they are not automatically deleted when all their components or subassemblies are deleted.

This command is valid in any processor.

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Picked Lines**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements**

**Utility Menu>Select>Comp/Assembly>Delete Comp/Assembly**

---

**CMDOMEGA**, *CM\_NAME*, *DOMEGAX*, *DOMEGAY*, *DOMEGAZ*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

**Specifies the rotational acceleration of an element component about a user-defined rotational axis.**

SOLUTION: Inertia

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

***CM\_NAME***,

The name of the element component (8 character maximum).

***DOMEGAX*, *DOMEGAY*, *DOMEGAZ***

If the *X2*, *Y2*, *Z2* fields are not defined, *DOMEGAX*, *DOMEGAY*, and *DOMEGAZ* specify the components of the rotational acceleration vector in the global Cartesian X, Y, Z directions.

If the *X2*, *Y2*, *Z2* fields are defined, only *DOMEGAX* is required. *DOMEGAX* specifies the scalar rotational acceleration about the rotational axis. The rotational direction of *DOMEGAX* is designated either positive or negative, and is determined by the "right hand rule."

***X1*, *Y1*, *Z1***

If the *X2*, *Y2*, *Z2* fields are defined, *X1*, *Y1*, and *Z1* define the coordinates of the beginning point of the rotational axis vector. Otherwise, *X1*, *Y1*, and *Z1* are the coordinates of a point through which the rotational axis passes.

***X2*, *Y2*, *Z2***

The coordinates of the end point of the rotational axis vector.

## Notes

Specifies the rotational acceleration components *DOMEGAX*, *DOMEGAY*, and *DOMEGAZ* of an element component *CM\_NAME* about a user-defined rotational axis. The rotational axis can be defined either as a vector passing through a single point, or a vector connecting two points.

You can use the **CMDOMEGA** command to specify acceleration based loading on up to 100 rotational element components.

You can define rotational acceleration and rotational axis with the **CMDOMEGA** command for STATIC, HARMIC (full), TRANS (full), and SUBSTR analyses. Rotational velocities are combined with the element mass matrices to form a body force load vector term. Units are radians/time<sup>2</sup>.

The **CMDOMEGA** command supports tabular boundary conditions (%*TABNAME\_X*%, %*TABNAME\_Y*%, and %*TABNAME\_Z*%) for *CMDOMEGA\_X*, *CMDOMEGA\_Y*, and *CMDOMEGA\_Z* input values (**\*DIM**) for full transient and harmonic analyses.

Related commands are **ACEL**, **CGLOC**, **CGLOC**, **OMEGA**, **CMOMEGA**, **DCGOMG**, **DOMEGA**.

You can use the **CMDOMEGA** command in conjunction with any one of the following two groups of commands, but not with both groups simultaneously:

GROUP ONE: **OMEGA**, **DOMEGA**.

GROUP TWO: **CGOMGA**, **DCGOMG**, **CGLOC**.

Components for which you want to specify rotational loading must consist of elements only. The elements you use cannot be part of more than one component, and elements that share nodes cannot exist in different element components. You cannot apply the loading to an assembly of element components.

In a modal harmonic or transient analysis, you must apply the load in the modal portion of the analysis. ANSYS calculates a load vector and writes it to the mode shape file, which you can apply via the **LVSCALE** command.

See [Acceleration Effect](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By Axis**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By origin**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpt**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpts**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Node**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Accel>On Component**

**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By Axis**

**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By origin**

**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpt**

**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpts**

**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Node**

**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Nodes**

**Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Accel>On Component**

---

**CMEDIT**, *Aname*, *Oper*, *Cnam1*, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*

**Edits an existing assembly.**

**DATABASE:** Components

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Aname***

Name of the assembly to be edited.

***Oper***

Operation label:



**ADD**

To add more components. The level of any assembly to be added must be lower than that of the assembly *Aname* (see **CMGRP** command).

**DELE**

To remove components.

*Cnam1, Cnam2, Cnam3, . . . , Cnam7*

Names of components and assemblies to be added to or deleted from the assembly.

**Notes**

This command is valid in any processor.

**Menu Paths**

**Utility Menu>Select>Comp/Assembly>Edit Assembly**

**CMGRP**, *Aname, Cnam1, Cnam2, Cnam3, Cnam4, Cnam5, Cnam6, Cnam7, Cnam8*

**Groups components and assemblies into an assembly.**

[DATABASE: Components](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Aname***

An alphanumeric name used to identify this assembly. *Aname* may be up to 32 characters, beginning with a letter and containing only letters, numbers, and underscores. Overwrites a previously defined *Aname* (and removes it from higher level assemblies, if any).

***Cnam1, Cnam2, Cnam3, . . . , Cnam8***

Names of existing components or other assemblies to be included in this assembly.

**Notes**

Groups components and other assemblies into an assembly identified by a name. **CMGRP** is used for the initial definition of an assembly. An assembly is used in the same manner as a component. Up to 5 levels of assemblies within assemblies may be used.

An assembly is a convenient grouping of previously defined components and other assemblies. Assemblies may contain components only, other assemblies, or any combination. A component may belong to any number of assemblies. Up to 5 levels of nested assemblies may be defined. Components and assemblies may be added to or deleted from an existing assembly by the **CMEDIT** command. Once defined, an assembly may be listed, deleted, selected, or unselected using the same commands as for a component. Assemblies are automatically updated to reflect deletions of one or more of their components or lower-level assemblies. Assemblies are not automatically deleted when all their components or subassemblies are deleted.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>Select>Comp/Assembly>Create Assembly**

**CMLIST**, *Name*, *Key*, *Entity*

**Lists the contents of a component or assembly.**

DATABASE: Components

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**Name**

Name of the component or assembly to be listed (if blank, list all selected components and assemblies).  
If *Name* is specified, then *Entity* is ignored.

**Key**

Expansion key:

**0**

Do not list individual entities in the component.

**1 or EXPA**

List individual entities in the component.

**Entity**

If *Name* is blank, then the following entity types can be specified:

**VOLU**

List the volume components only.

**AREA**

List the area components only.

**LINE**

List the line components only.

**KP**

List the keypoint components only

**ELEM**

List the element components only.

**NODE**

List the node components only.

**Notes**

This command is valid in any processor. For components, it lists the type of geometric entity. For assemblies, it lists the components and/or assemblies that make up the assembly.

Examples of possible usage:

**CMLIST** - List all selected components.

**CMLIST**, , EXPA - List all selected components and for each component list the underlying entity ID's.

**CMLIST**,*Name* - List the specified component.

**CMLIST**,*Name*,EXPA - List specified component along with all underlying entity ID's.

**CMLIST**, , EXPA,*Entity* - List all selected components of specified entity type. For each component also list the underlying entity ID's.

**Menu Paths**

**Utility Menu>List>Components**

Utility Menu>List>Other>Components  
 Utility Menu>Select>Comp/Assembly>List Comp/Assembly

**CMMOD**, *Cname*, *Keyword*, *Value*

**Modifies the specification of a component.**

DATABASE: Components

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Cname***

Name of the existing component or assembly to be modified.

***Keyword***

The label identifying the type of value to be modified.

NAME - Modify the NAME of the component

***Value***

If *Keyword* is NAME, then the value is the alphanumeric label to be applied. See the **CM** command for naming convention details. If a component named *Value* already exists, the command will be ignored and an error message will be generated.

**Notes**

The naming conventions for components, as specified in the **CM** command, apply for **CMMOD** (32 characters, "ALL," "STAT" and "DEFA" are not allowed, etc.). However, if you choose a component name that is already designated for another component, an error message will be issued and the command will be ignored.

This command is valid in any processor.

**Menu Paths**

**This command cannot be accessed from a menu.**

**CMOMEGA**, *CM\_NAME*, *OMEGAX*, *OMEGAY*, *OMEGAZ*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

**Specifies the rotational velocity of an element component about a user-defined rotational axis.**

SOLUTION: Inertia

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

***CM\_NAME***

The name of the element component (eight-character maximum).

***OMEGAX*, *OMEGAY*, *OMEGAZ***

If the *X2*, *Y2*, *Z2* fields are not defined, *OMEGAX*, *OMEGAY*, and *OMEGAZ* specify the components of the rotational velocity vector in the global Cartesian X, Y, Z directions.

If the *X2*, *Y2*, *Z2* fields are defined, only *OMEGAX* is required. *OMEGAX* specifies the scalar rotational velocity about the rotational axis. The rotational direction of *OMEGAX* is designated either positive or negative, and is determined by the "right hand rule."

***X1, Y1, Z1***

If the *X2, Y2, Z2* fields are defined, *X1, Y1*, and *Z1* define the coordinates of the beginning point of the rotational axis vector. Otherwise, *X1, Y1*, and *Z1* are the coordinates of a point through which the rotational axis passes.

***X2, Y2, Z2***

The coordinates of the end point of the rotational axis vector.

## Notes

Specifies the rotational velocity components *OMEGAX*, *OMEGAY*, and *OMEGAZ* of an element component *CM\_NAME* about a user-defined rotational axis. The rotational axis can be defined either as a vector passing through a single point or a vector connecting two points.

You can use the **CMOMEGA** command to specify velocity-based loading on up to 100 rotational element components.

You can define rotational velocity and rotational axis for these analysis types:

- Static (**ANTYPE**,STATIC)
- Harmonic (**ANTYPE**,HARMIC) -- Full or modal superposition
- Transient (**ANTYPE**,TRANS) -- Full or modal superposition
- Substructuring (**ANTYPE**,SUBSTR)
- Modal (**ANTYPE**,MODAL)

Rotational velocities are combined with the element mass matrices to form a body force load vector term. Units are radians/time. Related commands are **ACEL**, **CGLOC**, **CGLOC**, **CGOMGA**, **CMDOMEGA**, **DCGOMG**, **DOMEGA**.

You can use the **CMOMEGA** command in conjunction with either one of the following two groups of commands, but not with both groups simultaneously:

GROUP ONE: **OMEGA**, **DOMEGA**.

GROUP TWO: **CGOMGA**, **DCGOMG**, **CGLOC**.

Components for which you want to specify rotational loading must consist of elements only. The elements you use cannot be part of more than one component, and elements that share nodes cannot exist in different element components. You cannot apply the loading to an assembly of element components.

If you have applied the Coriolis effect (**CORIOLIS**) using a [stationary reference frame](#), the **CMOMEGA** command takes the gyroscopic damping matrix into account for the elements listed under “*Stationary Reference Frame*” in the notes section of the **CORIOLIS** command. ANSYS verifies that the rotation vector axis is parallel to the axis of the element; if not, the gyroscopic effect is not applied. If you issue a **CMOMEGA** command when the Coriolis or gyroscopic effect is present, a subsequently issued **OMEGA** command has no effect.

The **CMOMEGA** command supports tabular boundary conditions (*%TABNAME\_X%*, *%TABNAME\_Y%*, and *%TABNAME\_Z%*) for *CMOMEGA\_X*, *CMOMEGA\_Y*, and *CMOMEGA\_Z* input values (**\*DIM**) for full transient and harmonic analyses.

In a mod-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. ANSYS calculates a load vector and writes it to the *MODE* file, which you can apply via the **LVSCALE** command.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By Axis**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By origin**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpt**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpts**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Node**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Veloc>On Component**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By Axis**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By origin**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpt**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpts**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Node**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Nodes**  
**Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Veloc>On Component**

## CMPLOT, *Label, Entity, Keyword*

Plots the entities contained in a component or assembly.

DATABASE: Components

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Label*

Name of the component or assembly to be plotted.

### **(blank)**

All selected components and assemblies are plotted (default). If fewer than 11 components are selected, then all are plotted. If more than 11 components are selected, then only the first 11 are plotted.

### **ALL**

All selected components are plotted. If number of selected components is greater than 11, then the legend showing component names will not be shown.

### **N**

Next set of defined components and assemblies is plotted.

**P**

Previous set of defined components and assemblies is plotted.

**Cname**

The specified component or assembly is plotted.

**SetNo.**

The specified set number is plotted.

**Entity**

If *Label* is BLANK or ALL, then the following entity types can be specified:

**VOLU**

Plot the volume components only.

**AREA**

Plot the area components only.

**LINE**

Plot the line components only.

**KP**

Plot the keypoint components only.

**ELEM**

Plot the element components only.

**NODE**

Plot the node components only.

**Keyword**

For *Keyword* = ALL, plot the specified component name in the *Label* field in the context of all entities of the same type. Not valid if *Label* field is BLANK or ALL.

## Notes

Components are plotted with their native entities. For assemblies, all native entities for the underlying component types are plotted simultaneously. Although more components can be plotted, the legend displays only 11 at a time. When more than eleven are plotted, the legend is not displayed.

Possible usage:

**CMPLOT**,*CNAME* - Plots the specified component (if selected).

**CMPLOT**,*CNAME*, ALL - Plot component in the context of all other selected entity components of the same type as the component.

**CMPLOT** - Plot the first eleven selected components.

**CMPLOT**,ALL - Plot all selected components.

**CMPLOT**,N or **CMPLOT**,P - Plot next or previous set of eleven components.

**CMPLOT**,ALL,*Entity* - Plot all selected components of type specified in *Entity*.

**CMPLOT**,*Entity* - Plot components of type specified in *Entity*, from the first eleven components.

**CMPLOT**,N,*Entity* - Plot components of type specified in *Entity*, if any, from the next set of eleven components (substitute P for N to plot from previous set).

This command is valid in any processor.

## Menu Paths

Utility Menu>Plot>Components>By Name / Set Number

Utility Menu>Plot>Components>Next Set

Utility Menu>Plot>Components>Previous Set

Utility Menu>Plot>Components>Selected Components

**CMROTATE**, *CM\_NAME*, *ROTATX*, *ROTATY*, *ROTATZ*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

Specifies the rotational velocity of an element component about a user-defined rotational axis

SOLUTION: Inertia

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *CM\_NAME*

The name of the element component (eight-character maximum).

### *ROTATX*, *ROTATY*, *ROTATZ*

If the *X2*, *Y2*, *Z2* fields are not defined, *ROTATX*, *ROTATY*, and *ROTATZ* specify the components of the rotational angle vector in the global Cartesian X, Y, Z directions.

If the *X2*, *Y2*, *Z2* fields are defined, only *ROTATX* is required. *ROTATX* specifies the scalar rotational velocity about the rotational axis. The rotational direction of *ROTATX* is designated either positive or negative, and is determined by the "right hand rule."

### *X1*, *Y1*, *Z1*

If the *X2*, *Y2*, *Z2* fields are defined, *X1*, *Y1*, and *Z1* define the coordinates of the beginning point of the rotational axis vector. Otherwise, *X1*, *Y1*, and *Z1* are the coordinates of a point through which the rotational axis passes.

### *X2*, *Y2*, *Z2*

The coordinates of the end point of the rotational axis vector.

## Notes

The **CMROTATE** command specifies the rotational motion velocity components *ROTATX*, *ROTATY*, and *ROTATZ* of an element component *CM\_NAME* about a user-defined rotational axis. The rotational axis can be defined either as a vector passing through a single point or a vector connecting two points.

You can use the **CMROTATE** command to specify angular rotation-based displacement or angular velocity on up to 100 rotational element components.

You can define rotational velocity and rotational axis for static analyses (**ANTYPE,STATIC**) only.

The command sets the constant rotational velocity on the nodes of the element component despite any deformation at the nodes. This feature is primarily useful for generating [sliding contact at frictional contact interfaces](#) in a [brake squeal analysis](#).

A [brake squeal analysis](#) generally involves a linear perturbation modal analysis subsequent to a large-deformation static analysis with the Newton Raphson option set as **NROPT,UNSYM**. Therefore, **CMROTATE** is not applicable for multiple load step solves using the **LSSOLVE** command.

The **CMROTATE** command typically applies to brake squeal modeling where surface-to-surface contact occurs between the brake pad and the rotating disk. The applicable contact elements, therefore, are [CONTA173](#), [CONTA174](#), and [CONTA175](#).

This command is also valid in PREP7.

## Menu Paths

**This command cannot be accessed from a menu.**

---

### **CMSEL**, *Type*, *Name*, *Entity*

**Selects a subset of components and assemblies.**

DATABASE: Components

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### **Type**

Label identifying the type of select:

##### **S**

Select a new set (default).

##### **R**

Reselect a set from the current set.

##### **A**

Additionally select a set and extend the current set.

##### **U**

Unselect a set from the current set.

##### **ALL**

Also select all components.

##### **NONE**

Unselect all components.

#### **Name**

Name of component or assembly whose items are to be selected (valid only if *Type* = S, R, A, or U).

Graphical picking is enabled if *Type* is blank and *Name* = PICK (or simply "P").

#### **Entity**

If *Name* is blank, then the following entity types can be specified:

##### **VOLU**

Select the volume components only.

##### **AREA**

Select the area components only.

##### **LINE**

Select the line components only.

##### **KP**

Select the keypoint components only.



**ELEM**

Select the element components only.

**NODE**

Select the node components only.

## Notes

Selecting by component is a convenient adjunct to individual item selection (e.g., **VSEL**, **ESEL**, etc.). **CMSEL**, **ALL** allows you to select components **in addition** to other items you have already selected.

If  $Type = R$  for an assembly selection [**CMSEL**, $R$ , $\langle$ assembly-name $\rangle$ ], the reselect operation is performed on each component in the assembly in the order in which the components make up the assembly. Thus, if one reselect operation results in an empty set, subsequent operations will also result in empty sets. For example, if the first reselect operation tries to reselect node 1 from the selected set of nodes 3, 4, and 5, the operation results in an empty set (that is, no nodes are selected). Since the current set is now an empty set, if the second reselect operation tries to reselect any nodes, the second operation also results in an empty set, and so on. This is equivalent to repeating the command **CMSEL**, $R$ , $\langle$ component-name $\rangle$  once for each component making up the assembly.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Joint Elems**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Joint Elems**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Joint Elems**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Joint Elems**  
**Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements**  
**Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Joint Elems**  
**Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Joint Elems**  
**Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Joint Elems**  
**Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Joint Elems**  
**Utility Menu>Select>Comp/Assembly>Select Comp/Assembly**

---

**CMSFILE**, *Option*, *Fname*, *Ext*, *CmsKey*

**Specifies a list of component mode synthesis (CMS) results files for plotting results on the assembly.**

[POST1: Special Purpose](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *Option*

Specifies the command operation:

**ADD**

Add the specified component results file (*Fname*) to the list of files to plot. This option is the default.

**DELETE**

Remove the specified component results file (*Fname*) from the list of files to plot.

**LIST**

List all specified component results files.

**CLEAR**

Clear all previous files added.

**ALL**

Add *all* component results (.rst) files from the working directory to the list of files to plot.

**Fname**

The file name (with full directory path) of the component results file. The default file name is the Jobname (specified via the **/FILENAME** command).

**Ext**

The file name (*Fname*) extension. The default extension is .rst.

**CmsKey**

Valid only when adding a results file (*Option* = ADD or ALL), this key specifies whether or not to check the specified .rst file to determine if it was created via a CMS expansion pass:

**ON**

Check (default).

**OFF**

Do not check.

## Command Default

If issued with no arguments, the **CMSFILE** command uses these defaults:

```
CMSFILE,ADD,Jobname,rst,ON
```

The command adds the component results file Jobname.rst.

## Notes

The **CMSFILE** command specifies the list of component mode synthesis (CMS) results files to include when plotting the mode shape of an assembly.

During postprocessing (**/POST1**) of a CMS analysis, issue the **CMSFILE** command to point to component results files of interest. (You can issue the command as often as needed to include all or some of the component results files.) Issue the **SET** command to acquire the frequencies and mode shapes from substeps for all specified results files. Execute a plot (**PLNSOL**) or print (**PRNSOL**) operation to display the mode shape of the entire assembly.

When you specify a results file to add to the plot list, the default behavior of the command (*CmsKey* = ON) is to first verify that the file is from a CMS analysis and that the frequencies of the result sets on the file match the frequencies on the first file in the list. If *CmsKey* = OFF, you can add any .rst file to the list of files to plot, even if the file was not expanded via a CMS expansion pass.

If *CmsKey* = ON (default), output from the command appears as: ADD CMS FILE = filename.rst. If *CmsKey* = OFF, output from the command appears as: ADD FILE = filename.rst.

If *Option* = DELETE or CLEAR, you must clear the database (**/CLEAR**), then re-enter the postprocessor (**/POST1**) and issue a **SET** command for the change to take effect on subsequent plots.

Clearing the database does *not* clear the list of files specified via the **CMSFILE** command. Specify *Option* = CLEAR to clear the list of files.

## Menu Paths

Main Menu>General Postproc>Data & File Opts

**CMSOPT**, *CMSMETH*, *NMODE*, *FREQB*, *FREQE*, *FBDDEF*, *FBDVAL*, *IOKEY*

**Specifies component mode synthesis (CMS) analysis options.**

[SOLUTION: Analysis Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### ***CMSMETH***

The component mode synthesis [method](#) to use. This value is required.

#### **FIX**

Fixed-interface method.

#### **FREE**

Free-interface method.

#### **RFFB**

Residual-flexible free-interface method.

### ***NMODE***

The number of normal modes extracted and used in superelement generation. This value is required and the minimum is 1.

### ***FREQB***

Beginning, or lower end, of frequency range of interest. This value is optional.

ANSYS sets this value to zero if the residual-flexible free-interface method (*CMSMETH* = RFFB) is specified.

### ***FREQE***

Ending, or upper end, of frequency range of interest. This value is optional.

### ***FBDDEF***

In a free-interface (*CMSMETH* = FREE) or residual-flexible free-interface (*CMSMETH* = RFFB) CMS analysis, the method to use for defining free body modes:

#### **FNUM**

The number (*FBDVAL*) of rigid body modes in the calculation.

#### **FTOL**

Employ a specified tolerance (*FBDVAL*) to determine rigid body modes in the calculation.

#### **FAUTO**

Automatically determine rigid body modes in the calculation. This method is the default.

#### **RIGID**

If no rigid body modes exist, define your own via the **RIGID** command.

### ***FBDVAL***

In a free-interface CMS analysis (*CMSMETH* = FREE), the number of rigid body modes if *FBDDEF* = FNUM (where the value is an integer from 0 through 6), or the tolerance to employ if *FBDDEF* = FTOL (where the value is a positive real number representing rad/sec). This value is required only when *FBDDEF* = FNUM or *FBDDEF* = FTOL; otherwise, any specified value is ignored.

**IOKEY**

The key to output transformation matrix for FIX/FREE interface method.

**TCMS**

Write transformation matrix of the nodal component defined by **OUTPR** command to .tcms file. Please refer to [TCMS File Format](#) in the *Programmer's Manual* for more information on the .tcms file format.

**Command Default**

Issuing the **CMSOPT** command with no arguments is invalid. You must specify at least the CMS method (*CMSMETH*) and the number of modes (*NMODE*). In a free-interface (*CMSMETH* = FREE) or residual-flexible free-interface (*CMSMETH* = RFFB) CMS analysis, the default method for determining rigid body modes is FAUTO (automatic).

**Notes**

CMS employs the Block Lanczos eigensolution method in the [generation pass](#).

CMS does not yet support damping matrix reduction. ANSYS sets the matrix generation key to 2 automatically (**SEOPT**,*SEMATR*).

CMS does not support the **SEOPT**,,,,,RESOLVE command. Instead, ANSYS sets the expansion method for the expansion pass (*EXPMTH*) to BACKSUB.

This command is also valid in **/PREP7**.

For more information about performing a CMS analysis, see "[Component Mode Synthesis](#)" in the *Advanced Analysis Techniques Guide*.

If *IOKEY* = TCMS is used to output the transformation matrix, then only ITEM = NSOL is valid in the **OUTPR** command. In the interactive sessions, the transformation matrix will not be output if the model has more than 10 elements.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>Analysis Options**

---

**CMWRITE**, *Fname*, *Ext*, *--*, *--*, *Fmat*

**Writes components and assemblies to a file.**

DATABASE: Components

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

**Ext**

Filename extension (8 character maximum).

The extension defaults to CM if *Fname* is blank.

--

Unused field.

**Fmat**

Format of the output file (defaults to BLOCKED).

**BLOCKED**

Blocked format. This format allows faster reading of the file.

**UNBLOCKED**

Unblocked format.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**CNCHECK**, *Option, RID1, RID2, RINC, PairType, TRlevel*

**Provides and/or adjusts the initial status of contact pairs.**

PREP7: Database  
SOLUTION: Analysis Options

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

**Option**

Option to be performed:

**DETAIL**

List all contact pair properties (default).

**SUMMARY**

List only the open/closed status for each contact pair.

**POST**

Execute a partial solution to write the initial contact configuration to a results file, *Jobname*.RCN.

**ADJUST**

Physically move contact nodes to target to close gap or reduce penetration.

**RESET**

Reset target element and contact element key options and real constants to their default values.

**AUTO**

Automatically sets certain real constants and key options to recommended values or settings in order to achieve better convergence based on overall contact pair behaviors.

**TRIM**

Trim contact pair (remove certain contact and target elements).

**UNSE**

Unselect certain contact and target elements.

**RID1, RID2, RINC**

The range of real constant pair ID's for which *Option* will be performed. If *RID2* is not specified, it defaults to *RID1*. If no value is specified, all contact pairs in the selected set of elements are considered.

**PairType**

Type of contact pairs to be trimmed/unselected/auto-set (used only when *Option* = TRIM, UNSE, or AUTO):

**ANY**

All types (default).

**MPC**

MPC-based contact pairs (KEYOPT(2) = 2).

**BOND**

Bonded contact pairs (KEYOPT(12) = 3, 5, 6).

**NOSP**

No separation contact pairs (KEYOPT(12) = 2, 4).

**INAC**

Inactive contact pairs (symmetric contact pairs for MPC contact or KEYOPT(8) = 2).

**TRlevel**

Trimming level (used only when *Option* = TRIM or UNSE):

**(blank)**

Normal trimming (default): remove/unselect contact and target elements which are in far-field.

**AGGRE**

Aggressive trimming: remove/unselect contact and target elements which are in far-field, and certain elements in near-field.

## Notes

The **CNCHECK** command provides information for surface-to-surface, node-to-surface, and line-to-line contact pairs (element types [TARGE169](#), [TARGE170](#), [CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#), [CONTA175](#), [CONTA176](#), [CONTA177](#)). All contact and target elements of interest, along with the solid elements and nodes attached to them, must be selected for the command to function properly.

**CNCHECK** is available in both the PREP7 and SOLUTION processors, but only before the first solve operation (that is, only before the first load step or the first substep).

The following additional notes are available:

[Option = POST](#)

[Option = ADJUST](#)

[Option = RESET](#)

[Option = AUTO](#)

[Removing or Unselecting Contact and Target Elements](#)

### ***Option = POST***

The command **CNCHECK,POST** solves the initial contact configuration in one substep. After issuing this command, you can postprocess the contact result items as you would for any other converged load step; however, only the contact status, contact penetration or gap, and contact pressure will have meaningful values. Other contact quantities (friction stress, sliding distance, chattering) will be available but are not useful.

Because *Option* = POST forces a solve operation, the [PrepPost \(PP\) license](#) does not work with **CNCHECK,POST**.

If **CNCHECK,POST** is issued within the solution processor, the **SOLVE** command that solves the first load step of your analysis should appear in a different step, as shown in the following example:

```
/SOLU
CNCHECK,POST
FINISH
. . .

/SOLU
SOLVE
FINISH
. . .
```

**CNCHECK,POST** writes initial contact results to a results file named `Jobname.RCN`. When postprocessing the initial contact state, you need to explicitly read results from this file using the **FILE** and **SET,FIRST** commands in POST1 to properly read the corresponding contact data. Otherwise, the results file may be read improperly. The following example shows a valid command sequence for plotting the initial contact gap:

```
/SOLU
CNCHECK,POST
FINISH
/POST1
FILE,Jobname,RCN
SET,FIRST
PLNSOL,CONT,GAP,0,1
FINISH
. . .
```

### ***Option = ADJUST***

You can issue **CNCHECK,ADJUST** to physically move contact nodes to the target surface. See [Physically Moving Contact Nodes Towards the Target Surface](#) in the *Contact Technology Guide* for more information. Similar to the POST option, if **CNCHECK,ADJUST** is issued within the solution processor, the **SOLVE** command that solves the first load step of your analysis should appear in a different step:

```
/SOLU
CNCHECK,ADJUST
FINISH
. . .

/SOLU
SOLVE
FINISH
. . .
```

### ***Option = RESET***

The command **CNCHECK,RESET** allows you to reset all but a few key options and real constants associated with the specified contact pairs (*RID1*, *RID2*, *RINC*) to their default values.

The following key options and real constants remain unchanged when this command is issued:

Element type	Key options not affected by RESET	Real constants not affected by RESET
TARGE169, TARGE170	KEYOPT(2), KEYOPT(3)	R1, R2
CONTA171, CONTA172	KEYOPT(1), KEYOPT(3)	R1, R2

CONTA173, CONTA174, CONTA175, CONTA176, CONTA177	KEYOPT(1)	R1, R2
--	-----------	--------

### **Option = AUTO**

The command **CNCHECK,AUTO** automatically changes certain default or undefined key options and real constants to optimized settings or values. The changes are based on overall contact pair behaviors. In general, this command improves convergence for nonlinear contact analysis.

The tables below list typical KEYOPT and real constant settings implemented by **CNCHECK,AUTO**. The actually settings implemented for your specific model may vary from what is described here. You should always verify the modified settings by issuing **CNCHECK,DETAIL** to list current contact pair properties.

KEY-OPT	Description	Default (0 or blank)	CNCHECK,AUTO
1	Selects DOF set	Structural DOFs	Automatic selection based on DOFs of underlying elements.
2	Contact algorithm	Augmented Lagrange	1 - Penalty for rigid-rigid contact [1].
4	Location of contact detection point	Gauss point	2 - Normal to target surface if KEYOPT(2) > 1.
5	CNOF/ICONT adjustment	No adjustment	1 - Auto CNOF if tiny gap exists.
6	Contact stiffness variation	Use default range	1 - Nominal refinement for opening contact or underlying elements having <b>TB</b> plasticity. or 2 - Aggressive refinement for opening contact and underlying elements having <b>TB</b> plasticity.
7	Element level time increment control	No control	1 - Automatic bisection for self contact.
8	Assymmetric contact selection	No action	2 - Auto selection if KEYOPT(2) > 1.
9	Effect of initial penetration or gap	Include all	1 - Exclude if KEYOPT(5) = 1, or if ICONT was previously specified.
10	Contact stiffness update	Between load steps	2 - Between iterations except when underlying elements are superelements. [2]

1. Set to 0 if KEYOPT(2) > 1 for debonding.
2. Set to 0 if underlying elements are superelements, or if KEYOPT(9) = 2 was previously specified.

Real Constants		Description	Default	CNCHECK,AUTO
No.	Name			
3	FKN	Normal penalty stiffness factor	1	Set to 5 if KEYOPT(9) = 2 (ramp initial penetration) and KEYOPT(10) > 0.



6	PINB	Pinball region	see [7]	Cut in half if spurious contact is detected or contact searching is slow.
14	TCC	Thermal contact conductance	0	Calculated as a function of highest conductivity of underlying element and overall model size.
19	ECC	Electric contact conductance	0	Calculated as a function of highest permittivity or lowest resistivity of underlying element and overall model size.
26	MCC	Magnetic contact permeance	0	Calculated as a function of highest emissivity of underlying element and overall model size.

1. PINB default depends on contact behavior (rigid vs. flexible target), **NLGEOM**, ON or OFF, KEYOPT(9) setting, KEYOPT(12) setting, and the value of real constant CNOF (see [Using PINB](#)).

**CNCHECK,AUTO** also sets **PRED**,OFF for the case of a [force-distributed constraint](#) defined via [MPC contact](#).

## Removing or Unselecting Contact and Target Elements

For performance reasons, the program uses a subset of nodes and elements based on the specified contact regions (*RID1*, *RID2*, *RINC*) when executing **CNCHECK,POST** or **CNCHECK,ADJUST**.

You can issue **CNCHECK,TRIM** or **CNCHECK,UNSEL** to remove or unselect contact and target elements which are in far-field (that is, open and not near contact), or even in near field if aggressive trimming logic is used (*TRlevel* = AGGRE).

## Menu Paths

Main Menu > Preprocessor > Modeling > Create > Contact Pair

**CNKMOD**, *ITYPE*, *KNUM*, *VALUE*

**Modifies contact element key options.**

SOLUTION: Analysis Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *ITYPE*

Contact element type number as defined on the **ET** command.

### *KNUM*

Number of the KEYOPT to be modified (KEYOPT(*KNUM*)).

### *VALUE*

Value to be assigned to the KEYOPT.

## Notes

The **CNKMOD** command has the same syntax as the **KEYOPT** command. However, it is valid only in the SOLUTION processor. This command is intended primarily for modifying contact interface behavior between load steps in a linear perturbation analysis (see the **PERTURB** command); it allows the user to control the contact status locally per contact pair. Therefore, this command is limited to changing the contact interface behavior key option: KEYOPT(12) of [CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#), [CONTA175](#), [CONTA176](#), and [CONTA177](#); and KEYOPT(10) of [CONTA178](#).

In the linear perturbation analysis procedure, contact status can also be controlled or modified by the **PERTURB** command. The contact status always follows local controls defined by the **CNKMOD** command first, and is then adjusted by the global sticking or bonded setting (*ContKey* STICKING or BONDED) on the **PERTURB** command.

## Menu Paths

This command cannot be accessed from a menu.

**CNVTOL**, *Lab*, *VALUE*, *TOLER*, *NORM*, *MINREF*

Sets convergence values for nonlinear analyses.

[SOLUTION: Nonlinear Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *Lab*

Valid convergence labels. If STAT, list the status of the currently specified criteria.

Discipline	Displacement Convergence		Force Convergence	
	Label	Label Description	Label	Label Description
Structural	U	Displacements	F	Forces
	ROT	Rotations	M	Moments
	HDSP	Hydrostatic Pressure	DVOL	Volume
Thermal	TEMP	Temperature	HEAT	Heat Flow
Fluid	PRES	Pressures	FLOW	Fluid Flow
	V	Velocity	VF	Fluid Force
Electric	VOLT	Voltage	AMPS	Current Flow
	EMF	Electromotive Force	CURT	Current Flow
	CURR	Current Flow	VLTG	Voltage Drop
Magnetic	MAG	Scalar Potential	FLUX	Scalar Flux
	A	Vector Potential	CSG	Current Segments

### **VALUE**

Typical reference value for the above label for this analysis. If negative, and if this convergence label was previously specified explicitly, then convergence based on this label is removed. (A negative *VALUE* will not remove a default convergence label.) Defaults to the maximum of a program calculated reference or *MINREF*. For degrees of freedom, the reference is based upon the selected *NORM* and the current total DOF value. For forcing quantities, the reference is based upon the selected *NORM* and the applied loads.

### **TOLER**

When **SOLCONTROL,ON**, tolerance about *VALUE* defaults to 0.005 (0.5%) for force and moment, 1.0E-4 (0.01%) for DVOL, 0.05 (5%) for displacement when rotational DOFs are not present, and 0.05 (5%) for HDSP. When **SOLCONTROL,OFF**, the defaults are 0.001 (0.1%) for force and moment, and 1.0E-5 (0.001%) for DVOL.

**NORM**

Specifies norm selection:

**2**

L2 norm (check SRSS value) (default).

**1**

L1 norm (check absolute value sum).

**0**

Infinite norm (check each DOF separately).

**MINREF**

The minimum value allowed for the program calculated reference value. If negative, no minimum is enforced. Used only if *VALUE* is blank. Defaults to 0.01 for force, moment, and volume convergence, 1.0E-6 for heat flow, 1.0E-12 for VOLT and AMPS, 1.0E-6 for HDSP, and 0.0 otherwise. When **SOLCONTROL,OFF**, defaults to 1.0 for force, moment, and volume convergence. The default for heat flow (1.0E-6), VOLT and AMPS (1.0E-12), and others are independent of the **SOLCONTROL** setting.

**Command Default**

For static or transient analysis, check the out-of-balance load for any active DOF using the default *VALUE*, *TOLER*, *NORM*, and *MINREF*. Also check the displacement convergence for some problems. For analyses that include CONTA17x contact elements, displacement/rotation convergence checking is off by default. For harmonic magnetic analysis, check the out-of-balance of the degrees of freedom.

**Notes**

The default values given for this command assume **SOLCONTROL,ON** (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL,ON** and **SOLCONTROL,OFF**.

Values may be set for the degrees of freedom (DOF) and/or the out-of-balance load for the corresponding forcing quantities. When the GUI is on, if a "Delete" operation in a Nonlinear Convergence Criteria dialog box writes this command to a log file (*Jobname.LOG* or *Jobname.LGW*), you will observe that *Lab* is blank, *VALUE* = -1, and *TOLER* is an integer number. In this case, the GUI has assigned a value of *TOLER* that corresponds to the location of a chosen convergence label in the dialog box's list. It is *not* intended that you type in such a location value for *TOLER* in an ANSYS session. However, a file that contains a GUI-generated **CNVTOL** command of this form can be used for batch input or with the **/INPUT** command.

If you have multiple DOF types, issue **CNVTOL** for each DOF, because the default will be reset for the other DOFs the first time **CNVTOL** is issued.

This command is also valid in PREP7.

Convergence norms specified with **CNVTOL** may be graphically tracked while the solution is in process using the ANSYS program's Graphical Solution Tracking (GST) feature. Use the **/GST** command to turn GST on or off. By default, GST is ON for interactive sessions and OFF for batch runs.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Convergence Crit**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Harmonic**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Static**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Transient**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear**  
**Main Menu>Solution>Load Step Opts>Nonlinear>Convergence Crit**  
**Main Menu>Solution>Load Step Opts>Nonlinear>Harmonic**  
**Main Menu>Solution>Load Step Opts>Nonlinear>Static**  
**Main Menu>Solution>Load Step Opts>Nonlinear>Transient**

**/COLOR**, *Lab*, *Clab*, *N1*, *N2*, *NINC*

**Specifies the color mapping for various items.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lab*

Apply color to the items specified by the following labels:

#### **AXES**

Determines the color (specified in next argument, *Clab*) that the axes of a graph will be plotted in.

#### **AXNUM**

Determines the color (specified in next argument, *Clab*) that the numbering on the axes of a graph will be plotted in.

#### **NUM**

Discretely numbered items (such as element types, element materials, etc., as shown on the **/PNUM** command). Also specify number (1 to 11) in the *N1* field. For example, **/COLOR,NUM,RED,3** will assign the color red to all items having the discrete number 3 (material displays would show elements having material 3 as red).

#### **OUTL**

Outline of elements, areas, and volumes. Ex: **/COLOR,OUTL,BLUE**.

#### **ELEM**

Elements. Use *N1*, *N2*, *NINC* fields for element numbers.

#### **LINE**

Solid model lines. Use *N1*, *N2*, *NINC* fields for line numbers.

#### **AREA**

Solid model areas. Use *N1*, *N2*, *NINC* fields for area numbers.

#### **VOLU**

Solid model volumes. Use *N1*, *N2*, *NINC* fields for volume numbers.

#### **ISURF**

Isosurfaces (surfaces of constant stress, etc.). This option is particularly useful when capturing frames for animating a single isosurface value.

#### **WBAK**

Window background. Use *N1*, *N2*, *NINC* fields for window numbers. The options that you select using *Lab* = PBAK will supersede those applied using *Lab* = WBAK.

#### ***b.c.label***

Boundary condition label. Enter U, ROT, TEMP, PRES, V, ENKE, ENDS, SP01 through SP06 or their user-defined names, VOLT, MAG, A, EMF, CURR, F, M, HEAT, FLOW, VF, AMPS, FLUX, CSG, CURT, VLTG,

MAST, CP, CE, NFOR, NMOM, RFOR, RMOM, PATH. See the **/PBC** command for boundary condition label definitions.

**GRBAK**

Graph background.

**GRID**

Graph grid lines.

**AXLAB**

Graph X and Y axis labels.

**CURVE**

Graph curves (identify curve numbers (1-10) in *N1*, *N2*, *NINC* fields).

**CM**

Component group. Use *N1* field for component name, ignore *N2* and *NINC*.

**CNTR**

ANSYS contour stress colors. The maximum number of contours available is 128. The number of colors that can be specified interactively (GUI) is 9. (**/CONTOUR**, , 9). Any other setting will yield inconsistent results.

**SMAX**

Specifies that all stress values above the maximum value entered in **/CONTOUR** will be displayed in the color designated in the *C1ab* field. Defaults to dark grey.

**SMIN**

Specifies that all stress values below the minimum value entered in **/CONTOUR** will be displayed in the color designated in the *C1ab* field. Defaults to dark grey.

**PBAK**

Activates background shading options (see command syntax at end of argument descriptions below). The options that you select using *Lab* = PBAK will supersede those applied using *Lab* = WBAK.

***C1ab***

Valid color labels are:

**BLAC (0)**

Black

**MRED (1)**

Magenta-Red

**MAGE (2)**

Magenta

**BMAG (3)**

Blue-Magenta

**BLUE (4)**

Blue

**CBLU (5)**

Cyan-Blue

**CYAN (6)**

Cyan

**GCYA ((7)**

Green-Cyan

**GREE (8)**

Green

**YGRE (9)**

Yellow-Green

**YELL (10)**

Yellow

**ORAN (11)**

Orange

**RED (12)**

Red

**DGRA (13)**

Dark Gray

**LGRA (14)**

Light Gray

**WHIT (15)**

White

***N1, N2, NINC***

Apply color to *Lab* items numbered *N1* to *N2* (defaults to *N1*) in steps of *NINC* (defaults to 1). If *N1* is blank, apply color to entire selected range. If *Lab* is CM, use component name for *N1* and ignore *N2* and *NINC*. If *N1* = P, graphical picking of elements, lines, areas and volumes is enabled; you can assign colors to the entities via the picker. When picking is enabled, the *Lab* and *Clab* fields are ignored.

If *Lab* = PBAK, the command format is **/COLOR, PBAK, *Key\_On\_Off*, *KEY\_TYPE*, *KEY\_INDEX***.

The options that you select using *Lab* = PBAK will supersede those applied using *Lab* = WBAK.

***Key\_On\_Off***

Turns the background colors on and off. Acceptable values are ON (1) and OFF (0).

***KEY\_TYPE***

Determines the type of background. Acceptable values are 0 (smooth shading left to right), 1 (smooth shading top to bottom), 2 (smooth shading right to left), 3 (smooth shading bottom to top), and -1 (textured image background)

***KEY\_INDEX***

An integer value that corresponds to a background color or texture. If *Key\_Type* is -1, the background will correspond to values specified in the **/XTRE** command. If *Key\_Type* is any other acceptable value, the background will correspond to the color values listed above under *Clab*.

## **Command Default**

Use the default color mapping.

## **Notes**

Issue **/COLOR,STAT** to display the current color mapping. Issue **/COLOR,DEFA** to reset the default color mapping.

---

## Note

Color labels may also be reassigned any "color" with the **/CMAP** command.

This command is valid anywhere.

## Menu Paths

**Utility Menu>PlotCtrls>Style>Colors>BC Colors**  
**Utility Menu>PlotCtrls>Style>Colors>Component Colors**  
**Utility Menu>PlotCtrls>Style>Colors>Entity Colors**  
**Utility Menu>PlotCtrls>Style>Colors>Graph Colors**  
**Utility Menu>PlotCtrls>Style>Colors>Numbered Item Colors**  
**Utility Menu>PlotCtrls>Style>Colors>Window Colors**

---

## **/COM**, *Comment*

**Places a comment in the output.**

**SESSION: List Controls**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Comment*

Comment string, up to 75 characters.

## Notes

The output from this command consists of the comment string. This command is similar to **C\*\*\*** except that the comment produced by **C\*\*\*** is more easily identified in the output. Parameter substitution within the comment occurs for every valid expression delimited by percent (%) signs. Enclosing such an expression in single quotes prevents parameter substitution.

Another way to include a comment is to precede it with a ! character (on the same line). The ! may be placed anywhere on the line, and any input following it is ignored as a comment. No output is produced by such a comment, but the comment line is included on the log file. This is a convenient way to annotate the log file.

This command is valid anywhere.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**\*COMP**, *Matrix, Algorithm, Threshold*

**Compresses the columns of a matrix using a specified algorithm.**

[APDL: Matrix Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Matrix*

Name of the matrix to compress.

### *Algorithm*

Algorithm to use:

#### **SVD** --

Singular value decomposition algorithm (default).

#### **MGS** --

Modified Gram-Schmidt algorithm.

### *Threshold*

Numerical threshold value used to manage the compression. Default value for SVD is 1E-7; default value for MGS is 1E-14.

## Notes

The algorithms available through this command are only applicable to dense matrices that were created using the **\*DMAT** command.

Columns which are linearly dependent on others are removed, leaving the independent or basis vectors. The matrix is resized according to the new size determined by the algorithm.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## COMPRESS

**Deletes all specified sets.**

[AUX3: Results Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Issue this command to delete all sets specified with the **DELETE** command.

## Menu Paths

**This command cannot be accessed from a menu.**

---



**CON4**, *XCENTER*, *YCENTER*, *RAD1*, *RAD2*, *DEPTH*

**Creates a conical volume anywhere on the working plane.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***XCENTER*, *YCENTER***

Working plane X and Y coordinates of the center axis of the cone.

***RAD1*, *RAD2***

Radii of the faces of the cone. *RAD1* defines the bottom face and will be located on the working plane. *RAD2* defines the top face and is parallel to the working plane. A value of zero or blank for either *RAD1* or *RAD2* defines a degenerate face at the center axis (i.e., the vertex of the cone). The same value for both *RAD1* and *RAD2* defines a cylinder instead of a cone.

***DEPTH***

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the cone. *DEPTH* cannot be zero (see "[Notes](#)" (p. 301) below).

## Notes

Defines a solid conical volume with either the vertex or a face anywhere on the working plane. The cone must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) The face or faces will be circular (each area defined with four lines), and they will be connected with two areas (each spanning 180°). See the **CONE** command for an alternate way to create cones.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Primitives>Cone**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Cone>By Picking**

**CONE**, *RBOT*, *RTOP*, *Z1*, *Z2*, *THETA1*, *THETA2*

**Creates a conical volume centered about the working plane origin.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***RBOT*, *RTOP***

Radii of the bottom and top faces of the cone. A value of zero or blank for either *RBOT* or *RTOP* defines a degenerate face at the center axis (i.e., the vertex of the cone). The same value for both *RBOT* and *RTOP* defines a cylinder instead of a cone.

***Z1*, *Z2***

Working plane Z coordinates of the cone. The smaller value is always associated with the bottom face.

***THETA1*, *THETA2***

Starting and ending angles (either order) of the cone. Used for creating a conical sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0° and the ending angle defaults to 360°. See the [Modeling and Meshing Guide](#) for an illustration.

## Notes

Defines a solid conical volume centered about the working plane origin. The non-degenerate face (top or bottom) is parallel to the working plane but not necessarily coplanar with (i.e., "on") the working plane. The cone must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) For a cone of 360°, top and bottom faces will be circular (each area defined with four lines), and they will be connected with two areas (each spanning 180°). See the **CON4** command for an alternate way to create cones.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Volumes>Cone>By Dimensions**

**/CONFIG**, *Lab*, *VALUE*

**Assigns values to ANSYS configuration parameters.**

[SESSION: Run Controls](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

### *Lab*

Configuration parameter to be changed:

#### **NRES**

*VALUE* is maximum number of results sets (substeps) allowed on the result file. Defaults to 1000. For MFX analyses, defaults to 5000. Minimum is 10.

#### **NORSTGM**

Option to write or not write geometry data to the results file. *VALUE* is either 0 (write geometry data) or 1 (do not write geometry data). Useful when complex analyses will create abnormally large files. Default is 0.

#### **NBUF**

*VALUE* is the number of buffers (1 to 32) per file in the solver. Defaults to 4.

#### **LOCFL**

File open and close actions. For *VALUE* use: 0 for global (default); 1 for local. Applicable to `File.EROT`, `File.ESAV`, and `File.EMAT`. Typically used for large problems where locally closed files may be deleted earlier in the run with the **/FDELE** command.

#### **SZBIO**

*VALUE* is the record size (1024 to 4194304) of binary files (in integer words). Defaults to 16384 (system dependent).

#### **ORDER**

Automatic reordering scheme. For *VALUE* use: 0 for **WSORT,ALL**; 1 for **WAVES**; 2 for both **WSORT,ALL** and **WAVES** (default).

#### **FSPLIT**

Defines split points for binary files. *VALUE* is the file split point in megawords and defaults to the maximum file size for the system.

#### **MXND**

Maximum number of nodes. If not specified, defaults to 100 at first encounter. Dynamically expanded by doubling, even at first encounter, when maximum is exceeded.

**MXEL**

Maximum number of elements. Default and expansion as for MXND.

**MXKP**

Maximum number of keypoints. Default and expansion as for MXND.

**MXLS**

Maximum number of lines. Default and expansion as for MXND.

**MXAR**

Maximum number of areas. Default and expansion as for MXND.

**MXVL**

Maximum number of volumes. Default and expansion as for MXND.

**MXRL**

Maximum number of sets of real constants (element attributes). Default and expansion as for MXND.

**MXCP**

Maximum number of sets of coupled degrees of freedom. Default and expansion as for MXND.

**MXCE**

Maximum number of constraint equations. Default and expansion as for MXND.

**NOELDB**

Option to write or not write results into the database after a solution. When *VALUE* = 0 (default), write results into the database. When *VALUE* = 1, do not write results into the database.

**DYNA\_DBL**

Option to invoke the double precision version of the explicit dynamics solver LS-DYNA. When *VALUE* = 0 (default), the single precision version is used. When *VALUE* = 1, the double precision version is used.

**STAT**

Displays current values set by the **/CONFIG** command.

**VALUE**

Value (an integer number) assigned to the configuration parameter.

**Notes**

All configuration parameters have initial defaults, which in most cases do not need to be changed. Where a specially configured version of the ANSYS program is desired, the parameters may be changed with this command. Issue **/CONFIG,STAT** to display current values. Changes must be defined before the parameter is required. These changes (and others) may also be incorporated into the `config130.ans` file which is read upon execution of the program (see [The Configuration File](#) in the *Basic Analysis Guide*). If the same configuration parameter appears in both the configuration file and this command, this command overrides.

Distributed ANSYS uses the default FSPLIT value, and forces NOELDB = 1 and NORSTGM = 0 for all results files except the local results files (for example, `Jobname.n.RST`) written for each process during a distributed solution. For local results files, no geometry is written (NORSTGM = 1). The FSPLIT, NOELDB, and NORSTGM options cannot be changed when using Distributed ANSYS.

The **/CONFIG** command is not valid for the ANSYS Multiphysics 1, 2, or 3 products.

The ANSYS Multi-field solver (MFS and MFX) does not support **/CONFIG,NOELDB,1**. The ANSYS Multi-field solver needs the updated ANSYS database.

## Product Restrictions

Command Option Lab	Available Products
MXND	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS
MXEL	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Menu Paths

Utility Menu>List>Status>Configuration

**CONJUG**, *IR*, *IA*, *--*, *--*, *Name*, *--*, *--*, *FACTA*

Forms the complex conjugate of a variable.

POST26: Operations

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### *IR*

Arbitrary reference number assigned to the resulting variable (2 to NV [[NUMVAR](#)]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

### *IA*

Reference number of the variable to be operated on.

*--*, *--*

Unused fields.

### *Name*

Thirty-two character name for identifying the variable on printouts and displays. Embedded blanks are compressed for output.

*--*, *--*

Unused fields.

### *FACTA*

Scaling factor (positive or negative) applied to variable (default to 1.0).

## Notes

Used only with harmonic analyses ([ANTYPE](#), [HARMIC](#)).

## Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Complex Conjugat

**/CONTOUR**, *WN*, *NCONT*, *VMIN*, *VINC*, *VMAX*

**Specifies the uniform contour values on stress displays.**

GRAPHICS:Labeling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### **WN**

Window number (or ALL) to which command applies (defaults to 1).

#### **NCONT**

Number of contour values. *NCONT* defaults to 9 for X11 or WIN32 and to 128 for X11c or WIN32C. The default graphics window display for 3-D devices is a smooth continuous shading effect that spans the maximum of 128 contours available. Use the **/DV3D** command to create defined banding for your contour values (values of 9 and 128 are displayed in smooth shading only). The legend, however, will display only nine color boxes, which span the full range of colors displayed in the graphics window.

#### **VMIN**

Minimum contour value. If *VMIN* = AUTO, automatically calculate contour values based upon *NCONT* uniformly spaced values over the min-max extreme range. Or, if *VMIN* = USER, set contour values to those of the last display (useful when last display automatically calculated contours).

#### **VINC**

Value increment (positive) between contour values. Defaults to  $(VMAX-VMIN)/NCONT$ .

#### **VMAX**

Maximum contour value. Ignored if both *VMIN* and *VINC* are specified.

### **Command Default**

Nine contour values uniformly spaced between the extreme values, or no contours if the ratio of range to minimum value (or range to maximum if minimum = 0) is less than 0.001.

### **Notes**

See the **/CVAL** command for alternate specifications. Values represent contour lines in vector mode, and the algebraic maximum of contour bands in raster mode.

---

#### **Note**

No matter how many contours (*NCONT*) are specified by **/CONTOUR**, the actual number of contours that appear on your display depends also on the device name, whether the display is directed to the screen or to a file, the display mode (vector or raster), and the number of color planes. (All these items are controlled by **/SHOW** settings.) In any case, regardless of whether they are smoothed or banded, only 128 contours can be displayed. See [Creating Geometric Results Displays](#) in the *Basic Analysis Guide* for more information on changing the number of contours.

If the current ANSYS graphics are *not* displayed as Multi-Plots, then the following is true: If the current device is a 3-D device [**/SHOW,3D**], the model contours in all active windows will be the same, even if separate **/CONTOUR** commands are issued for each active window. For efficiency, ANSYS 3-D graphics logic maintains a single data structure (segment), which contains precisely one set of contours. The program displays the same segment in all windows. The view settings of each window constitute the only differences in the contour plots in the active windows.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Uniform Contours

---

**/COPY**, *Fname1*, *Ext1*, --, *Fname2*, *Ext2*, --

**Copies a file.**

SESSION: Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***Fname1***

File name to be copied and its directory path (248 characters maximum for both file name and directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to the current Jobname.

### ***Ext1***

Filename extension (8 character maximum).

--

Unused field.

### ***Fname2***

File name to be created and its directory path (248 characters maximum for both file name and directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

*Fname2* defaults to *Fname1*.

### ***Ext2***

Filename extension (8 character maximum).

*Ext2* defaults to *Ext1*.

--

Unused field.

## Notes

The original file is untouched. Ex: **/COPY**,A,,,B copies file A to B in the same directory. **/COPY**,A,DAT,,,INP copies the file A.DAT to A.INP. See the [Operations Guide](#) for details. ANSYS binary and ASCII files can be copied.

## Menu Paths

Utility Menu>File>File Operations>Copy

---

**CORIOLIS**, *Option*, *--*, *--*, *RefFrame*, *RotDamp*

**Applies the Coriolis effect to a rotating structure.**

SOLUTION: *Inertia*

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***Option***

Flag to activate or deactivate the Coriolis effect:

**1 (ON or YES)**

Activate. This value is the default.

**0 (OFF or NO)**

Deactivate.

*--*, *--*

Unused fields.

***RefFrame***

Flag to activate or deactivate a stationary reference frame.

**1 (ON or YES)**

Activate.

**0 (OFF or NO)**

Deactivate. This value is the default.

***RotDamp***

Flag to activate or deactivate rotating damping effect.

**1 (ON or YES)**

Activate.

**0 (OFF or NO)**

Deactivate. This value is the default.

**Notes**

The **CORIOLIS** command is used for analyses in either a rotating or a stationary reference frame, and performs differently according to the designated *RefFrame* value. Specific restrictions and elements apply to each case, as follows:

**Rotating Reference Frame** (*RefFrame* = OFF):

The command applies the Coriolis effect in the following structural element types: **MASS21**, **SHELL181**, **PLANE182**, **PLANE183**, **SOLID185**, **SOLID186**, **SOLID187**, **BEAM188**, **BEAM189**, **SOLSH190**, **SHELL281**, **PIPE288** and **PIPE289**. It also applies this effect in the **PLANE223**, **SOLID226**, and **SOLID227** analyses with structural degrees of freedom.

In a rotating reference frame, both the coriolis and **spin-softening** effects contribute to the gyroscopic moment. Therefore, ANSYS applies spin-softening by default for dynamic analyses. If a rotational velocity is specified (or), centrifugal forces will be included.

To include coriolis effects in a large deflection prestressed analysis (**NLGEOM**, ON), issue both the **CORIOLIS** and the **OMEGA** (or **CMOMEGA**) commands in the static prestress portion of the analysis.

In a nonlinear transient analysis (**ANTYPE**,TRANS and **NLGEOM**, ON), any spinning motion applied through either the **IC** of the **D** commands will include the coriolis effect without having to issue the **CORIOLIS** command. Refer to [Rotating Structure Analysis](#) in the *Advanced Analysis Techniques Guide* for more information.

**Stationary Reference Frame** (*RefFrame* = ON):

The command activates the gyroscopic damping matrix in the following structural elements: **MASS21**, **BEAM188**, **SHELL181**, **BEAM189**, **SOLID185**, **SOLID186**, **SOLID187**, **SOLID272**, **SOLID273**, **SHELL281**, **PIPE288**, and **PIPE289**.

The rotating structure must be axisymmetric about the axis of rotation.

Static analysis (**ANTYPE**, STATIC) does not support Coriolis effects with a stationary reference frame. However, you can include the gyroscopic effects in a small deflection prestresses analysis (**NLGEOM**, OFF and **PSTRES**, ON) by issuing the **CORIOLIS** command in the static prestressed dynamic analysis.

Rotating damping effect (*RotDamp* = ON) applies only for the stationary reference frame. Therefore, this effect is supported only by the elements listed above that generate a gyroscopic damping matrix. Proportional damping must be present in the element (**MP**,DAMP or **BETAD**). It is also supported by element **COMBI214** with non zero and axisymmetric damping characteristics (non zero real constants C11=C22 and C21=C12=0).

For more information about using the **CORIOLIS** command, see [Rotating Structure Analysis](#) in the *Advanced Analysis Techniques Guide* and also in the *Rotordynamic Analysis Guide*. For details about the Coriolis and gyroscopic effects, see the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>Coriolis**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>Coriolis**

---

## COUPLE

Specifies "Node coupling" as the subsequent status topic.

PREP7: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (*Jobname.LOG*) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>Preprocessor>Coupled Sets**

---



**COVAL**, *TBLNO1*, *TBLNO2*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

**Defines PSD cospectral values.**

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**TBLNO1**

First input PSD table number associated with this spectrum.

**TBLNO2**

Second input PSD table number associated with this spectrum.

**SV1**, **SV2**, **SV3**, . . . , **SV7**

PSD cospectral values corresponding to the frequency points [**PSDFRQ**].

## Notes

Defines PSD cospectral values to be associated with the previously defined frequency points. Two table references are required since values are off-diagonal terms. Unlike autospectra [**PSDVAL**], the cospectra can be positive or negative. The cospectral curve segment where there is a sign change is interpolated linearly (the rest of the curve segments use log-log interpolation). For better accuracy, choose as small a curve segment as possible wherever a sign change occurs.

Repeat **COVAL** command using the same table numbers for additional points. This command is valid for **SPOPT**, PSD only.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Cospectral**  
**Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Cospectral**

**CP**, *NSET*, *Lab*, *NODE1*, *NODE2*, *NODE3*, *NODE4*, *NODE5*, *NODE6*, *NODE7*, *NODE8*, *NODE9*, *NODE10*, *NODE11*, *NODE12*, *NODE13*, *NODE14*, *NODE15*, *NODE16*, *NODE17*

**Defines (or modifies) a set of coupled degrees of freedom.**

PREP7: Coupled DOF

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NSET**

Set reference number:

**n**

Arbitrary set number.

**HIGH**

The highest defined coupled set number will be used (default, unless *Lab* = ALL). This option is useful when adding nodes to an existing set.

**NEXT**

The highest defined coupled set number plus one will be used (default if *Lab* = ALL). This option automatically numbers coupled sets so that existing sets are not modified.

**Lab**

Degree of freedom label for coupled nodes (in the nodal coordinate system). Defaults to label previously defined with *NSET* if set *NSET* already exists. A different label redefines the previous label associated with *NSET*. Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations) (in radians); HDSP (hydrostatic pressure). Thermal labels: TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature). Fluid labels: PRES (pressure); VX, VY, or VZ (velocities). Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials); CURR (current). Explicit analysis labels: UX, UY, or UZ (displacements).

When *Lab* = ALL:

- Sets are generated for *each* active degree of freedom (that is, one set for the UX degree of freedom, another set for UY, etc.), and *NSET* is incremented automatically to prevent overwriting existing sets.
- Existing sets are *not* modified. *NSET* must be a new set number *n* or NEXT.
- The degree of freedom set is determined according to all element types defined and the **DOF** command, if used.
- Hydrostatic pressure (HDSP) is not included.

ALL is the only label applicable to FLOTRAN.

**NODE1, NODE2, NODE3, . . . , NODE17**

List of nodes to be included in set. Duplicate nodes are ignored. If a node number is input as negative, the node is deleted from the coupled set. The first node in the list is the primary (retained) node. If *NODE1* = ALL, *NODE2* through *NODE17* are ignored and all selected nodes (**NSEL**) are included in the set. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*.

**Notes**

Do not include the same degree of freedom in more than one coupled set. Repeat **CP** command for additional nodes.

Coupling degrees of freedom into a set causes the results calculated for one member of the set to be the same for all members of the set. Coupling can be used to model various joint and hinge effects. A more general form of coupling can be done with constraint equations (**CE**). For structural analyses, a list of nodes is defined along with the nodal directions in which these nodes are to be coupled. As a result of this coupling, these nodes are forced to take the same displacement in the specified *nodal coordinate direction*. The amount of the displacement is unknown until the analysis is completed. A set of coupled nodes which are not coincident, or which are not along the line of the coupled displacement direction, may produce an applied moment which will not appear in the reaction forces. The actual degrees of freedom available for a particular node depends upon the degrees of freedom associated with element types (**ET**) at that node. For scalar field analysis, this command is used to couple nodal temperatures, pressures, voltages, etc.

For an explicit dynamic analysis, the only valid DOF labels for coupling are UX, UY, and UZ. Since the rotational DOF (ROTX, ROTY, ROTZ) are not allowed, The **CP** family of commands should not be used in an explicit analysis to model rigid body behavior that involves rotations. If **CP** is used in this manner, it could lead to nonphysical responses.

A set of coupled nodes which are not coincident, or which are not along the line of the coupled displacement direction, produce an artificial moment constraint. If the structure rotates, a moment may be produced in the coupled set in the form of a force couple. This moment is in addition to the real reaction forces and may make it appear that moment equilibrium is not satisfied by just the applied forces and the reaction forces. Note, however, that in an explicit dynamic analysis, this artificial moment will not be produced. Rather, just the applied forces and the reaction forces will satisfy the moment equilibrium in the model. Thus, in an explicit analysis, the magnitude of nodal displacements for this set of nodes will depend on the distance from each node to the center of the coupled set, and the direction of displacement will depend on the resulting moment. This may lead to a nonphysical response in some cases.

Additional sets of coupled nodes may be generated from a specified set. Degrees of freedom are coupled within a set but are not coupled between sets. No degree of freedom should appear in more than one coupled set. Such an appearance would indicate that at least two sets were in fact part of a single larger set. The first degree of freedom of the coupled set is the "prime" degree of freedom. All other degrees of freedom in the coupled sets are eliminated from the solution matrices by their relationship to the prime degree of freedom. Forces applied to coupled nodes (in the coupled degree of freedom direction) will be summed and applied to the prime degree of freedom. Output forces are also summed at the prime degree of freedom. Degrees of freedom with specified constraints (**D**) should not be included in a coupled set (unless the degree of freedom is prime).

If master degrees of freedom are defined for coupled nodes, only the prime degree of freedom should be so defined. The use of coupled nodes reduces the set of coupled degrees of freedom to only one degree of freedom.

In FLOTRAN, you can apply periodic boundary conditions using the **CP** command along with the **PERI** macro. Attempts to use the **CP** command outside the context of the **PERI** macro may lead to unexpected results.

## Menu Paths

**Main Menu>Preprocessor>Coupling / Ceqn>Couple DOFs**

**Main Menu>Preprocessor>Coupling / Ceqn>Cupl DOFs w/Mstr**

---

**CPCYC**, *Lab*, *TOLER*, *KCN*, *DX*, *DY*, *DZ*, *KNONROT*

**Couples the two side faces of a cyclically symmetric model for loadings that are the same on every segment.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***Lab***

Degree of freedom label for coupled nodes (in the nodal coordinate system). If ALL, use all appropriate labels. Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations, in radians).

### ***TOLER***

Tolerance for coincidence (based on maximum coordinate difference in each global Cartesian direction for node locations and on angle differences for node orientations). Defaults to 0.0001. Only nodes within the tolerance are considered to be coincident for coupling.

### ***KCN***

In coordinate system *KCN*, node 1 of CP + dx dy dz = node 2 of CP.

***DX, DY, DZ***

Node location increments in the active coordinate system (DR, D $\theta$ , DZ for cylindrical, DR, D $\theta$ , D $\Phi$  for spherical or toroidal).

***KNONROT***

When *KNONROT* = 0, the nodes on coupled sets are rotated into coordinate system *KCN* (see **NROTAT** command description). When *KNONROT* = 1, the nodes are not rotated, and you should make sure that coupled nodal DOF directions are correct.

## Notes

Cyclic coupling requires identical node and element patterns on the low and high sector boundaries. The MSHCOPY operation allows convenient generation of identical node and element patterns. See [Using CPCYC and MSHCOPY Commands](#) in the *Modeling and Meshing Guide* for more information.

Although developed primarily for use with cyclically symmetric models, your use of the **CPCYC** command is not limited to cyclic symmetry analyses.

## Menu Paths

**Main Menu>Preprocessor>Coupling / Ceqn>Offset Nodes**

---

**CPDELE, NSET1, NSET2, NINC, Nsel**

**Deletes coupled degree of freedom sets.**

PREP7: Coupled DOF

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NSET1, NSET2, NINC***

Delete coupled sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL, *NSET2* and *NINC* are ignored and all coupled sets are deleted.

***Nsel***

Additional node selection control:

**ANY**

Delete coupled set if *any* of the selected nodes are in the set (default).

**ALL**

Delete coupled set only if *all* of the selected nodes are in the set.

## Notes

See the **CP** command for a method to delete individual nodes from a set.

## Menu Paths

**Main Menu>Preprocessor>Coupling / Ceqn>Del Coupled Sets**  
**Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements**

---

**CPINTF**, *Lab*, *TOLER*

**Defines coupled degrees of freedom at an interface.**

PREP7: Coupled DOF

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Lab**

Degree of freedom label for coupled nodes (in the nodal coordinate system). If ALL, use all appropriate labels *except* HDSP. Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations, in radians), HDSP (hydrostatic pressure). Thermal labels: TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature). Fluid labels: PRES (pressure); VX, VY, or VZ (velocities). Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials); CURR (current).

**TOLER**

Tolerance for coincidence (based on maximum coordinate difference in each global Cartesian direction for node locations and on angle differences for node orientations). Defaults to 0.0001. Only nodes within the tolerance are considered to be coincident for coupling.

**Notes**

Defines coupled degrees of freedom between coincident nodes (within a tolerance). May be used, for example, to "button" together elements interfacing at a seam, where the seam consists of a series of node pairs. One coupled set is generated for each selected degree of freedom for each pair of coincident nodes. For more than two coincident nodes in a cluster, a coupled set is generated from the lowest numbered node to each of the other nodes in the cluster. Coupled sets are generated only within (and not between) clusters. If fewer than all nodes are to be checked for coincidence, use the **NSSEL** command to select nodes. Coupled set reference numbers are incremented by one from the highest previous set number. Use **CPLIST** to display the generated sets. Only nodes having the same nodal coordinate system orientations ("coincident" within a tolerance) are included. Use the **CEINTF** command to connect nodes by constraint equations instead of by coupling. Use the **EINTF** command to connect nodes by line elements instead of by coupling.

**Menu Paths**

Main Menu>Preprocessor>Coupling / Ceqn>Coincident Nodes

**/CPLANE**, *KEY*

**Specifies the cutting plane for section and capped displays.**

GRAPHICS: Style

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KEY**

Specifies the cutting plane:

**0**

Cutting plane is normal to the viewing vector [**/VIEW**] and passes through the focus point [**/FOCUS**] (default).

**1**

The working plane [**WPLANE**] is the cutting plane.

## Command Default

The cutting plane is normal to the viewing vector at the focus point.

## Notes

Defines the cutting plane to be used for section and capped displays [**/TYPE**,,(1, 5, or 7)].

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

---

**CPLGEN**, *NSETF*, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*

**Generates sets of coupled nodes from an existing set.**

PREP7: Coupled DOF

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NSETF***

Generate sets from existing set *NSETF*.

### ***Lab1*, *Lab2*, *Lab3*, ..., *Lab5***

Generate sets with these labels (see **CP** command for valid labels). Sets are numbered as the highest existing set number + 1.

## Notes

Generates additional sets of coupled nodes (with different labels) from an existing set [**CP**, **CPNGEN**]. The same node numbers are included in the generated sets. If all labels of nodes are to be coupled and the nodes are coincident, the **NUMMRG** command should be used to automatically redefine the node number (for efficiency).

## Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Gen w/Same Nodes

---

**CPLIST**, *NSET1*, *NSET2*, *NINC*, *Nsel*

**Lists the coupled degree of freedom sets.**

PREP7: Coupled DOF

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NSET1*, *NSET2*, *NINC***

List coupled sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL (default), *NSET2* and *NINC* are ignored and all coupled sets are listed.

***Nsel***

Node selection control:

**ANY**

List coupled set if *any* of the selected nodes are in the set (default).

**ALL**

List coupled set only if *all* of the selected nodes are in the set.

**Notes**

This command is valid in any processor.

**Menu Paths**

**Utility Menu>List>Other>Coupled Sets>All CP nodes selected**

**Utility Menu>List>Other>Coupled Sets>Any CP node selected**

**CPMERGE, Lab**

**Merges different couple sets with duplicate degrees of freedom into one couple set.**

[PREP7: Coupled DOF](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Lab***

Degree of freedom label for coupled nodes (in the nodal coordinate system). Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations) (in radians). Thermal labels: TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature). Fluid labels: PRES (pressure); VX, VY, or VZ (velocities). Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials); CURR (current). Explicit analysis labels: UX, UY, or UZ (displacements). The degree of freedom set is determined from all element types defined and the **DOF** command, if used.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Capacitor**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Curr Cntl CS**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Curr Cntl VS**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Diode**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Constant Amplitude**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Exponential**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Piecewise**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Pulse**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Sinusoidal**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Constant Amplitude**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Exponential**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Piecewise**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Pulse**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Sinusoidal**

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Inductor  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Mass Cond 2D  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Mass Cond 3D  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Mutual Ind  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Resistor  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Strnd Coil  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Vltg Cntl CS  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Vltg Cntl VS  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Wire  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Zener Diode  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Linear Rotary  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Linear Trans  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Nonlin Rotary  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Nonlin Trans  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Slide Film  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Mass  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Linear Rotary  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Linear Trans  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Rotary  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Trans  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Capacitor  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Constant  
 Amplitude  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Exponential  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Piecewise  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Pulse  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Sinusoidal  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Constant  
 Amplitude  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Exponential  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Piecewise  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Pulse  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Sinusoidal  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Inductor  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Resistor  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Wire  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Transducer>ElecMech  
 Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

## CPNGEN, NSET, Lab, NODE1, NODE2, NINC

Defines, modifies, or adds to a set of coupled degrees of freedom.

PREP7: Coupled DOF

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NSET

Set reference number [CP].

### Lab

Degree of freedom label [CP].



***NODE1, NODE2, NINC***

Include in coupled set nodes *NODE1* to *NODE2* in steps of *NINC* (defaults to 1). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *-NODE1*, delete range of nodes from set instead of including. A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

**Notes**

Defines, modifies, or adds to a set of coupled degrees of freedom. May be used in combination with (or in place of) the **CP** command. Repeat **CPNGEN** command for additional nodes.

**Menu Paths**

**This command cannot be accessed from a menu.**

***CPSGEN, ITIME, INC, NSET1, NSET2, NINC***

**Generates sets of coupled nodes from existing sets.**

[PREP7: Coupled DOF](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***ITIME, INC***

Do this generation operation a total of *ITIMES*, incrementing all nodes in the existing sets by *INC* each time after the first. *ITIME* must be > 1 for generation to occur.

***NSET1, NSET2, NINC***

Generate sets from sets beginning with *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* is negative, *NSET2* and *NINC* are ignored and the last  $|NSET1|$  sets (in sequence from the maximum set number) are used as the sets to be repeated.

**Notes**

Generates additional sets of coupled nodes (with the same labels) from existing sets. Node numbers between sets may be uniformly incremented.

**Menu Paths**

**Main Menu>Preprocessor>Coupling / Ceqn>Gen w/Same DOF**

***CQC, SIGNIF, Label***

**Specifies the complete quadratic mode combination method.**

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***SIGNIF***

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multipoint, or DDAM response (**SPOPT**, **SPRS**, **MPRS** or **DDAM**), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode

## \*CREATE

---

whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**,PSD.)

### **Label**

Label identifying the combined mode solution output.

### **DISP**

Displacement solution (default). Displacements, stresses, forces, etc., are available.

### **VELO**

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

### **ACEL**

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

## Notes

Damping is required for this mode combination method. The **CQC** command is also valid for PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine**

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine**

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine**

**Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine**

**Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine**

**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine**

---

## \*CREATE, *Fname*, *Ext*, --

Opens (creates) a macro file.

[APDL: Macro Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Do not use a directory path if file is to be read with the macro *Name* option of the **\*USE** command.

### ***Ext***

Filename extension (8 character maximum).

*Ext* should not be used if file is to be read with the macro *Name* option of the **\*USE** command.

### **--**

Unused field.

## Notes

See the **\*USE** command for a discussion of macros. All commands following the **\*CREATE** command, up to the **\*END** command, are written to the specified file without being executed. An existing file of the same name, if any, will be overwritten. Parameter values are not substituted for parameter names in the commands when the commands are written to the file. Use **\*CFWRITE** to create a file if this is desired. The resulting macro may be executed with a **\*USE** command (which also allows parameters to be passed into the macro) or a **/INPUT** command (which does not allow parameters to be passed in). Several macros may be stacked into a library file [**\*ULIB**]. You cannot use **\*CREATE** within a DO loop.

This command is valid in any processor.

## Menu Paths

Utility Menu>Macro>Create Macro

### CRPLIM, *CRCR*, *Option*

Specifies the creep criterion for automatic time stepping.

[SOLUTION: Nonlinear Options](#)

MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS

#### *CRCR*

Value of creep criteria for the creep limit ratio control.

#### *Option*

Type of creep analysis:

##### **1 or ON**

Implicit creep analysis.

##### **0 or OFF**

Explicit creep analysis.

## Notes

The creep ratio control can be used at the same time for implicit creep and explicit creep analyses. For implicit creep (*Option* = 1), the default value of *CRCR* is zero (i.e., no creep limit control), and you are allowed to specify any value. For explicit creep (*Option* = 0), the default value of *CRCR* is 0.1. The maximum value allowed is 0.25. This command is also valid in PREP7. The **CUTCONTROL** command can be used for the same purpose and is the preferred command, when **SOLCONTROL** is ON.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Creep Criterion

Main Menu>Solution>Load Step Opts>Nonlinear>Creep Criterion

**CS**, *KCN*, *KCS*, *NORIG*, *NXAX*, *NXYPL*, *PAR1*, *PAR2*

**Defines a local coordinate system by three node locations.**

DATABASE: Coordinate System

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### **KCN**

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

#### **KCS**

Coordinate system type:

##### **0 or CART**

Cartesian

##### **1 or CYLIN**

Cylindrical (circular or elliptical)

##### **2 or SPHE**

Spherical (or spheroidal)

##### **3 or TORO**

Toroidal

#### **NORIG**

Node defining the origin of this coordinate system. If *NORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

#### **NXAX**

Node defining the positive x-axis orientation of this coordinate system.

#### **NXYPL**

Node defining the x-y plane (with *NORIG* and *NXAX*) in the first or second quadrant of this coordinate system.

#### **PAR1**

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

#### **PAR2**

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

## Notes

Defines and activates a local right-handed coordinate system by specifying three existing nodes: to locate the origin, to locate the positive x-axis, and to define the positive x-y plane. This local system becomes the active coordinate system. See the **CLOCAL**, **CSKP**, **CSWPLA**, and **LOCAL** commands for alternate definitions. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

## Menu Paths

**Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>By 3 Nodes**

**CSCIR, KCN, KTHET, KPFI****Locates the singularity for non-Cartesian local coordinate systems.**

DATABASE: Coordinate System

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**KCN**

Number of the local coordinate system in which singularity location is to be changed. Must be greater than 10.

**KTHET**

Theta singularity location for cylindrical, spherical, and toroidal systems:

**0**Singularity at  $\pm 180^\circ$ .**1**Singularity at  $0^\circ$  ( $360^\circ$ ).**KPFI**

Phi singularity location for toroidal systems:

**0**Singularity in phi direction at  $\pm 180^\circ$ .**1**Singularity in phi direction at  $0^\circ$  ( $360^\circ$ ).**Command Default**Singularities at  $\pm 180^\circ$ .**Notes**

Continuous closed surfaces (circles, cylinders, spheres, etc.) have a singularity (discontinuity) at  $\theta = \pm 180^\circ$ . For local cylindrical, spherical, and toroidal coordinate systems, this singularity location may be changed to  $0^\circ$  ( $360^\circ$ ).

An additional, similar singularity occurs in the toroidal coordinate system at  $\Phi = \pm 180^\circ$  and can be moved with *KPFI*. Additional singularities occur in the spherical coordinate system at  $\Phi = \pm 90^\circ$ , but cannot be moved.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>WorkPlane>Local Coordinate Systems>Move Singularity**

**CSDELE**, *KCN1*, *KCN2*, *KCINC*

**Deletes local coordinate systems.**

DATABASE: Coordinate System

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***KCN1***, ***KCN2***, ***KCINC***

Delete coordinate systems from *KCN1* (must be greater than 10) to *KCN2* (defaults to *KCN1*) in steps of *KCINC* (defaults to 1). If *KCN1* = ALL, *KCN2* and *KCINC* are ignored and all coordinate systems are deleted.

**Notes**

This command is valid in any processor.

**Menu Paths**

**Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node  
Utility Menu>WorkPlane>Local Coordinate Systems>Delete Local CS**

**CSKP**, *KCN*, *KCS*, *PORIG*, *PXAXS*, *PXYPL*, *PAR1*, *PAR2*

**Defines a local coordinate system by three keypoint locations.**

DATABASE: Coordinate System

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***KCN***

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

***KCS***

Coordinate system type:

**0 or CART**

Cartesian

**1 or CYLIN**

Cylindrical (circular or elliptical)

**2 or SPHE**

Spherical (or spheroidal)

**3 or TORO**

Toroidal

***PORIG***

Keypoint defining the origin of this coordinate system. If *PORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

***PXAXS***

Keypoint defining the positive x-axis orientation of this coordinate system.

**PXYPL**

Keypoint defining the x-y plane (with *PORIG* and *PXAXS*) in the first or second quadrant of this coordinate system.

**PAR1**

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

**PAR2**

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

## Notes

Defines and activates a local right-handed coordinate system by specifying three existing keypoints: to locate the origin, to locate the positive x-axis, and to define the positive x-y plane. This local system becomes the active coordinate system. See the **CLOCAL**, **CS**, **CSWPLA**, and **LOCAL** commands for alternate definitions. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

## Menu Paths

**Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>By 3 Keypoints**

## CSLIST, *KCN1*, *KCN2*, *KCINC*

**Lists coordinate systems.**

DATABASE: Coordinate System

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***KCN1*, *KCN2*, *KCINC***

List coordinate systems from *KCN1* to *KCN2* (defaults to *KCN1*) in steps of *KCINC* (defaults to 1). If *KCN1* = ALL (default), *KCN2* and *KCINC* are ignored and all coordinate systems are listed.

## Notes

This command is valid in any processor.

## Menu Paths

**Utility Menu>List>Other>Local Coord Sys**

**CSWPLA**, *KCN*, *KCS*, *PAR1*, *PAR2*

**Defines a local coordinate system at the origin of the working plane.**

DATABASE: Coordinate System

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***KCN***

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

***KCS***

Coordinate system type:

**0 or CART**

Cartesian

**1 or CYLIN**

Cylindrical (circular or elliptical)

**2 or SPHE**

Spherical (or spheroidal)

**3 or TORO**

Toroidal

***PAR1***

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

***PAR2***

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

## Notes

Defines and activates a local right-handed coordinate system centered at the origin of the working plane. The coordinate system's local x-y plane (for a Cartesian system) or R- $\theta$  plane (for a cylindrical or spherical system) corresponds to the working plane. This local system becomes the active coordinate system. See the **CS**, **LOCAL**, **CLOCAL**, and **CSKP** commands for alternate ways to define a local coordinate system. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

## Menu Paths

**Main Menu>General Postproc>Path Operations>Define Path>On Working Plane**

**Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane**

**Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>At WP Origin**



**CSYS**, *KCN*

**Activates a previously defined coordinate system.**

DATABASE: Coordinate System

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*KCN*

Specifies the active coordinate system, as follows:

**0 (default)**

Cartesian

**1**

Cylindrical with global Cartesian Z as the axis of rotation

**2**

Spherical

**4 or WP**

Working Plane

**5**

Cylindrical with global Cartesian Y as the axis of rotation

**11 or greater**

Any previously defined local coordinate system

## Notes

The **CSYS** command activates a previously defined coordinate system for geometry input and generation. The **LOCAL**, **CLOCAL**, **CS**, **CSKP**, and **CSWPLA** commands also activate coordinate systems as they are defined.

The active coordinate system for files created via the **CDWRITE** command is Cartesian (**CSYS,0**).

This command is valid in any processor.

**CSYS,4** (or **CSYS,WP**) activates working plane tracking, which updates the coordinate system to follow working plane changes. To deactivate working plane tracking, activate *any other* coordinate system (for example, **CSYS,0** or **CSYS,11**).

**CSYS,5** is a cylindrical coordinate system with global Cartesian Y as the axis. The local x, y and z axes are radial,  $\theta$ , and axial (respectively). The R-Theta plane is the global X-Z plane, as it is for an axisymmetric model. Thus, at  $\theta = 0.0$ , **CSYS,5** has a specific orientation: the local x is in the global +X direction, local y is in the global -Z direction, and local z (the cylindrical axis) is in the global +Y direction.

## Menu Paths

**Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node**

**Utility Menu>WorkPlane>Change Active CS to>Global Cartesian**

**Utility Menu>WorkPlane>Change Active CS to>Global Cylindrical**

**Utility Menu>WorkPlane>Change Active CS to>Global Spherical**

**Utility Menu>WorkPlane>Change Active CS to>Specified Coord Sys**

**Utility Menu>WorkPlane>Change Active CS to>Working Plane**

**Utility Menu>WorkPlane>Offset WP to>Global Origin**

**/CTYPE**, *KEY*, *DOTD*, *DOTS*, *DSHP*, *TLEN*

**Specifies the type of contour display.**

GRAPHICS:Style

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KEY**

Type of display:

- 0** Standard contour display.
- 1** Isosurface display.
- 2** Particle gradient display.
- 3** Gradient triad display.

**DOTD**

Maximum dot density for particle gradient display (*KEY* = 2). Density is expressed as dots per screen width (defaults to 30).

**DOTS**

Dot size for particle gradient display (*KEY* = 2). Size is expressed as a fraction of the screen width (defaults to 0.0 (single dot width)).

**DSHP**

Spherical dot shape precision for particle gradient display (*KEY* = 2). (3-D options are supported only on 3-D devices):

- 0** Flat 2-D circular dot.
- 1** Flat-sided 3-D polyhedron.
- n** 3-D sphere with *n* (>1) polygon divisions per 90° of radius.

**TLEN**

Maximum length of triads for gradient triad display (*KEY* = 3). Value is expressed as a fraction of the screen width (defaults to 0.067).

**Command Default**

Standard contour display.

**Notes**

Use **/CTYPE,STAT** to display the current settings. Only the standard contour display [**/CTYPE,0**] and the isosurface contour display [**/CTYPE,1**] are supported by PowerGraphics [**/GRAPHICS,POWER**].

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Contour Style

### CURR2D

Calculates current flow in a 2-D conductor.

POST1: Magnetics Calculations

MP ME <> <> <> <> <> EM <> <> PP <> EME MFS

### Notes

**CURR2D** invokes an ANSYS macro which calculates the total current flowing in a conducting body for a 2-D planar or axisymmetric magnetic field analysis. The currents may be applied source currents or induced currents (eddy currents). The elements of the conducting region must be selected before this command is issued. The total current calculated by the macro is stored in the parameter TCURR. Also, the total current and total current density are stored on a per-element basis in the element table [**ETABLE**] with the labels TCURR and JT, respectively. Use the **PLETAB** and **PRETAB** commands to plot and list the element table items.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Current

**CUTCONTROL**, *Lab*, *VALUE*, *Option*

Controls time-step cutback during a nonlinear solution.

SOLUTION: Analysis Options

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### *Lab*

Specifies the criteria for causing a cutback. Possible arguments are

#### **PLSLIMIT**

Maximum equivalent plastic strain allowed within a time-step (substep). If the calculated value exceeds the *VALUE*, the program performs a cutback (bisection). *VALUE* defaults to 0.15 (15%).

#### **CRPLIMIT**

Maximum equivalent creep ratio allowed within a time step (substep). If the calculated value exceeds *VALUE*, the program performs a cutback (bisection). The default depends on whether you are performing an implicit or explicit creep analysis (see *Option* argument below). The creep ratio control can be used at the same time for implicit creep and explicit creep analyses. For an implicit creep analysis, *VALUE* defaults to zero (i.e., no creep limit control), and you are allowed to enter any value. For an explicit creep analysis, *VALUE* defaults to 0.1 (10%) and the maximum value allowed is 0.25.

**DSPLIMIT**

Maximum incremental displacement within the solution field in a time step (substep). If the maximum calculated value exceeds *VALUE*, the program performs a cutback (bisection). *VALUE* defaults to  $1.0 \times 10^7$ .

**NPOINT**

Number of points in a cycle for a second order dynamic equation, used to control automatic time stepping. If the number of solution points per cycle is less than *VALUE*, the program performs a cutback in time step size. *VALUE* defaults to 13.

This option works well for linear problems. For nonlinear analyses, other factors such as contact status changes and solution convergence rate can overwrite NPOINT. See [Automatic Time Stepping](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on automatic time stepping.

**NOITERPREDICT**

If *VALUE* is 0 (default), an internal auto time step scheme will predict the number of iterations for nonlinear convergence and perform a cutback earlier than the number of iterations specified by the **NEQIT** command. This is the recommended option. If *VALUE* is 1, the solution will iterate (if nonconvergent) to **NEQIT** number of iterations before a cutback is invoked. It is sometimes useful for poorly convergent problems, but rarely needed in general.

Bisection is also controlled by contact status change, plasticity or creep strain limit, and other factors. If any of these factors occur, bisection will still take place, regardless of the NOITERPREDICT setting.

**CUTBACKFACTOR**

Changes the cutback value for bisection. Default is 0.5. *VALUE* must be greater than 0.0 and less than 1.0. This option is active only if **AUTOTS,ON** is set.

**VALUE**

Numeric value for the specified cutback criterion. For *Lab* = CRPLIMIT, *VALUE* is the creep criteria for the creep limit ratio control.

**Option**

Type of creep analysis. *Option* is valid for *Lab* = CRPLIMIT only.

**1 or ON**

Implicit creep analysis.

**0 or OFF**

Explicit creep analysis.

**Notes**

The default values given for this command assume **SOLCONTROL,ON** (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL,ON** and **SOLCONTROL,OFF**.

A cutback is a method for automatically reducing the step size when either the solution error is too large or the solution encounters convergence difficulties during a nonlinear analysis. Should a convergence failure occur, the program reduces the time step interval to a fraction of its previous size and automatically continue the solution from the last successfully converged time step. If the reduced time step again fails to converge, the program again reduces the time step size and proceed with the solution. This process continues until convergence is achieved or the minimum specified time step value is reached.

The **CRPLIM** command is functionally equivalent to *Lab* = CRPLIMIT.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Cutback Control**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear**  
**Main Menu>Solution>Load Step Opts>Nonlinear>Cutback Control**

---

**/CVAL**, *WN*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*, *V7*, *V8*

Specifies nonuniform contour values on stress displays.

GRAPHICS: Labeling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*WN*

Window number (or ALL) to which command applies (defaults to 1).

*V1*, *V2*, *V3*, ..., *V8*

Up to 8 contour values may be specified (in ascending order). The 0.0 value (if any) must not be the last value specified. If no values are specified, all contour specifications are erased and contours are automatically calculated.

### Command Default

Nine contour values uniformly spaced between the extreme values.

### Notes

This command is similar to the **/CONTOUR** command. With **/CVAL**, however, you define the upper level of each contour band instead of having the contours uniformly distributed over the range. The minimum value (including a zero value for the first band) for a contour band cannot be specified. If you use both **/CONTOUR** and **/CVAL**, the last command issued takes precedence.

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Style>Contours>Non-uniform Contours**

---

**CVAR**, *IR*, *IA*, *IB*, *ITYPE*, *DATUM*, *Name*

Computes covariance between two quantities.

POST26: Special Purpose

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

*IR*

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previous variable, the previous variable will be overwritten with this result.

**IA, IB**

Reference numbers of the two variables to be operated on. If only one, leave *IB* blank.

**ITYPE**

Defines the type of response PSD to be calculated:

**0,1**

Displacement (default).

**2**

Velocity.

**3**

Acceleration.

**DATUM**

Defines the reference with respect to which covariance is to be calculated:

**1**

Absolute value.

**2**

Relative to base (default).

**Name**

Thirty-two character name for identifying the variable on listings and displays. Embedded blanks are compressed upon output.

## Notes

This command computes the covariance value for the variables referenced by the reference numbers *IA* and *IB*. If *DATUM* = 2, the variable referenced by *IR* will contain the individual modal contributions (i.e., the dynamic or relative values). If *DATUM* = 1, the variable referenced by *IR* will contain the modal contributions followed by the contributions of pseudo-static and covariance between dynamic and pseudo-static responses. *File.PSD* must be available for the calculations to occur.

## Menu Paths

**Main Menu>TimeHist Postpro>Calc Covariance**

---

## **/CWD**, *DIRPATH*

**Changes the current working directory.**

[SESSION: Run Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***DIRPATH***

The full path name of the new working directory.

## Notes

After issuing the */CWD* command, all new files opened with no default directory specified (via the **FILE**, **/COPY**, or **RESUME** commands, for example) default to the new *DIRPATH* directory.

If issuing the command interactively and the specified directory does not exist, no change of directory occurs and the command generates an error message. If the command executes via a batch run and the specified directory does not exist, the batch program terminates with an error.

## Menu Paths

Utility Menu>File>Change Directory

**/CYCEXPAND**, *WN*, *OPTION*, *Value1*, *Value2*

**Graphically expands displacements, stresses and strains of a cyclically symmetric model.**

PREP7: Special Purpose

POST1: Special Purpose

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### **WN**

The window number to which the expansion applies. Valid values are 1 through 5. The default value is 1. The window number applies only to the AMOUNT argument.

### **OPTION**

One of the following options:

#### **ON**

Activates cyclic expansion using the previous settings (if any). If no previous settings exist, this option activates the default settings.

#### **DEFAULT**

Resets cyclic expansion to the default settings.

#### **OFF**

Deactivates cyclic expansion. This option is the default.

#### **STATUS**

Lists the current cyclic expansion settings.

#### **AMOUNT**

The number of repetitions *or* the total angle.

##### **Value1**

NREPEAT

##### **Value2**

The number of repetitions. The default is the total number of sectors in 360 degrees.

*or*

##### **Value1**

ANGLE

##### **Value2**

The total angle in degrees. The default is 360.

### **WHAT**

A specified portion or subset of the model to expand:

**Value1**

The component name of the elements to expand. The default is all selected components.

**EDGE**

Sector edge display key.

**1**

Suppresses display of edges between sectors even if the cyclic count varies between active windows.

---

**Caution**

Plots with fewer than the maximum number of repetitions may have missing element faces at the sector boundaries.

**0 or OFF**

Averages stresses or strains across sector boundaries. This value is the default (although the default reverts to 1 or ON if the cyclic count varies between active windows).

**1 or ON**

No averaging of stresses or strains occurs and sector boundaries are shown on the plot.

**PHASEANG**

The phase angle shift:

**Value1**

The phase angle shift in degrees. The valid range is 0 through 360. The default is 0. Typically, this value is the phase angle obtained via the **CYCPHASE** command.

**Command Default**

The default **/CYCEXPAND** command (issuing the command with no arguments) deactivates cyclic expansion (*OPTION* = OFF). The default window number (*WN*) is 1.

**Notes**

In preprocessing, the **/CYCEXPAND** command verifies a cyclically symmetric model by graphically expanding it partially or through the full 360 degrees.

For the postprocessing plot nodal solution (**PLNSOL**) operation, the command graphically expands displacements, stresses and strains of a cyclically symmetric model partially or through the full 360 degrees by combining the *real* (original nodes and elements) and *imaginary* (duplicate nodes and elements) parts of the solution.

For the print nodal solution (**PRNSOL**) operation, the command expands the printed output of displacements or stresses on a sector-by-sector basis.

Use of the **/CYCEXPAND** command does *not* change the database. The command does not modify the geometry, nodal displacements or element stresses.

The command affects element and result plots only. It has no effect on operations other than plot element solution (**PLESOL**), plot nodal solution (**PLNSOL**) and print nodal solution (**PRNSOL**). Operations other than **PLESOL**, **PLNSOL**, or **PRNSOL** work on the *unprocessed* real and imaginary parts of a cyclic symmetry solution



If you issue a **/CYCEXPAND,,OFF** command, you cannot then expand the model by simply issuing another **/CYCEXPAND** command (for example, to specify an **NREPEAT** value for the number of repetitions). In such a case, you must specify **/CYCEXPAND,,ON**, which activates expansion using the previous settings (if any) or the default settings.

The command requires PowerGraphics. Turn PowerGraphics on (**/GRAPHICS,POWER**) if not already active. (By default, PowerGraphics is active during an interactive session, but not during a batch run.) Any setting which bypasses PowerGraphics (for example, **/PBF**) also bypasses cyclic expansion; in such cases, the **/CYCEXPAND** command displays unprocessed real and imaginary results.

The **CYCPHASE** command uses full model graphics (**/GRAPHICS,FULL**) to compute peak values. Because of this, there may be slight differences between max/min values obtained with **CYCPHASE**, and those obtained via **/CYCEXPAND**, which uses power graphics (**/GRAPHICS,POWER**).

The **/CYCEXPAND** command does not work with PGR files.

The **/CYCEXPAND** command is incompatible with the **/ESHAPE,1** or **/ESHAPE,FAC** command.

The **/PBC** (plot boundary conditions) command is deactivated when cyclic expansion is active (**/CYCEXPAND,,ON**). To view BCs on the basic sector, deactivate cyclic expansion (**/CYCEXPAND,,OFF**) and issue this command: **/PBC,ALL,,1**

In cases involving **non-cyclically symmetric loading**, all boundary conditions (BCs) will plot when **/CYCEXPAND** is active. When examining the plot, however, it is not possible to distinguish between BCs that exist on every sector and those that exist (via a table referencing **SECTOR**) only on some sectors or which vary in value. (Only the **basic sector** is plotted.)

For **magnetic cyclic symmetry** analyses, the **/CYCEXPAND** command produces contour plots but not vector plots.

To learn more about analyzing a cyclically symmetric structure, see [Cyclic Symmetry Analysis](#) in the *Advanced Analysis Techniques Guide*.

## Menu Paths

**Main Menu>General Postproc>Cyclic Analysis>Cyc Expansion**

**Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyc Expansion**

## \*CYCLE

**Bypasses commands within a do-loop.**

APDL: Process Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Bypasses all commands between this command and the **\*ENDDO** command within a do-loop. The next loop (if applicable) is initiated. The cycle option may also be conditionally executed [Use the **\*IF**]. The **\*CYCLE** command must appear on the same file as the **\*DO** command.

This command is valid in any processor.

## Menu Paths

This command cannot be accessed from a menu.

**CYCLIC**, *NSECTOR*, *ANGLE*, *KCN*, *Name*, *USRCOMP*, *USRNMAP*

Specifies a cyclic symmetry analysis.

PREP7: Special Purpose

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### ***NSECTOR***

The number of sectors in the full 360 degrees, or one of the following options:

#### **STATUS**

Indicates the current cyclic status.

#### **OFF**

Resets model to normal (non-cyclic) status and removes the [duplicate sector](#) if it exists. This option also deletes automatically detected edge components (generated when *USRCOMP* = 0).

#### **UNDOUBLE**

Removes the [duplicate sector](#) if it exists. The duplicate sector is created during the solution (**SOLVE**) stage of a modal cyclic symmetry analysis.

---

### **Note**

The duplicate sector is necessary for displaying cyclic symmetry analysis results during postprocessing (**/POST1**).

If you specify a value of STATUS, OFF or UNDOUBLE, the command ignores all remaining arguments.

### ***ANGLE***

The sector angle in degrees.

### ***KCN***

An arbitrary reference number assigned to the cyclic coordinate system. The default value of 0 specifies automatic detection.

### ***Name***

The root name of sector low- and high-edge components (line, area, or node components). The default root name (when *USRCOMP* = 0) is "CYCLIC". A root name that you specify can contain up to 11 characters.

The naming convention for each low- and high-edge component pair is either of the following:

*Name\_mxxl*, *Name\_mxxh* (potentially matched node patterns)  
*Name\_uxxl*, *Name\_uxxh* (potentially unmatched node patterns)

The *Name* value is the default ("CYCLIC") or specified root name and *xx* is the component pair ID number (sequential, starting at 01).

### ***USRCOMP***

The number of pairs of user-defined low- and high-edge components on the cyclic sector (if any). The default value of 0 specifies automatic detection of sector edges; however, the automatic setting is not

valid in all cases. (For more information, see the Notes section below.) If the value is greater than 0, no verification of user-defined components occurs.

### **USRNMAP**

The name of a user-defined array specifying the matching node pairs between the sector low and high edges. Valid only when *USRCOMP*=0. Skips the automatic detection of sector edges. Node pairs may be input in any order, but the low edge node must be the first entry in each pair.

```
*DIM,MYMAP,ARRAY,2,14 ! specifying 14 low-high edge node pairs
*set,mymap(1, 1), 107, 108 ! low node 107 <> high node 108
*set,mymap(1, 2), 147, 211 ! low node 147 <> high node 211
*set,mymap(1, 3), 110, 109 ! low node 110 <> high node 109
      ! etc for node pairs 4 through 14...
cyclic,12,,1,,MYMAP ! use array MYMAP to generate cyclic CE's
```

## **Command Default**

The default **CYCLIC** command (issuing the command with no arguments) detects the number of sectors (*NSECTOR*), the sector angle (*ANGLE*), and the coordinate system (*KCN*) based upon the existing solid or finite-element model. The command also detects sector low- and high-edge components in most cases and assigns the default root name "CYCLIC" to the components.

## **Notes**

You can input your own value for *NSECTOR*, *ANGLE* or *KCN*; if you do so, the command verifies argument values before executing.

When *USRCOMP* = 0 and *USRNMAP* = blank (default), the **CYCLIC** command automatically detects **low- and high-edge components** for models comprised of any combination of line, area, or volume elements. If a solid model exists, however, the command uses only the lines, areas, and/or volumes to determine the low- and high-edge components; the elements, if any, are ignored.

If you issue a **CYCOPT**,*TOLER* command to set a tolerance for edge-component pairing *before* issuing the **CYCLIC** command, the **CYCLIC** command uses the specified tolerance when performing automatic edge-component detection.

For 2-D models, autodetection does not consider the **CSYS**,5 or **CSYS**,6 coordinate system specification. Autodetection for 180 degree (two-sector) models is not possible unless a central hole exists.

The **CYCLIC** command sets values and keys so that, if possible, the area-mesh (**AMESH**) or volume-mesh (**VMESH**) command meshes the sector with matching node and element face patterns on the low and high edges. (The command has no effect on any other element-creation command.)

Issue the **CYCLIC** command *prior* to the meshing command to, if possible, produce a mesh with identical node and element patterns on the low and high sector edges. Only the **AMESH** or **VMESH** commands can perform automated matching. (Other meshing operation commands such as **VSWEEP** cannot.) If you employ a meshing operation other than **AMESH** or **VMESH**, you should ensure that node and element face patterns match, if desired. The **CYCLIC** command output indicates whether each edge-component pair has or can produce a matching node pair.

A cyclic solution (via the **SOLVE** command) allows dissimilar mesh patterns on the extreme boundaries of a cyclically symmetric model. The allowance for dissimilar patterns is useful when you have only finite-element meshes for your model but not the geometry data necessary to remesh it to obtain identical node patterns. In such cases, it is possible to obtain solution results, although perhaps at the expense of accuracy. A warning message appears because results may be degraded near the sector edges.

The constraint equations (CEs) that tie together the low and high edges of your model are generated at the solution stage of the analysis from the low- and high-edge components (and nowhere else). You should verify that automatically detected components are in the correct locations and that you can account for all components; to do so, you can list (**CMLIST**) or plot (**CMPLLOT**) the components.

If you issue the **CYCLIC** command after meshing and have defined element types with rotational degrees of freedom (DOFs), ANSYS generates cyclic CEs for rotational DOFs that may not exist on the sector boundaries. Issue the **CYCOPT,DOF** command to prevent unused rotational terms from being generated.

Modal cyclic symmetry analysis is supported by the following eigensolvers:

- Block Lanczos (**MODOPT**, LANB)
- PCG Lanczos (**MODOPT**, LANPCG)
- Super Node (**MODOPT**, SNODE)

To learn more about analyzing a cyclically symmetric structure, see [Cyclic Symmetry Analysis](#) in the *Advanced Analysis Techniques Guide*.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Model>Auto Defined**

**Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Model>Status**

**Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Model>User Defined**

**Main Menu>Preprocessor>Modeling>Cyclic Sector>Del Dupl Sector**

**Main Menu>Preprocessor>Modeling>Cyclic Sector>Reset (OFF)**

---

**CYCOPT**, *OPTION*, *Value1*, *Value2*, *Value3*, *Value4*, *Value5*, *Value6*, *Value7*

**Specifies solution options for a cyclic symmetry analysis.**

PREP7: Special Purpose  
SOLUTION: Dynamic Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### **OPTION**

One of the following options:

#### **BCMULT**

Controls whether cyclic sector array parameter names are reused or created new for multiple entities.

#### **Value1**

To reuse array parameter names, specify YES, ON, or 1.

To create new array parameter names (the default behavior), specify NO, OFF, or 0.

#### **COMBINE**

For linear [static cyclic symmetry analysis](#) with [non-cyclically symmetric loading](#) only, expands and combines all harmonic index solutions and writes them to the results file during the solution phase of the analysis.

#### **Value1**

To enable combining of harmonic index solutions, specify YES, ON, or 1.

To disable combining (the default behavior), specify NO, OFF, or 0.

#### DEFAULT

Sets the default cyclic solution settings.

#### DOF

The degrees of freedom to couple from the nodes on the low sector boundary to nodes on the high boundary:

##### *Value1*

The component pair ID number.

##### *Value2, Value3, Value4, ..., Value7*

The constraint-equation/-coupling degree of freedom (DOF) for this pair. Repeat the command to add other DOFs. The default is constraint-equation/-coupling *all* applicable DOFs.

#### FACETOL

Tolerance for inclusion of surface nodes into your basic sector. Autodetect defaults to 15°, accommodating most sections. Specify a new *Value1* only when extreme cut angles or complex model geometry cause surface nodes to be excluded. See Notes (below) for more information.

ANSYS recommends that successful auto-detection depends more on the value of ANGTOL than the value of FACETOL. Please refer to [CYCOPT Auto Detection Tolerance Adjustments for Difficult Cases](#) for more information about auto-detection and the **CYCOPT** command.

##### *Value1*

The face tolerance applies only to auto detection from node/element models (already meshed and no solid model), and it defaults to 15°.

#### HINDEX

The [harmonic index](#) solution ranges for [modal](#) or [buckling](#) cyclic symmetry analyses. The **SOLVE** command initiates a cyclic symmetry solution sequence at the harmonic indices specified. (By default, the **SOLVE** command solves for *all* available harmonic indices.) [Static](#) and [harmonic](#) cyclic symmetry solutions always use all harmonic indices required for the applied loads.

#### EVEN / ODD

For low-frequency [electromagnetic analysis](#) only, EVEN specifies a symmetric solution and ODD specifies an antisymmetric solution.

The value you specify is based on the [harmonic index](#): EVEN (default) indicates harmonic index = 0, and ODD indicates harmonic index =  $N / 2$  (where  $N$  is an integer representing the number of sectors in 360°). A value of ODD applies only when  $N$  is an even number.

The **CYCOPT** command with this HINDEX option is cumulative. To remove an option (for example, EVEN), issue this command: **CYCOPT,HINDEX,EVEN,,,1**

#### ALL

Solve all applicable harmonic indices.

---

#### Note

Value2 must be blank.

##### *Value1, Value2, Value3*

Solve harmonic indices in range *Value1* through *Value2* in steps of *Value3*. Repeat the command to add other ranges. The default solves *all* applicable harmonic indices.

**Value4**

The only valid value is -1. If specified, it removes *Value1* through *Value2* in steps of *Value3* from the set to solve.

**LDSECT**

Restricts subsequently defined force loads and surface loads to a specified sector. The restriction remains in effect until you change or reset it.

**Value1**

The sector number. A value other than 0 (default) is valid for a cyclic symmetry analysis with [non-cyclically symmetric loading](#) only. A value of 0 (or ALL) resets the default behavior for cyclic loading (where the loads are identical on all sectors).

**MOVE**

Flag specifying whether ANSYS should move high- or low-edge component nodes paired within the specified tolerance (TOLER) to create *precisely matching* pairs.

**Value1**

The flag value, as follows:

0 -- Do not move edge component nodes. This value is the default.

1 or HIGH -- Move the high-edge component nodes to precisely match the low-edge component nodes.

-1 or LOW -- Move the low-edge component nodes to precisely match the high-edge component nodes.

**STATUS**

Lists the solution option settings active for the cyclic model.

**TOLER**

The tolerance used to determine whether a node on the low edge is paired with a node on the high edge.

**Value1**

The tolerance value, as follows:

> 0 -- The absolute distance tolerance for automatic sector-boundary detection and low-/high-edge component node pairing.

< 0 -- The relative tolerance for automatic sector-boundary detection and low-/high-edge component node pairing. In this case, the tolerance is  $Value1 * Length$ , where *Length* is the length of the diagonal of an imaginary box enclosing the model.

The default tolerance is -1.0E-4 (also represented by  $Value1 = 0$ ).

**Value2**

*ANGTOL* = Maximum allowable angle tolerance. The default value is 0.01°

The valid range for *ANGTOL* is model dependent.

If you input both the number of sectors and a sector angle, the angle must match  $360/(\text{number of sectors})$  within *ANGTOL*.

If you input only a sector angle, it must divide evenly into 360° within *ANGTOL*.

If you input a sector angle, the final cyclic sector must span that angle within *ANGTOL*.

For auto detected sector angle, the final cyclic sector must span 360/(number of sectors) within *ANGTOL*, everywhere along the LOW/HIGH boundaries.

If *ANGTOL* is too small, your CAD or FEA model may not be accurate enough to allow auto detection or verification.

If *ANGTOL* is too large, you may get an unexpected or incorrect boundary definition, or in other cases fail to detect the boundaries.

For some difficult cases from FEA models (not solid models), you may need to change the value of *FACETOL* to achieve auto detection. Please refer to [CYCOPT Auto Detection Tolerance Adjustments for Difficult Cases](#) for more information about auto-detection and the **CYCOPT** command.

### USRROT

Flag specifying whether ANSYS should override automatic nodal rotations to edge components and allow you to apply nodal rotations manually.

#### Value1

The flag value, as follows:

0 (OFF or NO) -- Allow automatic node rotation. This behavior is the default.

1 (ON or YES) -- Suppress automatic node rotation. If you select this option, you *must* apply appropriate nodal rotations to all edge component nodes; otherwise, your analysis will yield incorrect solution results.

LOW -- Suppresses automatic rotation of low-edge component nodes only, allowing you to apply them manually. (Automatic rotation of high-edge component nodes occurs to produce the matching edge nodes required for a valid cyclic solution.)

HIGH -- Suppresses automatic rotation of high-edge component nodes only, allowing you to apply them manually. (Automatic rotation of low-edge component nodes occurs to produce the matching edge nodes required for a valid cyclic solution.)

### VTSOL

Employ Variational Technology to accelerate your cyclic symmetry analysis.

#### Value1

To employ Variational Technology (VT), specify YES, ON, or 1.

To process without VT (the default behavior), specify NO, OFF, or 0.

## Command Default

No defaults are available for the **CYCOPT** command. You must specify an argument (*OPTION*) when issuing the command. Other values which may be necessary depend upon which argument you specify.

## Notes

ANSYS solves a cyclically symmetric model (set up via the **CYCLIC** command during preprocessing) at the harmonic indices specified via the **CYCOPT** command.



The **CYCOPT,COMBINE** option is an alternative to the **/CYCXPAND** command and is especially useful for testing purposes. However, ANSYS recommends specifying COMBINE only when the number of sectors is relatively small. (The option expands nodes and elements into the full 360° and can slow postprocessing significantly.)

If you issue a **CYCOPT,TOLER** command to set a tolerance for [edge-component pairing](#) before issuing the **CYCLIC** command, the **CYCLIC** command uses the specified tolerance when performing automatic edge-component detection.

In cases involving [non-cyclically symmetric loading](#) (that is, when LDSECT > 0), the underlying command operations create or modify the required SECTOR tabular boundary condition (BC) data to apply on the appropriate sector. Therefore, it is not necessary to manipulate tables for situations where the applied BC is not a function of other tabular BC variables such as TIME, X, Y, Z, and so on.

To delete a previously applied load on a specified sector, issue an **FDELE** command.

Because edge nodes are rotated into the cyclic coordinate system during solution, any applied displacements or forces on sector edges will be in the cyclic coordinate system.

The **CYCOPT** command is valid in the preprocessing and solution stages of an analysis.

To learn more about analyzing a cyclically symmetric structure, see [Cyclic Symmetry Analysis](#) in the *Advanced Analysis Techniques Guide*.

**Distributed ANSYS Restriction** The COMBINE option is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Options**  
**Main Menu>Solution>Solve>Cyclic Options**

---

## CYCPHASE, TYPE, OPTION

**Provides tools for determining minimum and maximum possible result values from frequency couplets produced in a modal cyclic symmetry analysis.**

POST1: Special Purpose

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

### TYPE

The type of operation requested:

#### DISP

Calculate the maximum and minimum possible displacement at each node in the original sector model. Store the values and the phase angle at which they occurred.

#### STRESS

Calculate the maximum and minimum possible stresses at each node in the original sector model. Store the values and the phase angle at which they occurred.

#### STRAIN

Calculate the maximum and minimum possible strains at each node in the original sector model. Store the values and the phase angle at which they occurred.



**ALL**

Calculate the maximum and minimum possible displacement, stress and strain at each node in the original sector model. Store the values and the phase angle at which they occurred.

**GET**

Places the value of a MAX or MIN item into the `_CYCVALUE` parameter, the node for that value in the `_CYCNODE` parameter, and the phase angle for the value in the `_CYCPHASE` parameter.

**PUT**

Put resulting sweep values for printing (via the **PRNSOL** command ) or plotting (via the **PLNSOL** command).

**LIST**

List the current minimum/maximum displacement, stress and strain nodal values.

**STAT**

Summarize the results of the last phase sweep.

**CLEAR**

Clear phase-sweep information from the database.

**OPTION**

If `TYPE = DISP, STRAIN, STRESS` or `ALL`, controls the sweep angle increment to use in the search:

**Angle**

The sweep angle increment in degrees, greater than 0.1 and less than 10. The default is 1.

If `TYPE = PUT`, controls which values are placed onto the model:

**MAX**

Put all existing nodal maximum values onto the model. This option is the default.

**MIN**

Put all existing nodal minimum values onto the model.

If `TYPE = GET`, controls the values placed into cyclic parameters:

**Item**

Specifies the type of values on which to operate:

U -- Displacement  
S -- Stress  
EPEL -- Strain

**Comp**

Specifies the specific component of displacement, stress or strain for which to get information:

X,Y,Z -- Basic components  
XY,YZ,XZ -- Shear components  
1,2,3 -- Principal values  
EQV -- Equivalent value  
SUM -- USUM

**MxMn**

Specifies whether the requested value information is for the maximum or minimum value:

MAX -- Maximum value.  
MIN -- Minimum value.

## Command Default

No defaults are available for the **CYCPHASE** command. You must specify an argument (*TYPE*) when issuing the command. Other values which may be necessary (*OPTION*) depend upon which *TYPE* argument you specify.

## Notes

When you [expand the results of a modal cyclic symmetry analysis](#) (via the **/CYCEXPAND** or **EXPAND** command), ANSYS combines the real and imaginary results for a given nodal diameter, assuming no phase shift between them; however, the modal response can occur at *any* phase shift.

**CYCPHASE** response results are valid only for the first cyclic sector. To obtain the response at *any* part of the expanded model, ANSYS recommends using [cyclic symmetry results expansion](#) at the phase angle obtained via **CYCPHASE**.

The phase angles returned by **CYCPHASE** contain the minimum and maximum values for USUM, SEQV and other scalar principal stress and strain quantities; however, they do not always return the true minimum and maximum values for directional quantities like UX or SX unless the values fall in the first sector.

**CYCPHASE** does not consider midside node values when evaluating maximum and minimum values, which may affect **DISPLAY** quantities but no others. (Typically, ANSYS ignores midside node stresses and strains during postprocessing.)

Issuing **CYCPHASE,PUT** clears the result values for midside nodes on high order elements; therefore, this option sets element faceting (**/EFACET**) to 1. The command reports that midside nodal values are set to zero and indicates that element faceting is set to 1.

If the sweep values are available after issuing a **CYCPHASE,PUT** command, the **PRNSOL** or **PLNSOL** command will print or plot (respectively) the sweep values of structure displacement Ux, Uy, Uz, component stress/strain X, Y, Z, XY, YZ, ZX, principal stress/strain 1, 2, 3 and equivalent stress/strain EQV. The vector sum of displacement (USUM) and stress/strain intensity (SINT) are not valid phase-sweep results.

You can specify any coordinate system via the **RSYS** command for displaying or printing **CYCPHASE** results. However, after **CYCPHASE** results have been extracted, you cannot then transform them via the **RSYS** command. If you try to do so, ANSYS issues a warning message.

The **CYCPHASE** command is valid in **/POST1** and for cyclically symmetric models only.

To learn more about analyzing a cyclically symmetric structure, see [Cyclic Symmetry Analysis](#) in the *Advanced Analysis Techniques Guide*.

## Menu Paths

**Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Clear**  
**Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Get**  
**Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>List**  
**Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Phase Sweep**  
**Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Put Phase Results**  
**Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Status**

**CYL4**, *XCENTER*, *YCENTER*, *RAD1*, *THETA1*, *RAD2*, *THETA2*, *DEPTH*

**Creates a circular area or cylindrical volume anywhere on the working plane.**

PREP7: Primitives

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***XCENTER*, *YCENTER***

Working plane X and Y coordinates of the center of the circle or cylinder.

***RAD1*, *RAD2***

Inner and outer radii (either order) of the circle or cylinder. A value of zero or blank for either *RAD1* or *RAD2*, or the same value for both *RAD1* and *RAD2*, defines a solid circle or cylinder.

***THETA1*, *THETA2***

Starting and ending angles (either order) of the circle or faces of the cylinder. Used for creating a partial annulus or partial cylinder. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0° and the ending angle defaults to 360°. See the [Modeling and Meshing Guide](#) for an illustration.

***DEPTH***

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the cylinder. If *DEPTH* = 0 (default), a circular area is created on the working plane.

## Notes

Defines a circular area anywhere on the working plane or a cylindrical volume with one face anywhere on the working plane. For a solid cylinder of 360°, the top and bottom faces will be circular (each area defined with four lines) and they will be connected with two surface areas (each spanning 180°). See the [CYL5](#), [PCIRC](#), and [CYLIND](#) commands for alternate ways to create circles and cylinders.

When working with a model imported from an IGES file (DEFAULT import option), you must provide a value for *DEPTH* or the command will be ignored.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Annulus**

**Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Partial Annulus**

**Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Solid Circle**

**Main Menu>Preprocessor>Modeling>Create>Primitives>Solid Cylindr**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>Hollow Cylinder**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>Partial Cylinder**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>Solid Cylinder**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Cylinder>Solid Cylinder**

**CYL5**, *XEDGE1*, *YEDGE1*, *XEDGE2*, *YEDGE2*, *DEPTH*

**Creates a circular area or cylindrical volume by end points.**

PREP7: Primitives

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***XEDGE1*, *YEDGE1***

Working plane X and Y coordinates of one end of the circle or cylinder face.

***XEDGE2*, *YEDGE2***

Working plane X and Y coordinates of the other end of the circle or cylinder face.

***DEPTH***

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the cylinder. If *DEPTH* = 0 (default), a circular area is created on the working plane.

## Notes

Defines a circular area anywhere on the working plane or a cylindrical volume with one face anywhere on the working plane by specifying diameter end points. For a solid cylinder of 360°, the top and bottom faces will be circular (each area defined with four lines) and they will be connected with two surface areas (each spanning 180°). See the **CYL4**, **PCIRC**, and **CYLIND** commands for alternate ways to create circles and cylinders.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Circle>By End Points**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>By End Pts & Z**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Cylinder>By End Pts & Z**

**CYLIND**, *RAD1*, *RAD2*, *Z1*, *Z2*, *THETA1*, *THETA2*

**Creates a cylindrical volume centered about the working plane origin.**

PREP7: Primitives

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***RAD1*, *RAD2***

Inner and outer radii (either order) of the cylinder. A value of zero or blank for either *RAD1* or *RAD2*, or the same value for both *RAD1* and *RAD2*, defines a solid cylinder.

***Z1*, *Z2***

Working plane Z coordinates of the cylinder. If either *Z1* or *Z2* is zero, one of the faces of the cylinder will be coplanar with the working plane.

***THETA1*, *THETA2***

Starting and ending angles (either order) of the cylinder. Used for creating a cylindrical sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0.0° and the ending angle defaults to 360.0°. See the *Modeling and Meshing Guide* for an illustration.

## Notes

Defines a cylindrical volume centered about the working plane origin. The top and bottom faces are parallel to the working plane but neither face need be coplanar with (i.e., "on") the working plane. The cylinder must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) For a solid cylinder of 360°, the top and bottom faces will be circular (each area defined with four lines), and they will be connected with two areas (each spanning 180°.) See the **CYL4** and **CYL5** commands for alternate ways to create cylinders.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>By Dimensions**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Cylinder>By Dimensions**

---

**CZDEL**, *grp1*, *grp2*, *grp3*

**Edits or clears cohesive zone sections.**

[PREP7: Meshing](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

*grp1*

Initial group of cohesive zone elements to be deleted.

*grp2*

Final group of cohesive zone elements to be deleted.

*grp3*

Increment for selected groups.

## Notes

The **CZDEL** command edits or deletes the interface elements and nodes, along with the associated changes made to the underlying plane or solid elements created during a previous **CZMESH** operation.

Each **CZMESH** operation will create groups of elements and nodes with component names in the format CZME\_EL01 (elements) and CZME\_ND01 (nodes). The final number of this format will be the number used for *grp1* and *grp2*. If *grp1* = ALL, all nodes and elements created by the **CZMESH** command will be deleted. After using **CZDEL**, all the user-defined components will be unselected.

The **CZDEL** command is valid for structural analyses only.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**CZMESH**, *ecomps1*, *ecomps2*, *KCN*, *KDIR*, *VALUE*, *CZTOL*

**Create and mesh an interface area composed of cohesive zone elements.**

PREP7: Meshing

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***ecomps1***

Component name or number for the group of plane or solid structural elements adjacent to the interface being meshed.

***ecomps2***

Component name or number for the opposing (from *ecomps1*) group of plane or solid structural elements adjacent to the interface being meshed.

***KCN***

Coordinate system number for the separation surface and normal direction. (if *ecomps1* and *ecomps2* not specified)

***KDIR***

Direction (x, y, or z) normal to separation surface in the *KCN* coordinate system (if *ecomps1* and *ecomps2* not specified).

***VALUE***

Coordinate value along the *KDIR* axis at which to locate the interface (if *ecomps1* and *ecomps2* not specified).

***CZTOL***

Optional absolute tolerance about *VALUE* (if *ecomps1* and *ecomps2* not specified). Allows nodes occurring slightly above or below the separation to be grouped properly. The following expression represents the default value:

$$\frac{\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}}{1000}$$

Where  $\Delta X$ ,  $\Delta Y$ , and  $\Delta Z$  are the dimensions of the model based on nodal locations (that is,  $\Delta X = X_{max} - X_{min}$ ).

## Notes

**CZMESH** is used on a mesh with shared nodes at the interface.

If *ecomps1* and *ecomps2* are specified, the **CZMESH** command creates/meshes interface elements ([INTER202](#), [INTER203](#), [INTER204](#), [INTER205](#)) along the boundary between the two components or groups of elements.

The elements in each of the components or groups of elements will share nodes with each other and also with the interface elements. This one-element thick boundary of interface elements will split the body between the two components or groups of elements.

Subsequent separation (delamination and failure) of the interface zone will result in an increasing displacement between the nodes (within the interface element) along the cohesive zone elements. Unless otherwise specified, the **CZMESH** command analyzes the configuration and geometry of the adjacent structural elements and provides the appropriate interface element.

The **CZMESH** operation copies any nodal temperatures you have defined on the split surface of the original mesh from the original nodes to the newly created coincident duplicate nodes. However, displacements, forces, and other boundary conditions are not copied.

The **CZMESH** command is valid for structural analyses only.

## Menu Paths

**This command cannot be accessed from a menu.**

---





---

## D Commands

**D**, *NODE*, *Lab*, *VALUE*, *VALUE2*, *NEND*, *NINC*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

**Defines degree-of-freedom constraints at nodes.**

**SOLUTION: FE Constraints**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NODE*

Node at which constraint is to be specified. If ALL, *NEND* and *NINC* are ignored and constraints are applied to all selected nodes (**NSEL**). If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

### *Lab*

Valid degree-of-freedom label. If ALL, use all appropriate labels.

**Structural labels:** UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping).

Thermal labels: TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature).

**FLOTRAN fluid labels:** PRES (pressure); VX, VY, or VZ (velocities); ENKE or ENDS (turbulent kinetic energy or turbulent kinetic energy dissipation rate); SP01 through SP06 (multiple species mass fractions) or their user-defined names (**MSSPEC**); UX, UY, or UZ (Arbitrary Lagrangian-Eulerian formulation mesh displacements).

**Electric labels:** VOLT (voltage); EMF (electromotive force).

**Magnetic labels:** MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials).

For structural static and transient analyses, translational and rotational velocities are also valid loads. Use these labels: VELX, VELY, VELZ (translational velocities); OMGX, OMGY, OMGZ (rotational velocities).

For structural analyses, HDSP (hydrostatic pressure) is also valid. However, HDSP is *not* included when *Lab* = ALL.

For structural transient analyses, the following acceleration loads are also valid: ACCX, ACCY, ACCZ (translational accelerations); DMGX, DMGY, DMGZ (rotational accelerations). The velocity and acceleration loads are *not* included when *Lab* = ALL.

If the node is connected to an **ELBOW290** element, the following pipe cross-section degree-of-freedom labels are also valid: SE, SO, SW, SRA, and SRT. (For details, see the **ELBOW290** documentation.) The degrees of freedom are *not* included when *Lab* = ALL. To constrain *all* cross-section degrees of freedom, specify *Lab* = SECT.

### *Value*

Degree-of-freedom value or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent (%) signs (for example, **D,NODE,TEMP,%tablename%**). Use the **\*DIM** command to define a table.

If *Value* = SUPPORT, you can specify pseudo-constraints when **using residual vectors** in a modal analysis (**RESVEC,ON**) or **CMS** analysis (**CMSOPT,RFFB**).

If the enforced motion is active in the modal analysis (**MODCONT**,,on), *Value* is the base identification number. It should be an integer larger than 1 and less than 10000.

#### **VALUE2**

Second degree-of-freedom value (if any). If the analysis type and the degree of freedom allow a complex input, *Value* (above) is the real component and *VALUE2* is the imaginary component.

#### **NEND, NINC**

Specifies the same values of constraint at the range of nodes from *NODE* to *NEND* (defaults to *NODE*), in steps of *NINC* (defaults to 1).

#### **Lab2, Lab3, Lab4, Lab5, Lab6**

Additional degree-of-freedom labels. The same values are applied to the nodes for these labels.

## Notes

The available degrees of freedom per node are listed under "Degrees of Freedom" in the input table for each element type in the [Element Reference](#). Degrees of freedom are defined in the nodal coordinate system. The positive directions of structural translations and rotations are along and about the positive nodal axes directions. Structural rotations should be input in radians. The node and the degree-of-freedom label must be selected (**NSEL, DOFSEL**).

If *Lab* = ENKE and *Value* = -1, a FLOTRAN flag is set to indicate a moving wall.

If *Lab* = ENDS and *Value* = -1, FLOTRAN generalized symmetry conditions are applied. Velocity components are set tangential to the symmetry surface if the ALE formulation is not activated. They are set equal to the mesh velocity if the ALE formulation is activated.

In a structural analysis, you can apply only one displacement, velocity, or acceleration load at any degree of freedom. If multiple loads are specified, the last applied load overrides the previous ones. For example, the following commands apply loads to node 100:

```
D,100,UX,Value
D,100,VELX,Value
```

In this case, the velocity load (VELX) applied in the last command will override the displacement load (UX).

For elements **HF119** and **HF120**, used in high-frequency electromagnetic analysis, the AX degree of freedom is not an x-component of a vector potential, but rather a tangential component of E (the electric field) on the element edges and faces. To specify an Electric Wall condition, set AX to zero. For more information, see the [High-Frequency Electromagnetic Analysis Guide](#).

For elements used in static and low frequency electromagnetic analysis (**SOLID117**, **SOLID236** and **SOLID237**), the AZ degree of freedom is not a z-component of a vector potential, but rather the flux contribution on the element edge. To specify a flux-parallel condition, set AZ = 0. For more information, see [3-D Magnetostatics and Fundamentals of Edge-based Analysis](#) in the [Low-Frequency Electromagnetic Analysis Guide](#).

In an explicit dynamic analysis, the **D** command can only be used to fix nodes in the model. The degree-of-freedom value must be zero; no other values are valid. Use the **EDLOAD** command to apply a nonzero displacement in an explicit dynamic analysis.

For **ELBOW290** cross-section degrees of freedom (*Lab* = SE, SO, SW, SRA, SRT, or SECT), the **D** command can only specify fixed constraints. The degree-of-freedom value must be zero; no other values are valid.

For hydrostatic fluid elements ([HSFLD241](#) and [HSFLD242](#)), the HDSP degree-of-freedom constraint at the pressure node prescribes the pressure value for all the fluid elements sharing the pressure node.

Tabular boundary conditions ( $VALUE = \%tabname\%$ ) are available only for the following degree-of-freedom labels: Electric (VOLT), FLOTRAN (UX, UY, UZ, PRES, VX, VY, VZ, ENKE, ENDS, TEMP, SP01, SP02, SP03, SP04, SP05, and SP06); structural (UX, UY, UZ, ROTX, ROTY, ROTZ, and velocity and acceleration loads VELX, VELY, VELZ, OMGX, OMGY, OMGZ, ACCX, ACCY, ACCZ, DMGX, DMGY, DMGZ), and temperature (TEMP, TBOT, TE2, TE3, . . . , TTOP). All labels are valid only in static (**ANTYPE,STATIC**) and full transient (**ANTYPE,TRANS**) analyses.

$\%\_FIX\%$  is an ANSYS reserved table name. When  $VALUE$  is set to  $\%\_FIX\%$ , ANSYS will prescribe the degree of freedom to the “current” displacement value. Alternatively, functions UX(), UY(), etc. may be used (see **\*GET** for a complete list of available functions). However, note that these functions are not available when multiframe restart features are invoked. In most cases,  $\%\_FIX\%$  usage is efficient and recommended for all structural degrees of freedom.

When  $Value = SUPPORT$ , specify only the minimum number of displacement constraints necessary to prevent rigid body motion: three constraints (or fewer, depending on the element type) for 2-D models and six (or fewer) for 3-D models.

If constraints and initial conditions (**IC**) are applied at the same node, the constraint specification overrides. This combination is useful when a constraint degree-of-freedom value needs to start with a nonzero value at time = 0.0. For example, if the constraint degree-of-freedom value is prescribed to be a cosine function, then specifying an initial condition for the same node and degree of freedom ensures that the initial value for the constraint degree of freedom at time = 0.0 is same as the cosine function evaluated at time = 0.0. If initial conditions are not specified, the constraint degree-of-freedom value ramps from zero in the first substep of the first loadstep.

If more than one rotational degrees of freedom are constrained with non-zero rotations (ROTX, ROTY, ROTZ), rotational velocities (OMGX, OMGY, OMGZ), or rotational accelerations (DMGX, DMGY, DMGZ), then the rotation of the constrained node from its initial configuration to its final configuration depends on the combination and the sequence in which the constraints are applied. See [Rotations in a Large-Deflection Analysis](#) in *Structural Analysis Guide*.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PerfEC>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Nodes**

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Node Components  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>BasePSD>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>BasePSD>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Nodes  
 Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>On Nodes  
 Main Menu>Solution>Constraints>Apply>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PerfEC>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'I>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'I>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Node Components  
 Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>BasePSD>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>BasePSD>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Nodes

**DA**, *AREA*, *Lab*, *Value1*, *Value2*

Defines DOF constraints on areas.

SOLUTION: Solid Constraints

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### **AREA**

Area on which constraints are to be specified. If ALL, apply to all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *AREA*.

#### **Lab**

Symmetry label (see 2 (p. 354) below):

#### **SYMM**

Generate symmetry constraints for non-FLOTRAN models. Requires no *Value1* or *Value2*.

#### **ASYM**

Generate antisymmetry constraints for non-FLOTRAN models. Requires no *Value1* or *Value2*.

ANSYS DOF labels (see 1 (p. 354), 2 (p. 354), and 3 (p. 354) below):

#### **UX**

Displacement in X direction.

**UY**

Displacement in Y direction.

**UZ**

Displacement in Z direction.

**ROTX**

Rotation about X axis.

**ROTY**

Rotation about Y axis.

**ROTZ**

Rotation about Z axis.

**HDSP**

Hydrostatic pressure.

**TEMP, TBOT, TE2, TE3, . . . , TTOP**

Temperature.

**MAG**

Magnetic scalar potential.

**VOLT**

Electric scalar potential.

**AX**

Magnetic vector potential in X direction (see 4 (p. 354)).

**AY**

Magnetic vector potential in Y direction.

**AZ**

Magnetic vector potential in Z direction (see 1 (p. 354)).

**ALL**

Applies all appropriate DOF labels *except* HDSP.

FLOTRAN Standard DOF Labels (see 7 (p. 354)): VX, VY, VZ, PRES, TEMP, ENKE, ENDS

FLOTRAN Species Labels (See 8 (p. 354)): SP01, SP02, SP03, SP04, SP05, SP06

FLOTRAN Arbitrary Lagrangian-Eulerian formulation Mesh Displacement Labels (See 9 (p. 354)): UX, UY, UZ

**Value1**

Value of DOF or table name reference on the area. Valid for all DOF labels. To specify a table, enclose the table name in % signs (e.g., **DA,AREA,TEMP,%tablename%**). Use the **\*DIM** command to define a table.

If  $Lab = ENKE$  and  $Value1 = -1$ , a FLOTRAN flag is set to indicate a moving wall.

If  $Lab = ENDS$  and  $Value1 = -1$ , FLOTRAN generalized symmetry conditions are applied. Velocity components are set tangential to the symmetry surface if the ALE formulation is not activated. They are set equal to the mesh velocity if the ALE formulation is activated.

**Value2**

For FLOTRAN DOFs:

**0**

Values are applied only to nodes within the area.

## 1

Values are applied to the edges of the area as well as to the internal nodes. (See 7 (p. 354).)

For MAG and VOLT DOFs:

Value of the imaginary component of the degree of freedom.

## Notes

- For elements [SOLID117](#), [SOLID236](#), and [SOLID237](#), if  $Lab = AZ$  and  $Value1 = 0$ , this sets the flux-parallel condition for the edge formulation. (A flux-normal condition is the natural boundary condition.) Do *not* use the **DA** command to set the edge-flux DOF, AZ to a nonzero value.
- If  $Lab = MAG$  and  $Value1 = 0$ , this sets the flux-normal condition for the magnetic scalar potential formulations (MSP) (A flux-parallel condition is the natural boundary condition for MSP.)
- If  $Lab = VOLT$  and  $Value1 = 0$ , the J-normal condition is set (current density (J) flow normal to the area). (A J-parallel condition is the natural boundary condition.)
- For elements [HF119](#) and [HF120](#), used in high-frequency electromagnetic analysis, the AX DOF is not an x-component of a vector potential, but rather a tangential component of E (the electric field) on the element edges and faces. To specify an Electric Wall condition, set AX to zero. For more information, see the [High-Frequency Electromagnetic Analysis Guide](#).
- You can transfer constraints from areas to nodes with the **DTRAN** or **SBCTAN** commands. See the **DK** command for information about generating other constraints on areas for non-FLOTRAN models.
- Symmetry and antisymmetry constraints are generated as described for the **DSYM** command.
- For the velocity DOF (VX, VY, VZ), a zero value will override a nonzero value at the intersection of two areas.
- You can use the **MSSPEC** command to change FLOTRAN species labels to user-defined labels. You must define these labels with the **MSSPEC** command before using them on the **DA** command.
- Tabular boundary conditions ( $VALUE = \%tabname\%$ ) are available only for the following degree of freedom labels: Electric (VOLT), FLOTRAN (UX, UY, UZ, PRES, VX, VY, VZ, ENKE, ENDS, TEMP, SP01, SP02, SP03, SP04, SP05, and SP06); Structural (UX, UY, UZ, ROTX, ROTY, ROTZ), and temperature (TEMP, TBOT, TE2, TE3, . . . , TTOP).
- Constraints specified by the **DA** command can conflict with other specified constraints. See [Resolution of Conflicting Constraint Specifications](#) in the [Basic Analysis Guide](#) for details.
- The **DA** command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PerfEC>On Areas**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Areas**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>On Areas**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Areas**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Areas**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Areas**



Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Areas  
 Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>On Areas  
 Main Menu>Solution>Constraints>Apply>On Areas  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PerfEC>On Areas  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Areas  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>On Areas  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Areas  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Areas  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Areas  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Areas  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Areas  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Areas  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Areas  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Areas  
 Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Areas  
 Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Areas  
 Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Areas  
 Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Areas

## DADELE, AREA, Lab

Deletes DOF constraints on an area.

**SOLUTION:** Solid Constraints

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### AREA

Area for which constraints are to be deleted. If ALL, delete for all selected areas [ASEL]. If AREA = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). You can substitute a component name for AREA.

### Lab

Valid constraint labels are:

#### ALL

All constraints.

#### SYMM

Symmetry constraints.

**ASYM**

Antisymmetry constraints.

**UX**

Displacement in X direction.

**UY**

Displacement in Y direction.

**UZ**

Displacement in Z direction.

**ROTX**

Rotation about X axis.

**ROTY**

Rotation about Y axis.

**ROTZ**

Rotation about Z axis.

**VX**

Velocity component in X direction.

**VY**

Velocity component in Y direction.

**VZ**

Velocity component in Z direction.

**PRES**

Pressure.

**TEMP, TBOT, TE2, TE3, . . . , TTOP**

Temperature.

**ENKE**

Turbulent Kinetic Energy.

**ENDS**

Energy Dissipation Rate.

**MAG**

Magnetic scalar potential.

**VOLT**

Electric scalar potential.

**SP01 . . . SP06**

Multiple Species Mass Fraction.

**AX**

Magnetic vector potential in X direction (see notes).

**AY**

Magnetic vector potential in Y direction.

**AZ**

Magnetic vector potential in Z direction (see notes).



## Notes

Deletes the degree of freedom constraints at an area (and all corresponding finite element constraints) previously specified with the **DA** command. See the **DDELE** command for delete details.

If the multiple species labels have been changed to user-defined labels via the **MSSPEC** command, use the user-defined labels.

See the **DA** or the **DA** commands for details on element applicability.

### Warning

*On previously meshed areas, all constraints on affected nodes will be deleted, whether or not they were specified by the **DA** command.*

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PerfEC>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>TimeInt>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Displacement>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Species>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Turbulence>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Velocity>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Areas**  
**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>On Areas**  
**Main Menu>Solution>Constraints>Delete>On Areas**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All Areas**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PerfEC>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>TimeInt>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Displacement>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Species>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Turbulence>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Velocity>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Areas**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Areas**

**DALIST**, *AREA*

**Lists the DOF constraints on an area.**

[SOLUTION: Solid Constraints](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**AREA**

List constraints for this area. If ALL (default), list for all selected areas [**ASEL**]. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *AREA*.

**Notes**

Lists the degree of freedom constraints on an area previously specified with the **DA** command.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>List>Loads>DOF Constraints>On All Areas**  
**Utility Menu>List>Loads>DOF Constraints>On Picked Areas**

**DAMORPH**, *AREA, XLINE, RSMHXY*

**Move nodes in selected areas to conform to structural displacements.**

[PREP7: Morphing](#)

MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

**AREA**

Non-structural area to which mesh movement (morph) applies. If ALL, apply morphing to all selected areas [ASEL]. If *AREA* = P, graphical picking is enabled. A component may be substituted for *AREA*.

**XLINE**

Lines to be excluded from morphing. If ALL, exclude all selected lines [LSEL] from morphing. If *XLINE* = P, graphical picking is enabled. A component may be substituted for *XLINE*. If *XLINE* is blank (default), allow morphing of nodes attached to lines of the selected areas (*AREA*) which are not shared by unselected areas. See Notes for clarification.

**RSMHXY**

Remesh flag option:

- 0**  
Remesh the selected non-structural areas only if mesh morphing fails.
- 1**  
Remesh the selected non-structural areas and bypass mesh morphing.
- 2**  
Perform mesh morphing only and do not remesh.

## Notes

The selected areas should include only non-structural regions adjacent to structural regions. **DAMORPH** will morph the non-structural areas to coincide with the deflections of the structural regions.

Nodes in the structural regions move in accordance with computed displacements. Displacements from a structural analysis must be in the database prior to issuing **DAMORPH**.

By default, nodes attached to lines can move along the lines, or off the lines (if a line is interior to the selected areas). You can use *XLINK* to restrain nodes on certain lines.

By default (*RMSHKEY* = 0), **DAMORPH** will remesh the selected non-structural areas entirely if a satisfactory morphed mesh cannot be provided.

If boundary conditions and loads are applied directly to nodes and elements, the **DAMORPH** command requires that these be removed before remeshing can take place.

Exercise care with initial conditions defined by the **IC** command. Before a structural analysis is performed for a sequentially coupled analysis, the **DAMORPH** command requires that initial conditions be removed from all null element type nodes in the non-structural regions. Use **ICDELE** to delete the initial conditions.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Phys Morphing>Areas

**DATA**, *IR*, *LSTRT*, *LSTOP*, *LINC*, *Name*, *KCPLX*

Reads data records from a file into a variable.

POST26: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *IR*

Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

### *LSTRT*

Start at location *LSTRT* (defaults to 1).

### *LSTOP*

Stop at location *LSTOP* (defaults to *LSTRT*). Maximum location available is determined from data previously stored.

### *LINC*

Fill every *LINC* location between *LSTRT* and *LSTOP* (defaults to 1).

### *Name*

Eight character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

**KCPLX**

Complex number key:

- 0**  
Data stored as the real part of the complex number.
- 1**  
Data stored as the imaginary part of the complex number.

**Notes**

This command must be followed by a format statement (on the next line) and the subsequent data records, and all must be on the same file (that may then be read with the **/INPUT** command). The format specifies the number of fields to be read per record, the field width, and the placement of the decimal point (if one is not included in the data value). The read operation follows the available FORTRAN FORMAT conventions of the system. See the system FORTRAN manual for details. Any standard FORTRAN real format (such as (4F6.0), (F2.0,2X,F12.0), etc.) may be used. Integer (I), character (A), and list-directed (\*) descriptors may *not* be used. *The parentheses must be included in the format.* Up to 80 columns per record may be read. Locations may be filled within a range. Previous data in the range will be overwritten.

**Menu Paths**

**This command cannot be accessed from a menu.**

---

**DATADEF**

Specifies "Directly defined data status" as the subsequent status topic.

POST1:Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

**Menu Paths**

**Utility Menu>List>Status>General Postproc>Modify Results**

---

**DCGOMG**, *DCGOX*, *DCGOY*, *DCGOZ*

Specifies the rotational acceleration of the global origin.

SOLUTION: Inertia

MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS

*DCGOX*, *DCGOY*, *DCGOZ*

Rotational acceleration of the global origin about the acceleration system X, Y, and Z axes.

**Notes**

Specifies the rotational acceleration of the global origin about each of the acceleration coordinate system axes [**CGLOC**]. Rotational accelerations may be defined in analysis types **ANTYPE**, **STATIC**, **HARMIC** (full or mode superposition), **TRANS** (full or mode superposition), and **SUBSTR**. See [Acceleration Effect](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details. Units are radians/time<sup>2</sup>.

The **DCGOMG** command supports tabular boundary conditions (%*TABNAME\_X*%, %*TABNAME\_Y*%, and %*TABNAME\_Z*%) for *DCGOMG\_X*, *DCGOMG\_Y*, and *DCGOMG\_Z* input values (**\*DIM**) for full transient and harmonic analyses.

Related commands are **ACEL**, **CGLOC**, **CGOMGA**, **DOMEGA**, and **OMEGA**.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Coriolis Effects**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Coriolis Effects**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Coriolis Effects**  
**Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Coriolis Effects**

**DCUM**, *Oper*, *RFACT*, *IFACT*, *TBASE*

Specifies that DOF constraint values are to be accumulated.

SOLUTION: FE Constraints

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

*Oper*

Accumulation key:

**REPL**

Subsequent values replace the previous values (default).

**ADD**

Subsequent values are added to the previous values.

**IGNO**

Subsequent values are ignored.

*RFACT*

Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

**IFACT**

Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

**TBASE**

Base temperature for temperature difference. Used only with temperature degree of freedom. Scale factor is applied to the temperature difference ( $T-TBASE$ ) and then added to  $TBASE$ .  $T$  is the current temperature.

**Command Default**

Replace previous values.

**Notes**

Allows repeated degree of freedom constraint values (displacement, temperature, etc.) to be replaced, added, or ignored. Operations apply to the selected nodes [**NSEL**] and the selected degree of freedom labels [**DOFSEL**]. This command also operates on velocity and acceleration loads applied in a structural analysis.

The operations occur when the next degree of freedom constraints are defined. For example, issuing the command **D,1,UX,.025** after a previous **D,1,UX,.020** causes the new value of the displacement on node 1 in the x-direction to be 0.045 with the add operation, 0.025 with the replace operation, or 0.020 with the ignore operation. Scale factors are also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a displacement of 0.070. Scale factors are applied even if no previous values exist. Issue **DCUM,STAT** to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

---

**Note**

FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

**DCUM** does not work for tabular boundary conditions.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Constraints**  
**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Constraints**

---

**DCVSWP**, *Option, Elem, Cnum, Vmax, Vinc1, Vinc2, Gap*

**Performs a DC voltage sweep on a ROM element.**

REDUCED ORDER MODELING: Use Pass

MP <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

**Option**

Sweep option:

**GV**

Perform voltage sweep up to given voltage  $V_{max}$ .

**PI**

Perform a voltage sweep simulation up to the pull-in voltage.

**Elem**

Element number of the ROM element for the ROM use pass analysis.

**Cnum**

Number of sweep conductor.

**Vmax**

Maximum voltage. For the PI option, this voltage should be below the pull-in voltage value.

**Vinc1**

Voltage increment for Vmax (default =  $V_{max}/20$ ).

**Vinc2**

Voltage increment for pull-in voltage (default = 1).

**Gap**

Gap elements option:

**0**

Create gap elements ([COMBIN40](#)) (default).

**1**

Do not create gap elements

## Notes

*Vinc1* is used to ramp the sweep conductor voltage from 0 to  $V_{max}$ . *Vinc2* is used to increase the sweep conductor voltage from  $V_{max}$  to the pull-in value if the PI sweep option is used.

Because ramping the voltage may lead to the unstable region of an electromechanical system, **DCVSWP** might not converge when the sweep conductor voltage approaches the pull-in value. To avoid non-converged solutions, you should use the gap option to create a set of spring-gap elements ([COMBIN40](#)). By default, **DCVSWP** creates two spring-gap elements with opposite orientations for each active modal displacement DOF of the ROM element. The gap size is set to the maximum absolute values of the deflection range for the corresponding mode, as calculated by [RMMSELECT](#) or modified using the [RMMRANGE](#) command. The spring constants are set to 1.E5 for all the [COMBIN40](#) elements. Along with the spring-gap elements, **DCVSWP** creates a set of constraint equations relating the ROM element modal displacements DOF (EMF) and the displacement DOF (UX) of the gap elements. Constraining the modal displacements using the spring-gap elements allows **DCVSWP** to converge in the pull-in range. The **DCVSWP** macro has a limit of 900 equilibrium iterations. If this limit is not sufficient to reach convergence, try the advanced techniques given in [Overcoming Convergence Problems](#) in the *Structural Analysis Guide*. For more information on gap elements, see [Using Gap Elements with ROM144](#) in the *Coupled-Field Analysis Guide*.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>ROM Tools>Voltage Sweep**

**Main Menu>Solution>ROM Tools>Voltage Sweep**

**DDELE**, *NODE*, *Lab*, *NEND*, *NINC*, *Rkey*

**Deletes degree-of-freedom constraints.**

SOLUTION: FE Constraints

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NODE***

Node for which constraint is to be deleted. If ALL, *NEND* and *NINC* are ignored and constraints for all selected nodes [**NSEL**] are deleted. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

***Lab***

Valid degree of freedom label. If ALL, use all selected labels [**DOFSEL**]. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping). Thermal labels: TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature). FLOTRAN fluid labels: PRES (pressure); VX, VY, or VZ (velocities); ENKE or ENDS (turbulent kinetic energy or turbulent energy dissipation); SP01 through SP06 (multiple species mass fractions) or their user-defined names. Electric label: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials). High-frequency electromagnetic label: AX (Electric Wall or Magnetic Wall boundary condition).

In structural analyses, the following velocity and acceleration load labels are also valid: VELX, VELY, VELZ (translational velocities); OMGX, OMGY, OMGZ (rotational velocities); ACCX, ACCY, ACCZ (translational accelerations); DMGX, DMGY, DMGZ (rotational accelerations).

In structural analyses, HDSP (hydrostatic pressure) is also valid.

If the node is connected to an **ELBOW290** element, the following pipe cross-section degree of freedom labels are also valid: SE, SO, SW, SRA, and SRT. (For details, see the **ELBOW290** documentation.) The degrees of freedom are *not* included when *Lab* = ALL. To constrain *all* cross-section degrees of freedom, specify *Lab* = SECT.

***NEND*, *NINC***

Delete constraints from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1).

***Rkey***

Ramping key:

**OFF**

Loads are step-removed (default).

**ON or FORCE**

Forces on the specified degrees of freedom (*Lab*) are ramped during the next load step. The forces are ramped from the reaction forces of the previous load step to zero or currently defined force values (if any).

**Notes**

Deleting a constraint is not the same as setting it to zero (which "fixes" the degree of freedom to a zero value). Deleting a constraint has the same effect as deactivating, releasing, or setting the constraint "free." The node and the degree of freedom label must be selected [**NSEL**, **DOFSEL**].

For elements **HF119** and **HF120**, used in high-frequency electromagnetic analysis, the AX DOF is not an x-component of a vector potential, but rather a tangential component of E (the electric field) on the element



edges and faces. To specify an Electric Wall condition, set AX to zero. For more information, see the [High-Frequency Electromagnetic Analysis Guide](#).

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PerfEC>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>TimeInt>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Displacement>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Species>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Turbulence>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Velocity>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Node Components**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Nodes**  
**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>On Nodes**  
**Main Menu>Solution>Constraints>Delete>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All Nodes**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PerfEC>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>TimeInt>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Displacement>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Species>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Turbulence>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Velocity>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Node Components**  
**Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Nodes**

**DDOPTION**, *Decomp*, --, --, *ContKey***Sets domain decomposer option for Distributed ANSYS.**

SOLUTION: Analysis Options

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; EM EH &lt;&gt; PP &lt;&gt; EME MFS

**Decomp**

Controls which domain decomposition algorithm to use.

**AUTO**

Use the default domain decomposition algorithm when splitting the model into domains for Distributed ANSYS. Currently, the default for all jobs is the GREEDY algorithm.

**GREEDY**

Use the "greedy" domain decomposition algorithm.

**METIS**

Use the METIS graph partitioning domain decomposition algorithm.

-- , --

Unused fields.

**ContKey**

Tells the domain decomposition algorithm how to handle contact pairs.

**DEFAULT**

By default, all elements belonging to a single contact pair must belong to a single CPU domain. However, each contact pair may reside on a different CPU domain unless multiple contact pairs (along with the required underlying elements) overlap, in which case the entire overlapping set must reside within a single CPU domain.

**GROUP**

This option forces all of the contact pairs in the model (along with the required underlying elements) to be grouped onto a single CPU domain.

**Command Default**

The greedy domain decomposition algorithm is used.

**Notes**

This command controls options relating to the domain decomposition algorithm used by Distributed ANSYS to split the original model into pieces, with each piece being solved on a different CPU.

The greedy domain decomposition algorithm starts from a single element at a corner of the model. The domain grows by taking the properly connected neighboring elements and stops after reaching the optimal size.

The METIS domain decomposition algorithm starts by creating a graph from the finite element mesh. It then uses a multilevel graph partitioning scheme which reduces the size of the original graph, creates domains using the reduced graph, and then creates the final CPU domains by expanding the smaller domains from the reduced graph back to the original mesh.

Typically, when using just a few different element types in the model, the METIS decomposer will result in less communication and better load balancing. This often leads to decreased overall run times, particularly when using higher numbers of processors (for example, 8 or more CPUs) or when using cluster configurations

with slower interconnect speeds (for example, 4 machines with 1 CPU per machine using a GigE interconnect). However, the METIS decomposition method is sometimes not as robust as the greedy algorithm and, therefore, is not the default.

## Menu Paths

This command cannot be accessed from a menu.

---

## DEACT

Specifies "Element birth and death" as the subsequent status topic.

SOLUTION:Status

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

## Notes

This is a status [STAT] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Solution>Elem Birth/Death

---

## DEFINE

Specifies "Data definition settings" as the subsequent status topic.

POST1:Status

POST26:Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [STAT] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>General Postproc>Read Options

Utility Menu>List>Status>TimeHist Postproc>Variables

**\*DEL**, *Val1*, *Val2*

**Deletes a parameter or parameters (GUI).**

APDL: Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Val1*

*Val1* can be:

#### **ALL** --

Indicates that you want to delete all user-defined parameters, or both all user-defined and all system parameters, as indicated by the *Val2* argument.

#### **(blank)** --

Indicates that you want to delete the parameter(s) indicated by *Val2*.

### *Val2*

*Val2* can be:

#### **LOC** --

When *Val1* is (blank), use *Val2* to specify the location of the parameter within the [Array Parameters](#) dialog box. The location number is based on an alphabetically ordered list of all parameters in the database. Not valid when *Val1* is ALL.

#### **\_PRM** --

When *Val1* is ALL, specifying **\_PRM** for *Val2* deletes all parameters, including those named with an initial underbar ( `_` ) (except `_STATUS` and `_RETURN`). When *Val1* is (blank), specifying **\_PRM** for *Val2* deletes only those parameters named with an initial underbar ( `_` ) (except `_STATUS` and `_RETURN`).

#### **PRM\_** --

When *Val1* is (blank), specifying **PRM\_** for *Val2* deletes only those parameters named with a trailing underbar ( `_` ). Not valid when *Val1* is ALL.

#### **(blank)** --

When *Val1* is ALL, specifying (blank) for *Val2* causes all user-defined parameters to be deleted.

## Notes

This is a command generally created by the Graphical User Interface (GUI). It will appear in the log file (`Jobname.LOG`) if an array parameter is deleted from within the Array Parameters dialog box.

To delete all user-defined parameters, issue the command **\*DEL,ALL**. To delete only those user-defined parameters named with a trailing underbar, issue the command **\*DEL,,PRM\_**. To delete all user-defined and all system parameters (except for `_STATUS` and `_RETURN`), issue the command **\*DEL,ALL,\_PRM**. To delete a parameter by specifying its location within the Array Parameters dialog box, issue the command **\*DEL,,LOC**.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

**DELETE**, *SET*, *Nstart*, *Nend*

**Specifies sets in the results file to be deleted before postprocessing.**

[AUX3: Results Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***SET***

Specifies that sets in the results file are to be deleted.

***Nstart***

The first set in a results file to be deleted.

***Nend***

The final set in a results file to be deleted. This field is used only if deleting more than one sequential sets.

**Notes**

**DELETE** is a specification command that flags sets in the results file for deletion. It should be followed by a **COMPRESS** command, the corresponding action command that *deletes* the specified sets.

The **DELETE** command is valid only in the results file editing processor (ANSYS auxiliary processor AUX3).

**Menu Paths**

**This command cannot be accessed from a menu.**

**/DELETE**, *Fname*, *Ext*, --

**Deletes a file.**

[SESSION: Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to the current `Jobname`.

***Ext***

Filename extension (8 character maximum).

--

Unused field.

**Notes**

For Distributed ANSYS, if *Fname* is the current `Jobname` (or blank, which defaults to the current `Jobname`), the command will be executed on the master and all slave processors. However, if *Fname* is anything other than the current `Jobname`, this command is ignored on the slave processors and is executed only on the master processor.

## Menu Paths

Utility Menu>File>File Operations>Delete  
 Utility Menu>PlotCtrls>Redirect Plots>To GRPH File  
 Utility Menu>PlotCtrls>Redirect Plots>To HPGL File  
 Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File  
 Utility Menu>PlotCtrls>Redirect Plots>To PSCR File

**DELTIM**, *DTIME*, *DTMIN*, *DTMAX*, *Carry*

Specifies the time step sizes to be used for this load step.

[SOLUTION: Load Step Options](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### ***DTIME***

Time step size for this step. If automatic time stepping is being used [**AUTOTS**], *DTIME* is the starting time substep. If **SOLCONTROL,ON** and contact elements [TARGE169](#), [TARGE170](#), [CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#), [CONTA175](#), [CONTA176](#), or [CONTA177](#) are used, defaults to 1 or 1/20 the total time span of the load step, depending on the physics of the problem. If **SOLCONTROL,ON** and none of these contact elements are used, defaults to 1 time span of the load step. If **SOLCONTROL,OFF**, defaults to the previously specified value.

### ***DTMIN***

Minimum time step (if automatic time stepping is used). If **SOLCONTROL,ON**, default determined by ANSYS depending on the physics of the problem. If **SOLCONTROL,OFF**, defaults to the previously specified value (or *DTIME*, if there is no previously specified value).

### ***DTMAX***

Maximum time step (if automatic time stepping is used). If **SOLCONTROL,ON**, default determined by ANSYS depending on the physics of the problem. If **SOLCONTROL,OFF**, defaults to the previously specified value (or the time span of the load step, if there is no previously specified value).

### ***Carry***

Time step carry over key:

#### **OFF**

Use *DTIME* as time step at start of each load step.

#### **ON**

Use final time step from previous load step as the starting time step (if automatic time stepping is used).

If **SOLCONTROL,ON**, default determined by ANSYS depending on the physics of the problem. If **SOLCONTROL,OFF**, defaults to OFF.

## Notes

See **NSUBST** for an alternative input.

Use consistent values for *DTIME* and *TIME* (**TIME**). For example, using 0.9 for *DTIME* and 1.0 for *TIME* results in one time step because 1.0 (*TIME*) is divisible by .9 (*DTIME*) at most once. If your intent is to load in 10 increments over a time span of 1.0, then use 0.1 for *DTIME* and 1.0 for *TIME*.

ANSYS calculates the initial incremental time so that  $(EndingTime - StartingTime) / \mathbf{DELTIM}$  is an integer, which may affect the initial incremental time that you specify. For example, if the starting time is 0, the ending time is 1, and the initial incremental time is 0.4, ANSYS rounds to the nearest integer and adjusts the time to 0.33333.

For the sake of solution efficiency, ANSYS recommends specifying values for all fields of this command.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Basic**  
**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step**

## DEMORPH, ELEM, DIMN, RSMHXY

**Move nodes in selected elements to conform to structural displacements.**

[PREP7: Morphing](#)

MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

### **ELEM**

Non-structural elements to which mesh movement (morph) applies. If ALL, apply morphing to all selected elements [**ESEL**]. If ELEM = P, graphical picking is enabled. A component may be substituted for ELEM.

### **DIMN**

Problem dimensionality. Use "2" for a 2-D problem and "3" for a 3-D problem (no default).

### **RSMHXY**

Remesh flag option:

- 0**  
Remesh the selected non-structural regions only if mesh morphing fails.
- 1**  
Remesh the selected non-structural regions and bypass mesh morphing.
- 2**  
Perform mesh morphing only and do not remesh.

## Notes

The selected elements should include only non-structural regions adjacent to structural regions. The exterior nodes of the selected elements will usually be on the boundary of the region which will have node positions displaced. For  $DIMN = 2$ , elements must lie on a flat plane. The **DEMORPH** command requires a single domain grouping of elements be provided (multiple domains of elements are not permitted). Exterior nodes will be assumed fixed (no nodes will be morphed) unless they coincide with structural nodes having nonzero displacements.

Nodes in the structural regions move in accordance with computed displacements. Displacements from a structural analysis must be in the database prior to issuing **DEMORPH**.

By default ( $RMSHKY = 0$ ), **DEMORPH** will remesh the selected non-structural regions entirely if a satisfactory morphed mesh cannot be provided.

If boundary conditions and loads are applied directly to nodes and elements, the **DEMORPH** command requires that these be removed before remeshing can take place.

Exercise care with initial conditions defined by the **IC** command. Before a structural analysis is performed for a sequentially coupled analysis, the **DEMORPH** command requires that initial conditions be removed from all null element type nodes in the non-structural regions. Use **ICDELE** to delete the initial conditions.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Phys Morphing>Elements

**DERIV**, *IR*, *IY*, *IX*, --, *Name*, --, --, *FACTA*

**Differentiates a variable.**

POST26: Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

***IY*, *IX***

Reference numbers of variables to be operated on. *IY* is differentiated with respect to *IX*.

--

Unused field.

***Name***

Thirty-two character name for identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --

Unused fields.

***FACTA***

Scaling factor (positive or negative) applied as shown below (defaults to 1.0).

## Notes

Differentiates variables according to the operation:

$$IR = FACTA \times d(IY)/d(IX)$$

## Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Derivative



**DESIZE**, *MINL*, *MINH*, *MXEL*, *ANGL*, *ANGH*, *EDGMN*, *EDGMX*, *ADJF*, *ADJM***Controls default element sizes.**

PREP7: Meshing

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**MINL**

Minimum number of elements that will be attached to a line when using lower-order elements (defaults to 3 elements per line). If *MINL* = DEFA, all arguments will be set back to default values. If *MINL* = STAT, list status of command (Including on/off status). If *MINL* = OFF, deactivate default element sizing. If *MINL* = ON, reactivate default element sizing.

**MINH**

Minimum number of elements that will be attached to a line when using higher-order elements. Defaults to 2 elements per line.

**MXEL**

Maximum number of elements that will be attached to a single line (lower or higher-order elements). Defaults to 15 elements per line for h-elements. To deactivate this limit, specify a large number (such as 9999).

**ANGL**

Maximum spanned angle per lower-order element for curved lines. Defaults to 15 degrees per element.

**ANGH**

Maximum spanned angle per higher-order element for curved lines. Defaults to 28 degrees per element.

**EDGMN**

Minimum element edge length. Defaults to no minimum edge length. The *MINL* or *MINH* argument can override this value.

**EDGMX**

Maximum element edge length. Defaults to no maximum edge length. The *MXEL* argument can override this value.

**ADJF**

Target aspect ratio for adjacent line. Used only when free meshing. Defaults to 1.0, which attempts to create equal-sided h-elements.

**ADJM**

Target aspect ratio for adjacent line. Used only when map meshing. Defaults to 4.0, which attempts to create rectangular h-elements.

**Command Default**

Default settings as described for each argument are used.

**Notes**

**DESIZE** settings are usually used for mapped meshing. They are also used for free meshing if SmartSizing is turned off [**SMRTSIZE**,OFF], which is the default. Even when SmartSizing is on, some **DESIZE** settings (such as maximum and minimum element edge length) can affect free mesh density. The default settings of the **DESIZE** command are used only when no other element size specifications [**KESIZE**, **LESIZE**, **ESIZE**] exist for a certain line.

This command is also valid for [rezoning](#).

## Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Other

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Default Element Sizing Options

---

**DESOL**, *ELEM*, *NODE*, *Item*, *Comp*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

**Defines or modifies solution results at a node of an element.**

POST1: Set Up

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### ***ELEM***

Element number for which results are defined or modified. If ALL, apply to all selected elements [**ESEL**].

### ***NODE***

Node of element (actual node number, not the position) to which results are specified. If ALL, specify results for all selected nodes [**NSEL**] of element. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

### ***Item***

Label identifying results. Valid item labels are shown in [Table 143: DESOL - Valid Item and Component Labels \(p. 375\)](#) below. Some items also require a component label (*Comp*).

### ***Comp***

Component of the item (if required); see [Table 143: DESOL - Valid Item and Component Labels \(p. 375\)](#).

### ***V1***

Value (in the element coordinate system) assigned to the database item (and component, if any). If zero, a zero value will be assigned. If blank, value remains unchanged.

### ***V2, V3, V4, . . . , V6***

Additional values (if any) assigned to the remaining components (in the order corresponding to the *Comp* list shown below) for the specified *Item* (starting from the specified *Comp* label and proceeding to the right).

## Notes

The **DESOL** command defines or modifies solution results in the database at a node of an area or volume element. For example, **DESOL,35,50,S,X,1000,2000,1000** assigns values 1000, 2000, and 1000 to SX, SY, and SZ (respectively) of node 50 of element 35.

The settings of the POST1 **FORCE**, **SHELL**, and **LAYER** commands, if applicable, further specify which database items are affected.

For layered composite shells, specify the current element layer (**LAYER**) *before* issuing the **DESOL** command.

All data is stored in the solution coordinate system but is displayed in the results coordinate system (**RSYS**). To list the current results, use the **PRESOL** command.

Modified solution results are not saved automatically. To save separate records of modified results, use either the **RAPPND** or **LCWRITE** command.

Result items are available depending on element type; check the individual element for availability. Valid item and component labels for element results are:

**Table 143 DESOL - Valid Item and Component Labels**

Item	Comp	Description
ELEM		Element number.
S	X, Y, Z, XY, YZ, XZ	Component stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
TG	X, Y, Z	Component thermal gradient.
TF	X, Y, Z	Component thermal flux.
PG	X, Y, Z	Component pressure gradient.
EF	X, Y, Z	Component electric field.
D	X, Y, Z	Component electric flux density.
H	X, Y, Z	Component magnetic field intensity.
B	X, Y, Z	Component magnetic flux density.
FMAG	X, Y, Z	Component electromagnetic force.
F	X, Y, Z	X, Y, or Z structural force.
M	X, Y, Z	X, Y, or Z structural moment.
HEAT		Heat flow.
FLOW		Fluid flow.
AMPS		Current flow.
FLUX		Magnetic flux.
CSG	X, Y, Z	X, Y, or Z magnetic current segment component.

## Menu Paths

**Main Menu>General Postproc>Define/Modify>Elem Results**

**DETAB**, *ELEM*, *Lab*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

**Modifies element table results in the database.**

[POST1: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***ELEM***

Element for which results are to be modified. If ALL, modify all selected elements [**ESEL**] results. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*.

***Lab***

Label identifying results. Valid labels are as defined with the **ETABLE** command. Issue **ETABLE,STAT** to display labels and values.

***V1***

Value assigned to this element table result in the database. If zero, a zero value will be assigned. If blank, value remains unchanged.

***V2, V3, V4, . . . , V6***

Additional values (if any) assigned to consecutive element table columns.

**Notes**

Modifies element table [**ETABLE**] results in the database. For example, **DETAB,35,ABC,1000,2000,1000** assigns 1000, 2000, and 1000 to the first three table columns starting with label ABC for element 35. Use the **PRETAB** command to list the current results. After deleting a column of data using **ETABLE,Lab,ERASE**, the remaining columns of data are not shifted to compress the empty slot. Therefore, the user must allocate null (blank) values for *V1, V2...V6* for any ETABLE entries which have been deleted by issuing **ETABLE,Lab,ERASE**. All data are stored in the solution coordinate system but will be displayed in the results coordinate system [**RSYS**].

**Menu Paths**

**Main Menu>General Postproc>Define/Modify>ElemTabl Data**

***/DEVDISP, Label, KEY***

**Controls graphics device options.**

[DISPLAY: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Label***

Device function label:

**BBOX**

Disables display information sorting for PowerGraphics displays. When activated (*KEY* = 1 or ON), model rotations and replots are performed without recalculating edge and surface data. This will speed up the rotation (especially for 2-D displays) of large models, although the display information will not be resolved as quickly (you will see a bounding box instead of the model during dynamic rotations). The default is OFF (*KEY* = 0).

**DITHER**

Dithering. When turned on (default), dithering smooths transitions in color intensity. Applies only to Z-buffered displays.

**FONT**

Font selection for the ANSYS graphics window. When *Label* = FONT, the command format is: **/DEVDISP, FONT, KEY, Val1, Val2, Val3, Val4, Val5, Val6**, where *KEY* determines the type of font being controlled, and values 1 through 6 control various font parameters. *Note that these values are device specific; using the same command input file [/INPUT] on different machines may yield different results.* The following *KEY* values determine the font information that will be supplied to the appropriate driver (e.g., Postscript, X11, Win32, JPEG, ...):

**KEY = 1**

The command controls the LEGEND (documentation column) font.

**KEY = 2**

The command controls the ENTITY (node and keypoint number) font.

**KEY = 3**

The command controls the ANNOTATION/GRAPH font.

**UNIX:** Values 1 through 4 are used to find a match in the X11 database of font strings. Values 1, 2, and 3 are character strings; value 4 is a nonzero integer:

**Val1**

Family name (e.g., Courier\*New). Substitute an asterisk (\*) for any blank character that appears in a family name. If *Val1* = MENU, all other values are ignored, and a font selection menu appears (GUI must be active).

**Val2**

Weight (e.g., medium)

**Val3**

Slant (e.g., r)

**Val4**

Pixel size (e.g., 14). Note that this value does not affect the annotation fonts (*KEY* = 3). Use the **/TSPEC** command to control the pixel size of your annotation fonts.

**Val5**

unused

**Val6**

unused

**PC:** The values are encoded in a PC logical font structure. Value 1 is a character string, and the remaining values are integers:

**Val1**

Family name (e.g., Courier\*New) Substitute an asterisk (\*) for any blank character that appears in a family name. If *Val1* = MENU, all other values are ignored and a font selection menu appears (GUI must be active). A value containing all blank characters causes ANSYS to use the first available resource it finds.

**Val2**

Weight (0 - 1000)

**Val3**

Orientation (in tenths of a degree)

**Va14**

Height (in logical units) Note that this value does not affect the annotation fonts (*KEY* =3). Use the **/TSP** command to control the height of your annotation fonts.

**Va15**

Width (in logical units)

**Va16**

Italics (0 = OFF, 1 = ON)

**TEXT**

Text size specification for the ANSYS Graphics window. When Label = TEXT, the command format is: **/DEVDISP,TEXT,KEY,PERCENT**, where *KEY* determines the type of text being controlled (1 for LEGEND, and 2 for ENTITY), and *PERCENT* specifies the new text size as a percent of the default text size. If *PERCENT* = 100, the new text size is precisely the default size. If *PERCENT* = 200, the new text size is twice the default text size.

**KEY**

Control key:

**OFF or 0**

Turns specified function off.

**ON or 1**

Turns specified function on.

**Command Default**

Dithering on.

**Menu Paths**

It is part of the DISPLAY program.

---

**/DEVICE, Label, KEY****Controls graphics device options.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Label**

Device function label:

**BBOX**

Bounding box mode. For PowerGraphics plots involving elements with **/SHOW,x11** and **/SHOW,win32**, ANSYS generally displays dynamic rotations faster. If *KEY* = 1 (ON), then a bounding box (not the elements) encompassing the model is displayed and rotated, rather than the element outlines (ON is default in preprocessing). When *KEY* = 0 (OFF), then dynamic rotations may be slower (ANSYS redraws the element outlines) for plots involving elements with **/SHOW,x11** and **/SHOW,win32**. OFF is default in postprocessing. This command is ignored if **/EDGE,WN,1** is set for any WN. This is ignored in POST1 and SOLUTION plots.

For any PowerGraphics plots involving elements, regardless of **/SHOW** settings, plots will generally be displayed faster.

**VECTOR**

Vector mode. In vector mode, areas, volumes, elements, and postprocessing display geometries are shown as outlines (wireframes). When vector mode is off (default), these entities are shown filled with color.

**DITHER**

When dithering is turned on (default), color intensity transitions are smoothed. This selection applies only to smooth-shaded images, i.e., Z-buffered [**TYPE**], or raster plots with Gouraud or Phong shading [**SHADE**].

**ANIM**

Select the animation type used on 2-D devices on the PC platform. A *KEY* value of BMP (or 0) sets animation mode to ANSYS Animation Controller (default). A *KEY* value of AVI (or 2) sets animation mode to AVI movie player file.

**FONT**

Font selection for the ANSYS graphics window. When *Label* = FONT, the command format is: **/DEVICE, FONT, KEY, Val1, Val2, Val3, Val4, Val5, Val6** where *KEY* determines the type of font being controlled, and values 1 through 6 control various font parameters. *Note that these values are device specific; using the same command input file [INPUT] on different machines may yield different results.* The following *KEY* values determine the font information that will be supplied to the appropriate driver (e.g., Postscript, X11, Win32, JPEG, ...):

**KEY = 1**

The command controls the LEGEND (documentation column) font.

**KEY = 2**

The command controls the ENTITY (node and keypoint number) font.

**KEY = 3**

The command controls the ANNOTATION/GRAPH font.

**UNIX:** Values 1 through 4 are used to find a match in the X11 database of font strings. Values 1, 2, and 3 are character strings; value 4 is a nonzero integer:

**Val1**

Family name (e.g., Courier). If *Val1* = MENU, all other values are ignored and a font selection menu appears (GUI must be active).

**Val2**

Weight (e.g., medium)

**Val3**

Slant (e.g., r)

**Val4**

Pixel size (e.g., 14). Note that this value does not affect the annotation fonts (*KEY* = 3). Use the **/TSPEC** command for annotation font size.

**Val5**

unused

**Val6**

unused

**PC:** The values are encoded in a PC logical font structure. Value 1 is a character string, and the remaining values are integers:

**Val1**

Family name (e.g., Courier\*New) Substitute an asterisk (\*) for any blank character that appears in a family name. If *Val1* = MENU, all other values are ignored and a font selection menu appears (GUI must be active). When this value is blank ANSYS uses the first available resource it finds.

**Val2**

Weight (0 - 1000)

**Val3**

Orientation (in tenths of a degree)

**Val4**

Height (in logical units)

**Val5**

Width (in logical units)

**Val6**

Italics (0 = OFF, 1 = ON)

**TEXT**

Text size specification for the ANSYS Graphics window. Using this label with the **/DEVICE** command requires the following form: **/DEVICE,TEXT,KEY,PERCENT**. *KEY* = 1 for LEGEND fonts; *KEY* = 2 for ENTITY fonts. *PERCENT* specifies the new text size as a percent of the default text size. If *PERCENT* = 100, the new text size is precisely the default size. If *PERCENT* = 200, the new text size is twice the default text size.

**KEY**

Control key:

**OFF or 0**

Turns specified function off.

**ON or 1**

Turns specified function on or designates the LEGEND font.

**2**

Designates the ENTITY font.

**3**

Designates the ANNOTATION/GRAPH font.

## Command Default

Vector mode off (i.e., raster mode); dithering on.

## Notes

This command is valid in any processor.

The **/DEVICE,BBOX** command is ignored in POST1 and SOLUTION plots. Also, the elements are displayed and rotated if you use **/DEVICE,BBOX,ON** and **/EDGE,WN,1,ANGLE** (effectively ignoring the BBOX option).

## Menu Paths

**Utility Menu>PlotCtrls>Device Options**

**Utility Menu>PlotCtrls>Font Controls>Entity Font**

**Utility Menu>PlotCtrls>Font Controls>Legend Font**



---

**Utility Menu>PlotCtrls>Redirect Plots>To PSCR File**


---

**/DFLAB**, *DOF*, *DispLab*, *ForceLab*

**Changes degree-of-freedom labels for user custom elements.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

## Argument Descriptions

### *DOF*

Number between 1 and 32 indicating which degree of freedom is to have its labels changed. For a list of these quantities, see the degree-of-freedom table in the `echprm.inc` file. The first few quantities follow:

- 1 = UX,FX
- 2 = UY,FY
- 3 = UZ,FZ
- 4 = ROTX,MX

### *DispLab*

New label (four-character maximum) for the displacement label. The prior label is no longer valid.

### *ForceLab*

New label (four-character maximum) for the force label for this degree of freedom. The prior label is no longer valid.

## Notes

The **/DFLAB** command is rarely used. Use it if you are writing a custom element and want to use degrees of freedom that are not part of the standard element set.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**DIG**, *NODE1*, *NODE2*, *NINC*

**Digitizes nodes to a surface.**

[PREP7: Digitizing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NODE1*, *NODE2*, *NINC*

Digitize nodes *NODE1* through *NODE2* in steps of *NINC*. *NODE2* defaults to *NODE1* and *NINC* defaults to 1.

## Command Default

No surface digitizing.

## Notes

Digitizes nodes to the surface defined by the **DSURF** command. The nodes indicated must be digitized from the tablet after this command is given. The program must be in the interactive mode and the graphics terminal show option [**/SHOW**] must be active. The global Cartesian coordinates of the nodes are stored.

## Menu Paths

**Main Menu>Preprocessor>Create>Nodes>Digitize Nodes>Digitize Nodes**

---

## DIGIT

Specifies "Node digitizing" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>Preprocessor>Digitize Module**

---

**\*DIM**, *Par*, *Type*, *IMAX*, *JMAX*, *KMAX*, *Var1*, *Var2*, *Var3*, *CSYSID*

Defines an array parameter and its dimensions.

[APDL: Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Par*

Name of parameter to be dimensioned. See **\*SET** for name restrictions.

### *Type*

Array type:

#### **ARRAY** --

Arrays are similar to standard FORTRAN arrays (indices are integers) (default). Index numbers for the rows, columns, and planes are sequential values beginning with one. Used for 1-, 2-, or 3-D arrays.

#### **ARR4** --

Same as ARRAY, but used to specify 4-D arrays.

**ARR5 --**

Same as ARRAY, but used to specify 5-D arrays.

**CHAR --**

Array entries are character strings (up to 8 characters each). Index numbers for rows, columns, and planes are sequential values beginning with one.

**TABLE --**

Array indices are real (non-integer) numbers which must be defined when filling the table. Index numbers for the rows and columns are stored in the zero column and row "array elements" and are initially assigned a near-zero value. Index numbers must be in ascending order and are used only for retrieving an array element. When retrieving an array element with a real index that does not match a specified index, linear interpolation is done among the nearest indices and the corresponding array element values [**\*SET**]. Used for 1-, 2-, or 3-D tables.

**TAB4 --**

Same as TABLE, but used to specify 4-D tables.

**TAB5 --**

Same as TABLE, but used to specify 5-D tables.

**STRING --**

Array entries are character strings (up to IMAX each). Index numbers for columns and planes are sequential values beginning with 1. Row index is character position in string.

**IMAX**

Extent of first dimension (row) (For *Type* = STRING, *IMAX* is rounded up to the next multiple of eight and has a limit of 256). Defaults to 1.

**JMAX**

Extent of second dimension (column). Defaults to 1.

**KMAX**

Extent of third dimension (plane). Defaults to 1.

**Var1**

Variable name corresponding to the first dimension (row) for *Type* = TABLE. Defaults to Row.

**Var2**

Variable name corresponding to the second dimension (column) for *Type* = TABLE. Defaults to Column.

**Var3**

Variable name corresponding to the third dimension (plane) for *Type* = TABLE. Defaults to Plane.

**CSYSID**

An integer corresponding to the coordinate system ID Number.

## Notes

Up to three dimensions (row, column, and plane) may be defined using ARRAY and TABLE. Use ARR4, ARR5, TAB4, and TAB5 to define up to five dimensions (row, column, plane, book, and shelf). An index number is associated with each row, column, and plane. For array and table type parameters, element values are initialized to zero. For character and string parameters, element values are initialized to (blank). A defined parameter must be deleted [**\*SET**] before its dimensions can be changed. Scalar (single valued) parameters should not be dimensioned. **\*DIM,A,,3** defines a vector array with elements A(1), A(2), and A(3). **\*DIM,B,,2,3** defines a 2x3 array with elements B(1,1), B(2,1), B(1,2), B(2,2), B(1,3), and B(2,3). Use **\*STATUS,Par** to display elements of array *Par*. You can write formatted data files (tabular formatting) from data held in arrays through the **\*VWRITE** command.

If you use table parameters to define boundary conditions, then *Var1*, *Var2*, and/or *Var3* can either specify a primary variable (listed in [Table 144: \\*DIM - Primary Variables \(p. 384\)](#)) or can be an independent parameter. If specifying an independent parameter, then you must define an additional table for the independent parameter. The additional table must have the same name as the independent parameter and may be a function of one or more primary variables or another independent parameter. All independent parameters must relate to a primary variable.

Tabular load arrays can be defined in both global Cartesian (default) or local (see below) coordinate systems by specifying *CSYSID*, as defined in **LOCAL**. For batch operations, you must specify your coordinate system first.

The following constraints apply when you specify a local coordinate system for your tabular loads:

Only Cartesian, cylindrical and spherical coordinate systems are supported

Angle values for Y in cylindrical or spherical coordinate systems must be input in degrees and must be positive values between 0 and 360 degrees ( $0 \leq Y \leq 360$ )

Angle values for Z in spherical coordinate system must be input in degrees and must be positive values between -90 and +90 ( $-90 \leq Z \leq 90$ )

If you are specifying a 4- or 5-D array or table, four additional fields (*LMAX*, *MMAX*, *Var4*, and *Var5*) are available. Thus, for a 4-D table, the command syntax would be:

```
*DIM, Par, Type, IMAX, JMAX, KMAX, LMAX, Var1, Var2, Var3, Var4, CSYSID
```

For a 5-D table, the command syntax would be:

```
*DIM, Par, Type, IMAX, JMAX, KMAX, LMAX, MMAX, Var1, Var2, Var3, Var4, Var5, CSYSID
```

You cannot create or edit 4- or 5-D arrays or tables using the GUI.

See [Array Parameters](#) for a detailed discussion on and examples for using array parameters.

**Table 144 \*DIM - Primary Variables**

Primary Variable	Label for <i>Var1</i> , <i>Var2</i> , <i>Var3</i>
Time	TIME
Frequency	FREQ
X-coordinate location	X
Y-coordinate location	Y
Z-coordinate location	Z
Temperature	TEMP
Velocity	VELOCITY
Pressure	PRESSURE
Cyclic sector number	SECTOR

**Note**

The X, Y, and Z coordinate locations listed above are valid in global Cartesian, or local (Cartesian, cylindrical and spherical) coordinate systems. The VELOCITY label is applicable only to the calculated fluid velocity in element [FLUID116](#).

When using PRESSURE as a primary variable, the underlying element must have the pressure DOF associated with it. For example, if you use [FLUID141](#) or [FLUID142](#), apply **SF,ALL,CONV,%presh%,20.0** to apply a tabular film coefficient that will be evaluated correctly, based on the pressure. However, if the underlying element is [SURF151](#) or [SURF152](#), the film coefficient will not change because there is no underlying pressure.

The frequency label (FREQ) is valid for harmonic analyses only.

If you use table parameters to define boundary conditions, the table names (*Par*) must not exceed 32 characters.

In thermal analyses, if you apply tabular loads as a function of temperature but the rest of the model is linear (e.g., includes no temperature-dependent material properties or radiation), you should turn on Newton-Raphson iterations (**NROPT,FULL**) to evaluate the temperature-dependent tabular boundary conditions correctly.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>Parameters>Array Parameters>Define/Edit**

---

**/DIRECTORY**, *StrArray*, *FileName*, *Ext*, *Dir*

**Put the file names in the current directory into a string parameter array.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Argument Descriptions*****StrArray***

Name of the "string array" parameter which will hold the returned values. String array parameters are similar to character arrays, but each array element can be as long as 128 characters. If the string parameter does not exist, it will be created. The array will be created as: \*DIM,*StrArray*,STRING,64,2,numFileName

***FileName***

File name (64 characters maximum). Only files matching this name will be returned. The *FileName* ALL may match any file name.

***Ext***

File name extension (8 characters maximum). Only files with an extension matching this name will be returned. A blank or ALL will match any extension.

## Directory

The directory in which the files reside. The default is the current working directory.

## Notes

The **/DIRECTORY** command gets the file names in the current directory and puts them into a string parameter array. Each file will be included in the array as a name-extension pair.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## DISPLAY

Specifies "Display settings" as the subsequent status topic.

[POST1: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>General Postproc>Plot Results**

---

**/DIST**, *WN*, *DVAL*, *KFACT*

Specifies the viewing distance for magnifications and perspective.

[GRAPHICS: Views](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *WN*

Window number (or ALL) to which command applies (defaults to 1).

### *DVAL*

Distance along the view line from the observer to the focus point (defaults to value producing full-window display). Distances "too close" to the object will produce excessive magnifications. If *DVAL* = AUTO, zero, or blank, the program will calculate the distance automatically. If *DVAL* = USER, the distance of last display will be used (useful when last display automatically calculated distance).

### *KFACT*

*DVAL* interpretation key:

0

Interpret numerical *DVAL* values as described above.

1

Interpret *DVAL* as a multiplier on the current distance (*DVAL* of 2 gives twice the current distance; 0.5 gives half the current distance, etc.).

### Command Default

Distance is automatically calculated to produce full window magnification.

### Notes

The scale factor is relative to the window shape. For example, for objects centered in a square window and with parallel projection (no perspective), a distance of  $\ell / 2$  (+10%) produces a full window magnification, where  $\ell$  is the largest in-plane vertical or horizontal dimension. See also **/AUTO** and **/USER** commands.

This command is valid in any processor.

### Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Utility Menu>PlotCtrls>View Settings>Magnification

**DJ**, *ELEM*, *LABEL*, *VALUE*

**Specifies boundary conditions on the components of relative motion of a joint element.**

[SOLUTION: FE Constraints](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

#### ***ELEM***

Element number or ALL to be specified.

#### ***LABEL***

Valid labels are:

#### **UX**

Displacement in local x direction.

#### **UY**

Displacement in local y direction.

#### **UZ**

Displacement in local z direction.

#### **ROTX**

Rotation about local x axis.

#### **ROTY**

Rotation about local y axis.

#### **ROTZ**

Rotation about local y axis.

**VELX**

Linear velocity in local x direction.

**VELY**

Linear velocity in local y direction.

**VELZ**

Linear velocity in local z direction.

**OMGX**

Angular velocity in local x direction.

**OMGY**

Angular velocity in local y direction.

**OMGZ**

Angular velocity in local z direction.

**ACCX**

Linear acceleration in local x direction.

**ACCY**

Linear acceleration in local y direction.

**ACCZ**

Linear acceleration in local z direction.

**DMGX**

Angular acceleration in local x direction.

**DMGY**

Angular acceleration in local y direction.

**DMGZ**

Angular acceleration in local z direction.

**VALUE**

Value of the label.

## Notes

This command is valid for [MPC184](#) joint elements. See [DJDELE](#) for information on deleting boundary conditions applied with the **DJ** command.

You can apply only one displacement, velocity, or acceleration load at any relative degree of freedom. If multiple loads are specified, the last applied load overrides the previous ones. For example, the following commands apply loads to element 100:

```
D,100,UX,Value  
D,100,VELX,Value
```

In this case, the velocity load (VELX) applied in the last command will override the displacement load (UX).

Tabular boundary conditions (*VALUE = %tabname%*) can be used.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Joint Elems**  
**Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Joint Elems**



**DJDELE**, *ELEM*, *LAB*

**Deletes boundary conditions on the components of relative motion of a joint element.**

SOLUTION: FE Constraints

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

***ELEM***

Element number or ALL. ALL (or leaving this field blank) will delete all joint element boundary conditions specified by *LAB*.

***LAB***

Valid labels are:

**UX**

Displacement in local x direction.

**UY**

Displacement in local y direction.

**UZ**

Displacement in local z direction.

**ROTX**

Rotation about local x axis.

**ROTY**

Rotation about local y axis.

**ROTZ**

Rotation about local z axis.

**VELX**

Linear velocity in local x direction.

**VELY**

Linear velocity in local y direction.

**VELZ**

Linear velocity in local z direction.

**OMGX**

Angular velocity in local x direction.

**OMGY**

Angular velocity in local y direction.

**OMGZ**

Angular velocity in local z direction.

**ACCX**

Linear acceleration in local x direction.

**ACCY**

Linear acceleration in local y direction.

**ACCZ**

Linear acceleration in local z direction.

**DMGX**

Angular acceleration in local x direction.

**DMGY**

Angular acceleration in local y direction.

**DMGZ**

Angular acceleration in local z direction.

**ALL, or (blank)**

Delete all applied boundary conditions.

## Notes

This command is valid for [MPC184](#) joint elements. See [DJ](#) for information on specifying boundary conditions on the components of relative motion of a joint element.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On Joint Elems**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Joint Elems**

**Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On Joint Elems**

**Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Joint Elems**

---

## DJLIST, *Elem*

**Lists boundary conditions applied to joint elements.**

[SOLUTION:FE Constraints](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

***Elem***

Element number or ALL (or blank). Lists joint element boundary conditions on the specified element(s).

## Notes

This command is valid for [MPC184](#) joint elements. See [DJ](#) for information on specifying boundary conditions on joint elements.

## Menu Paths

**Utility Menu>List>Loads>Joint Element DOF Constraints>On Picked Element**

---

**DK**, *KPOI*, *Lab*, *VALUE*, *VALUE2*, *KEXPND*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

**Defines DOF constraints at keypoints.**

**SOLUTION: Solid Constraints**

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### ***KPOI***

Keypoint at which constraint is to be specified. If ALL, apply to all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

### ***Lab***

Valid degree of freedom label. If ALL, use all appropriate labels *except* HDSP. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping); HDSP (hydrostatic pressure). Thermal labels: TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature). Electric labels: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials).

### ***VALUE***

Degree of freedom value or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent signs (%) (e.g., **DK,NODE,TEMP,%tablename%**). Use the **\*DIM** command to define a table.

### ***VALUE2***

Second degree of freedom value (if any). If the analysis type and the degree of freedom allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

### ***KEXPND***

Expansion key:

**0**

Constraint applies only to the node at this keypoint.

**1**

Flags this keypoint for constraint expansion.

### ***Lab2, Lab3, Lab4, . . . , Lab6***

Additional degree of freedom labels. The same values are applied to the keypoints for these labels.

## **Notes**

A keypoint may be flagged using *KEXPND* to allow its constraints to be expanded to nodes on the attached solid model entities having similarly flagged keypoint constraints. Constraints are transferred from keypoints to nodes with the **DTRAN** or **SBCTRAN** commands. The expansion uses interpolation to apply constraints to the nodes on the lines between flagged keypoints. If all keypoints of an area or volume region are flagged *and the constraints (label and values) are equal*, the constraints are applied to the interior nodes of the region. See the **D** command for a description of nodal constraints.

Tabular boundary conditions (*VALUE* = %tablename%) are available only for the following degree of freedom labels: Electric (VOLT), structural (UX, UY, UZ, ROTX, ROTY, ROTZ), and temperature (TEMP, TBOT, TE2, TE3, . . . , TTOP).

Constraints specified by the **DK** command can conflict with other specified constraints. See [Resolution of Conflicting Constraint Specifications](#) in the *Basic Analysis Guide* for details.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>BasePSD>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>BasePSD>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Keypoints  
 Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>On Keypoints  
 Main Menu>Solution>Constraints>Apply>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>BasePSD>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Keypoints  
 Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>BasePSD>On Keypoints  
 Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Keypoints

## DKDELE, *KPOI*, *Lab*

Deletes DOF constraints at a keypoint.

[SOLUTION: Solid Constraints](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *KPOI*

Keypoint for which constraint is to be deleted. If ALL, delete for all selected keypoints [[KSEL](#)]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

A component name may also be substituted for *KPOI*.

### *Lab*

Valid degree of freedom label. If ALL, use all appropriate labels. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). FLOTRAN fluid labels: PRES (pressure); VX, VY, or VZ (velocities); ENKE or ENDS (turbulent kinetic energy or turbulent energy dissipation); SP01 through SP06 (multiple species mass fractions) or their user-defined names. Electric label: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials). High-frequency electromagnetic label: AX (Electric Wall or Magnetic Wall boundary condition).

## Notes

Deletes the degree of freedom constraints (and all corresponding finite element constraints) at a keypoint. See the **DDELE** command for details.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All KPs**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>TimeInt>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Displacement>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Velocity>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Keypoints**  
**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>On Keypoints**  
**Main Menu>Solution>Constraints>Delete>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All KPs**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>TimeInt>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Displacement>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Velocity>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Keypoints**

## DKLIST, *KPOI*

Lists the DOF constraints at keypoints.

**SOLUTION: Solid Constraints**

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *KPOI*

List constraints for this keypoint. If ALL (default), list for all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

## Notes

Listing applies to the selected keypoints [**KSEL**] and the selected degree of freedom labels [**DOFSEL**].

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Keypoints

Utility Menu>List>Loads>DOF Constraints>On Picked KPs

**DL**, *LINE*, *AREA*, *Lab*, *Value1*, *Value2*

**Defines DOF constraints on lines.**

[SOLUTION: Solid Constraints](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### **LINE**

Line at which constraints are to be specified. If ALL, apply to all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*

### **AREA**

Area containing line. The normal to the symmetry or antisymmetry surface is assumed to lie on this area. Defaults to the lowest numbered selected area containing the line number.

### **Lab**

Symmetry label (see [2 \(p. 395\)](#)):

#### **SYMM**

Generate symmetry constraints for non-FLOTRAN models.

#### **ASYM**

Generate antisymmetry constraints for non-FLOTRAN models.

ANSYS DOF labels (see [3 \(p. 395\)](#), [4 \(p. 396\)](#), and [5 \(p. 396\)](#)):

#### **UX**

Displacement in X direction.

#### **UY**

Displacement in Y direction.

#### **UZ**

Displacement in Z direction.

#### **ROTX**

Rotation about X axis.

#### **ROTY**

Rotation about Y axis.

#### **ROTZ**

Rotation about Z axis.

#### **HDSP**

Hydrostatic pressure.

#### **WARP**

Warping magnitude.

**TEMP, TBOT, TE2, TE3, . . . , TTOP**

Temperature

**VOLT**

Electric scalar potential.

**AX**

Magnetic vector potential in X direction.

**AY**

Magnetic vector potential in Y direction.

**AZ**

Magnetic vector potential in Z direction.

**ALL**

Applies all appropriate DOF labels *except* HDSP.

FLOTRAN standard DOF labels (see 3 (p. 395)): VX, VY, VZ, PRES, TEMP, ENKE, ENDS

FLOTRAN Species Labels (see 4 (p. 396)): SP01, SP02, SP03, SP04, SP05, SP06

FLOTRAN Arbitrary Lagrangian-Eulerian formulation Mesh Displacement Labels (see 6 (p. 396)): UX, UY, UZ

**Value1**

Value of DOF (real part) or table name reference on the line. Valid for all DOF labels. To specify a table, enclose the table name in % signs (e.g., **DL,LINE,AREA,TEMP,%tablename%**). Use the **\*DIM** command to define a table.

If *Lab* = ENKE and *Value1* = -1, a FLOTRAN flag is set to indicate a moving wall.

If *Lab* = ENDS and *Value1* = -1, FLOTRAN generalized symmetry conditions are applied. Velocity components are set tangential to the symmetry surface if the ALE formulation is not activated. They are set equal to the mesh velocity if the ALE formulation is activated.

**Value2**

For FLOTRAN DOFs:

**0**

Values are applied only to nodes within the line.

**1**

Values are applied to the endpoints of the line as well as to the internal nodes (see 3 (p. 395)).

For VOLT DOFs:

Actual value of the imaginary component of the degree of freedom.

**Notes**

1. You can transfer constraints from lines to nodes with the **DTRAN** or **SBCTRAN** commands. See the **DK** command for information about generating other constraints at lines.
2. Symmetry and antisymmetry constraints are generated as described on the **DSYM** command.
3. For the velocity DOF (VX, VY, VZ), a zero value will override a nonzero value at the intersection of two lines.

4. You can use the **MSSPEC** command to change FLOTRAN species labels to user-defined labels. You must define these labels with the **MSSPEC** command before using them on the **DL** command.
5. Setting *Lab* = VOLT and *Value1* = 0 applies the J-normal boundary condition (current density vector (J) flows normal to the line). No input is required for the J-parallel condition because it is the natural boundary condition.
6. Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following degree of freedom labels: Electric (VOLT), FLOTRAN (UX, UY, UZ, PRES, VX, VY, VZ, ENKE, ENDS, TEMP, SP01, SP02, SP03, SP04, SP05, and SP06); Structural (UX, UY, UZ, ROTX, ROTY, ROTZ), and temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP).
7. Constraints specified by the **DL** command can conflict with other specified constraints. See [Resolution of Conflicting Constraint Specifications](#) in the *Basic Analysis Guide* for details.
8. This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PerfEC>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'I>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>...with Area**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>...with Area**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Lines**  
**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>...with Area**  
**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>On Lines**  
**Main Menu>Solution>Constraints>Apply>...with Area**  
**Main Menu>Solution>Constraints>Apply>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PerfEC>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'I>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>...with Area**  
**Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>...with Area**  
**Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Lines**



**DLDELE**, *LINE*, *Lab*

**Deletes DOF constraints on a line.**

SOLUTION: Solid Constraints

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**LINE**

Line for which constraints are to be deleted. If ALL, delete for all selected lines [LSEL]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*

**Lab**

Constraint label:

**ALL**

All constraints.

**SYMM**

Symmetry constraints.

**ASYM**

Antisymmetry constraints.

**UX**

Displacement in X direction.

**UY**

Displacement in Y direction.

**UZ**

Displacement in Z direction.

**ROTX**

Rotation about X axis.

**ROTY**

Rotation about Y axis.

**ROTZ**

Rotation about Z axis.

**WARP**

Warping magnitude.

**VX**

Velocity component in X direction.

**VY**

Velocity component in Y direction.

**VZ**

Velocity component in Z direction.

**PRES**

Pressure.

**TEMP, TBOT, TE2, TE3, . . . , TTOP**

Temperature.

**ENKE**

Turbulent Kinetic Energy.

**ENDS**

Energy Dissipation Rate.

**VOLT**

Electric scalar potential.

**SP01SP06**

Multiple Species Mass Fraction.

**AX**

Magnetic vector potential in X direction.

**AY**

Magnetic vector potential in Y direction.

**AZ**

Magnetic vector potential in Z direction.

**Notes**

Deletes the degree of freedom constraints (and all corresponding finite element constraints) on a line previously specified with the **DL** command. See the **DDELE** command for delete details.

**Warning**

On previously meshed lines, *all* constraints on affected nodes will also be deleted, whether or not they were specified by the **DL** command.

If the multiple species labels have been changed to user-defined labels via the **MSSPEC** command, use the user-defined labels.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PerfEC>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Displacement>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Species>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Turbulence>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Velocity>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Lines**  
**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>On Lines**  
**Main Menu>Solution>Constraints>Delete>On Lines**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All Lines**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PerfEC>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Displacement>On Lines**

**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Species>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Turbulence>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Velocity>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Lines**

---

**DLIST**, *NODE1*, *NODE2*, *NINC*

**Lists DOF constraints.**

SOLUTION: FE Constraints

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

*NODE1*, *NODE2*, *NINC*

List constraints for nodes *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If ALL (default), *NODE2* and *NINC* are ignored and constraints for all selected nodes [**NSEL**] are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

## Notes

Listing applies to the selected nodes [**NSEL**] and the selected degree of freedom labels [**DOFSEL**].

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Nodes

Utility Menu>List>Loads>DOF Constraints>On Picked Nodes

---

**DLLIST**, *LINE*

**Lists DOF constraints on a line.**

SOLUTION: Solid Constraints

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

*LINE*

List constraints for this line. If ALL (default), list for all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*.

## Notes

Lists the degree of freedom constraints on a line previously specified with the **DL** command.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Lines  
Utility Menu>List>Loads>DOF Constraints>On Picked Lines

---

**\*DMAT**, *Matrix, Type, Method, Val1, Val2, Val3, Val4*

Creates a dense matrix.

[APDL: Matrix Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Matrix*

Name used to identify the matrix. Must be specified.

### *Type*

Matrix type:

**D** --

Double precision real values (default).

**Z** --

Complex double precision values.

**I** --

Integer values.

### *Method*

Method used to create the matrix:

**ALLOC** --

Allocate space for a matrix (default).

**COPY** --

Copy an existing matrix.

**IMPORT** --

Import the matrix from a file.

### *Val1, Val2, Val3, Val4*

Additional input. The meaning of *Val1* through *Val4* will vary depending on the specified *Method*. See details below.

**The following *Valx* fields are used with *Method* = ALLOC:**

### *Val1*

Number of rows in the matrix.

### *Val2*

Number of columns in the matrix.

### *Val3*

Memory allocation type:

**INCORE** --

In-core memory allocation (default).

**OUTOFCORE --**

Out-of-core memory allocation.

The following *Valx* fields are used with *Method = COPY*:

**Val1**

Name of the matrix to copy.

**Val2**

Memory allocation type:

**INCORE --**

In-core memory allocation (default).

**OUTOFCORE --**

Out-of-core memory allocation.

The following table describes the *Valx* fields used with *Method = IMPORT*:

<i>Method = IMPORT</i>				
<i>Val1</i>	<i>Val2</i>	<i>Val3</i>	<i>Val4</i>	Description
FULL	File name	Matrix type: STIFF - Stiffness (or conductivity) matrix MASS - Mass (or specific heat) matrix DAMP - Damping matrix RHS - Load vector	(not used)	Import a matrix or load vector from an existing FULL file.
MODE	File name	First mode number	Last mode number	Import a set of modes from an existing MODE file.
CMS	File name	Mode type: NOR - normal modes IRF - inertial relief modes CST - constraint modes	(not used)	Import a modal basis from an existing CMS file.
RST	File name	First data set number	Last data set number	Import degree of freedom results from an existing RST file.
SUB	File name	Matrix type: STIFF - Stiffness (or conductivity) matrix MASS - Mass (or specific heat) matrix DAMP - Damping matrix RHS - Load vector	(not used)	Import a matrix or load vector from an existing SUB file.
HB-MAT	File name	File format: ASCII BINARY	(not used)	Import a matrix from an existing Harwell-Boeing format file.

EMAT	File name	Matrix type: STIFF - Stiffness (or conductivity) matrix MASS - Mass (or specific heat) matrix DAMP - Damping matrix STRESS - Stress-stiffness matrix RHS - Load vector	Element number	Import an element matrix or load vector from an existing EMAT file.
MAT	File name	(not used)	(not used)	Restore from a previous <b>*EXPORT</b> ( <i>FORMAT</i> = MAT) command.
AP-DL	Array parameter name	(not used)	(not used)	Import an existing array parameter.
DMIG	File name	Separator character (default = blank)	(not used)	Import a matrix from an existing Nastran DMIG file.

## Notes

This command allows you to create a dense matrix. To create a sparse matrix, use the **\*SMAT** command. **\*SMAT** is recommended for large matrices obtained from the .FULL or .HBMAT file.

Use the **\*VEC** command to create a vector.

For very large matrices, use the OUTFCORE option (*Method* = ALLOC or COPY) to keep some of the matrix on disk if there is insufficient memory.

## Menu Paths

**This command cannot be accessed from a menu.**

## DMOVE, NODE1, NODE2, NINC

Digitizes nodes on surfaces and along intersections.

[PREP7: Digitizing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NODE1, NODE2, NINC

Digitize nodes *NODE1* through *NODE2* in steps of *NINC*. *NODE2* defaults to *NODE1* and *NINC* defaults to 1.

## Notes

Digitizes nodes on undefined surfaces, warped surfaces, and along intersection lines. Two orthogonal views showing the nodes on a plane in each view are required. No surfaces need be specified. Two coordinates are determined from the second view and the other coordinate is retained from the first view. Use the **DIG** command to first define nodes in one view (as determined from the **DSET** command). Then reset the view and use this command to move the nodes to the proper location.

## Menu Paths

Main Menu>Preprocessor>Create>Nodes>Digitize Nodes>2-View Digitize

**DMPEXT**, *SMODE*, *TMODE*, *Dmpname*, *Freqb*, *Freqe*, *NSTEPS*

**Extracts modal damping coefficients in a specified frequency range.**

[SOLUTION: Analysis Options](#)

MP ME <> <> <> <> <> <> <> <> <> PP <> EME MFS

### ***SMODE***

Source mode number. There is no default for this field; you must enter an integer greater than zero.

### ***TMODE***

Target mode. Defaults to *SMODE*.

### ***Dmpname***

Array parameter name containing the damping results. Defaults to *d\_damp*.

### ***Freqb***

Beginning frequency range (real number greater than zero) or 'EIG' at eigenfrequency of source mode. 'EIG' is valid only if *SMODE* = *TMODE*. Note that EIG must be enclosed in single quotes when this command is used on the command line or in an input file. There is no default for this field; you must enter a value.

### ***Freqe***

End of frequency range. Must be blank for *Freqb* = EIG. Default is *Freqb*.

### ***NSTEPS***

Number of substeps. Defaults to 1.

## Notes

DMPEXT invokes an ANSYS macro that uses modal projection techniques to compute the damping force by the modal velocity of the source mode onto the target mode. From the damping force, damping parameters are extracted. DMPEXT creates an array parameter *Dmpname*, with the following entries in each row:

- response frequency
- modal damping coefficient
- modal squeeze stiffness coefficient
- damping ratio
- squeeze-to-structural stiffness ratio

The macro requires the modal displacements from the file *Jobname.EFL* obtained from the **RMFLVEC** command. In addition, a node component FLUN must exist from all **FLUID136** nodes. The computed damping ratio may be used to specify constant or modal damping by means of the **DMPRAT** or **MDAMP** commands. For Rayleigh damping, use the **ABEXTRACT** command to compute ALPHAD and BETAD damping parameters. See [Thin Film Analysis](#) for more information on thin film analyses.

The macro uses the **LSSOLVE** command to perform two load steps for each frequency. The first load case contains the solution of the source mode excitation and can be used for further postprocessing. Solid model boundary conditions are deleted from the model. In addition, prescribed nodal boundary conditions

are applied to the model. You should carefully check the boundary conditions of your model prior to executing a subsequent analysis.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>ThinFilm>DampExtract>Eigenfrequency**  
**Main Menu>Solution>ThinFilm>DampExtract>Frequency Range**

---

## DMPRAT, RATIO

**Sets a constant damping ratio.**

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### RATIO

Damping ratio (for example, 2% is input as 0.02).

## Command Default

Use damping as defined in the [Structural Analysis Guide](#).

## Notes

Sets a constant damping ratio for use in the mode superposition transient (**ANTYPE,TRANS**) analysis and the spectrum (**ANTYPE,SPECTR**) analysis.

Sets a constant structural damping ratio for use in the harmonic response (**ANTYPE,HARMIC**) analysis (full, reduced, and VT).

Note that for structures with multiple materials, **MP,DMPR** can be used to specify constant structural material damping coefficients for full harmonic analyses. **MP,DMPR** is not applicable for transient or spectrum analyses.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping**  
**Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping**

---



**DNSOL**, *NODE*, *Item*, *Comp*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

**Defines or modifies solution results at a node.**

POST1: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### **NODE**

Node for which results are specified. If ALL, apply to all selected nodes [NSEL]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

#### **Item**

Label identifying results, see [Table 145: DNSOL - Valid Item and Component Labels \(p. 405\)](#). Items also require a component label.

#### **Comp**

Component of the item. Valid component labels are shown [Table 145: DNSOL - Valid Item and Component Labels \(p. 405\)](#) below.

#### **V1, V2, V3, . . . , V6**

Value assigned to result. If zero, a zero value will be assigned. If blank, the value remains unchanged. Additional values (if any) assigned to the remaining components (in the order corresponding to the *Comp* list shown below for the specified *Item* (starting from the specified *Comp* label and proceeding to the right).

## Notes

**DNSOL** can be used only with FULL graphics activated (/GRAPHICS,FULL); it will not work correctly with PowerGraphics activated.

**DNSOL** defines or modifies solution results in the database at a node. For example, **DNSOL,35,U,X,.001,.002,.001** assigns values 0.001, 0.002, and 0.001 to UX, UY, and UZ (respectively) for node 35. All results that are changed in the database, including the nodal degree of freedom results, are available for all subsequent operations. All data is stored in the solution coordinate system, but will be displayed in the results coordinate system [RSYS]. Use the **PRNSOL** command to list the current results.

Data input by **DNSOL** is stored in temporary space and does not replace information in the database. Therefore, data input by this command may be overwritten if a change is made to the selected set of nodes.

Issuing the **DNSOL** command or its GUI equivalent requires you to place the data type (stress/strain) in the element nodal records. To get around this requirement, use the **DESOL** command or equivalent path to add a "dummy" element stress/strain record.

Result items are available depending on element type; check the individual element for availability. Valid item and component labels for element results are:

**Table 145 DNSOL - Valid Item and Component Labels**

Valid Item and Component Labels for Nodal DOF Results		
Item	Comp	Description
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.

**Valid Item and Component Labels for Nodal DOF Results**

<b>Item</b>	<b>Comp</b>	<b>Description</b>
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity.
A	X, Y, Z	X, Y, or Z magnetic vector potential.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.

**Valid Item and Component Labels for Area and Volume Element Results**

<b>Item</b>	<b>Comp</b>	<b>Description</b>
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.

### Valid Item and Component Labels for Area and Volume Element Results

Item	Comp	Description
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
TG	X,Y,Z,SUM	Component thermal gradient or vector sum.
TF	X,Y,Z,SUM	Component thermal flux or vector sum.
PG	X,Y,Z,SUM	Component pressure gradient or vector sum.
EF	X,Y,Z,SUM	Component electric field or vector sum.
D	X,Y,Z,SUM	Component electric flux density or vector sum.
H	X,Y,Z,SUM	Component magnetic field intensity or vector sum.
B	X,Y,Z,SUM	Component magnetic flux density or vector sum.
FMAG	X,Y,Z,SUM	Component electromagnetic force or vector sum.

### Valid Item Labels for FLOTRAN Nodal Results

Item	Description
TTOT	Total temperature.
HFLU	Heat flux.
HFLM	Heat transfer (film) coefficient.
COND	Fluid laminar conductivity.
PCOE	Pressure coefficient.
PTOT	Total (stagnation) pressure.
MACH	Mach number.
STRM	Stream function. (2-D applications only.)
DENS	Fluid density.
VISC	Fluid laminar viscosity.
EVIS	Fluid effective viscosity.
CMUV	Turbulent viscosity coefficient.
ECON	Fluid effective conductivity.
YPLU	$Y^+$ , a turbulent law of the wall parameter.
TAUW	Shear stress at the wall.

1. For [SHELL131](#) and [SHELL132](#) elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, ..., TTOP instead of TEMP.

## Menu Paths

**Main Menu>General Postproc>Define/Modify>Nodal Results**

**\*DO**, *Par*, *IVAL*, *FVAL*, *INC*

**Defines the beginning of a do-loop.**

APDL: Process Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Par*

The name of the scalar parameter to be used as the loop index. See **\*SET** for name restrictions. Any existing parameter of the same name will be redefined. There is no character parameter substitution for the *Par* field.

### *IVAL*, *FVAL*, *INC*

Initially assign *IVAL* to *Par*. Increment *IVAL* by *INC* for each successive loop. If *IVAL* exceeds *FVAL* and *INC* is positive, the loop is not executed. *INC* defaults to 1. Negative increments and non-integer numbers are allowed.

## Notes

The block of commands following the **\*DO** command (up to the **\*ENDDO** command) is executed repeatedly until some loop control is satisfied. Printout is automatically suppressed on all loops after the first (include a **/GOPR** command to restore the printout). The command line loop control (*Par*,*IVAL*,*FVAL*,*INC*) must be input; however, a Use the **\*IF** within the block can also be used to control looping [**\*EXIT**, **\*CYCLE**]. One level of internal file switching is used for each nested **\*DO**. Twenty levels of nested do-loops are allowed.

---

### Note

Do-loops that include **/INPUT**, **\*USE**, or an "Unknown Command" macro, have less nesting available because each of these operations also uses a level of file switching. The **\*DO**, **\*ENDDO**, and any **\*CYCLE** and **\*EXIT** commands for a do-loop must all be read from the same file (or keyboard). You cannot use the **MULTIPRO** or **\*CREATE** commands within a **\*DO**-loop. Picking operations should also not be used within a **\*DO**-loop.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**DOF**, *Lab1, Lab2, Lab3, Lab4, Lab5, Lab6, Lab7, Lab8, Lab9, Lab10*

**Adds degrees of freedom to the current DOF set.**

[PREP7: Element Type](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Lab1, Lab2, Lab3, . . . , Lab10*

Valid labels are: UX, UY, UZ (structural displacements); ROTX, ROTY, ROTZ (structural rotations); TEMP, TBOT, TE2, TE3, . . . , TTOP (temperatures); PRES (pressure); VOLT (voltage); MAG (magnetic scalar potential); AX, AY, AZ (magnetic vector potentials); CURR (current); EMF (electromotive force drop); DELETE.

## Command Default

Use degree of freedom set determined from element types.

## Notes

The degree of freedom (DOF) set for the model is determined from all element types defined. This command may be used to add to the current set. The ALL label may be used on some commands to represent all labels of the current degree of freedom set for the model. Issue the **DOF** command with no arguments to list the current set. Use the DELETE label to delete any previously added DOFs and return to the default DOF set.

## Product Restrictions

Command Option Lab	Available Products
Ux,Uy,Uz,Rx,Ry,Rz	MP ME ST PR PRN <> <> FL <> <> DY PP <> EME MFS
MAG,VOLT	MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>
Ax,Ay,Az	MP <> <> <> <> <> <> <> EM EH <> PP <> EME <>
PRES	MP ME <> <> <> <> <> <> <> <> <> <> PP <> EME MFS
TEMP,TBOT,TE2,TE3	MP ME <> PR PRN <> <> FL <> <> <> PP <> EME MFS
CURR,EMF	MP ME <> <> <> <> <> <> <> EM <> <> PP <> EME MFS
DELETE	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Menu Paths

**Main Menu>Preprocessor>Element Type>Add DOF**

**Main Menu>Preprocessor>Element Type>Remove DOFs**

**DOFSEL**, *Type, Dof1, Dof2, Dof3, Dof4, Dof5, Dof6*

**Selects a DOF label set for reference by other commands.**

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Type*

Label identifying the type of select:

**S**

Select a new set of labels.

**A**

Add labels to the current set.

**U**

Unselect (remove) labels from the current set.

**ALL**

Restore the full set of labels.

**STAT**

Display the current select status.

***DoF1, DoF2, DoF3, . . . , DoF6***

Used only with *Type* = S, A, or U. Valid structural labels: UX, UY, or UZ (displacements); U (UX, UY, and UZ); ROTX, ROTY, or ROTZ (rotations); ROT (ROTX, ROTY, and ROTZ); DISP (U and ROT); HDSP (Hydrostatic pressure). Valid thermal labels: TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature). Valid fluid flow labels: PRES (pressure); VX, VY, or VZ (fluid velocities); V (VX, VY, and VZ); ENKE, ENDS (turbulent kinetic energy, turbulent energy dissipation); EN (ENKE and ENDS turbulent energies) (FLOTRAN). Valid electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Valid magnetic labels: MAG (scalar magnetic potential); AX, AY or AZ (vector magnetic potentials); A (AX, AY and AZ); CURR (current). Valid structural force labels: FX, FY, or FZ (forces); F (FX, FY, and FZ); MX, MY, or MZ (moments); M (MX, MY, and MZ); FORC (F and M); DVOL (fluid mass flow rate). Valid thermal force labels: HEAT, HBOT, HE2, HE3, . . . , HTOP (heat flow). Valid fluid flow force labels: FLOW (fluid flow). Valid electric force labels: AMPS (current flow), CHRG (electric charge). Valid magnetic force labels: FLUX (scalar magnetic flux); CSGX, CSGY, or CSGZ (magnetic current segments); CSG (CSGX, CSGY, and CSGZ).

**Command Default**

Degree of freedom (and the corresponding force) labels are determined from the model.

**Notes**

Selects a degree of freedom label set for reference by other commands. The label set is used on certain commands where ALL is either input in the degree of freedom label field or implied. The active label set has no effect on the solution degrees of freedom. Specified labels which are not active in the model (from the **ET** or **DOF** command) are ignored. As a convenience, a set of force labels corresponding to the degree of freedom labels is also selected. For example, selecting UX also causes FX to be selected (and vice versa). The force label set is used on certain commands where ALL is input in the force label field.

This command is valid in any processor.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Constraints**

**Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Forces**

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Constraints**

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Forces**

**Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Constraints**

**Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Forces**

**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Constraints**

**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Forces**

**DOMEGA**, *DOMGX*, *DOMGY*, *DOMGZ*

Specifies the rotational acceleration of the structure.

SOLUTION: Inertia

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

*DOMGX*, *DOMGY*, *DOMGZ*

Rotational acceleration of the structure about the global Cartesian X , Y, and Z axes.

**Notes**

Specifies the rotational acceleration of the structure about each of the global Cartesian axes. Rotational accelerations may be defined in analysis types **ANTYPE**, **STATIC**, **HARMIC** (full or mode superposition), **TRANS** (full or mode superposition), and **SUBSTR**. See [Acceleration Effect](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details. Units are radians/time<sup>2</sup>.

The **DOMEGA** command supports tabular boundary conditions (*%TABNAME\_X%*, *%TABNAME\_Y%*, and *%TABNAME\_Z%*) for *DOMEGA\_X*, *DOMEGA\_Y*, and *DOMEGA\_Z* input values (**\*DIM**) for full transient and harmonic analyses.

Related commands are **ACEL**, **CGLOC**, **CGOMGA**, **DCGOMG**, and **OMEGA**.

In a modal harmonic or transient analysis, you must apply the load in the modal portion of the analysis. ANSYS calculates a load vector and writes it to the mode shape file, which you can apply via the **LVSCALE** command.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>Global**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Accel>Global**  
**Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>Global**  
**Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Accel>Global**

**\*DOWHILE**, *Par*

Loops repeatedly through the next **\*ENDDO** command.

APDL: Process Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Argument Descriptions***Par*

The name of the scalar parameter to be used as the loop index. There is no character parameter substitution for the *Par* field.

## Notes

**\*DOWHILE** loops repeatedly through the next **\*ENDDO** command as long as *Par* is greater than zero. The block of commands following the **\*DOWHILE** command (up to the **\*ENDDO** command) is executed repeatedly until some loop control is satisfied. Printout is automatically suppressed on all loops after the first (include a **/GOPR** command to restore the printout). The command line loop control (*Par*) must be input; however, **\*IF** within the block can also be used to control looping [**\*EXIT**, **\*CYCLE**]. One level of internal file switching is used for each nested **\*DOWHILE**. Twenty levels of nested do-loops are allowed.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## DSCALE, RFACT, IFACT, TBASE

### Scales DOF constraint values.

[SOLUTION: FE Constraints](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

#### **RFACT**

Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

#### **IFACT**

Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

#### **TBASE**

Base temperature for temperature difference. For temperatures, the scale factor is applied to the temperature difference ( $T - T_{BASE}$ ) and then added to *TBASE*. *T* is the current temperature.

## Notes

Scales degree of freedom constraint values (displacement, temperature, etc.) in the database. If velocity and acceleration boundary conditions are applied in a structural analysis, they are also scaled by this command. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

---

### Note

Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

Scaling applies to the previously defined values for the selected nodes [**NSEL**] and the selected degree of freedom labels [**DOFSEL**]. Issue **DLIST** command to review results.

**DSCALE** does not work for tabular boundary conditions.

This command is also valid in PREP7.



## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Constraints  
 Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Constraints

**/DSCALE**, *WN*, *DMULT*

Sets the displacement multiplier for displacement displays.

GRAPHICS: Scaling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*WN*

Window number (or ALL) to which command applies (defaults to 1).

*DMULT*

**AUTO or 0**

Scale displacements automatically so that maximum displacement (vector amplitude) displays as 5 percent of the maximum model length, as measured in the global Cartesian X, Y, or Z directions.

**1**

Do not scale displacements (i.e., scale displacements by 1.0, true to geometry). Often used with large deflection results.

**FACTOR**

Scale displacements by numerical value input for FACTOR.

**OFF**

Remove displacement scaling (i.e., scale displacements by 0.0, no distortion).

**USER**

Set *DMULT* to that used for last display (useful when last *DMULT* value was automatically calculated).

## Command Default

The default value is AUTO or 0 except when:

Large deflection effects are included (**NLGEOM,ON**) and it is not a modal analysis; then the default is 1.

It is a spectrum analysis (**ANTYPE,SPECTR**); then the default is OFF.

The amplitude of a time-harmonic solution is computed using the **HRCPLX** command ( $\text{OMEGAT} \geq 360^\circ$ ); then the default is OFF.

The amplitude of a complex modal or harmonic solution is stored into the database using the **SET** command ( $\text{KIMG} = \text{AMPL}$ ); then the default is OFF.

## Notes

If Multi-Plots are not being displayed, and the current device is a 3-D device [**/SHOW,3D**], then the displacement scale in all active windows will be the same, even if separate **/DSCALE** commands are issued for each active window. For efficiency, ANSYS 3-D graphics logic maintains a single data structure (segment), which contains only one displacement scale. The program displays the same segment (displacement scale) in all windows. Only the view settings will be different in each of the active windows.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Displacement Scaling

---

**DSET**, *NODE1*, *NODE2*, *NODE3*, *DDEV*

Sets the scale and drawing plane orientation for a digitizing tablet.

[PREP7: Digitizing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NODE1*, *NODE2*, *NODE3*

Any three (noncolinear) nodes defining a plane parallel to the drawing. Nodes and actual locations (in any coordinate system) must have been previously defined.

*DDEV*

Digitizing device type number (device dependent).

## Notes

Sets drawing scale size and defines the drawing plane orientation for use with a digitizing tablet. Drawings must be to scale. Views must represent standard orthogonal parallel projections. The three nodes indicated must be digitized [**DIG**] from the tablet after this command is issued.

## Menu Paths

Main Menu>Preprocessor>Create>Nodes>Digitize Nodes>Set Plane/Device

---

**DSPOPTION**, *Reord\_Option*, *Memory\_Option*, *Memory\_Size*, --, --, *Solve\_Info*

Sets memory option for the distributed sparse solver.

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

*Reord\_Option*

Reordering option:

### DEFAULT

Use the default reordering scheme.

### PARORDER

Use a distributed memory parallel equation reordering scheme within the distributed sparse solver. This option can often help the distributed sparse solver achieve greater scalability (typically, at 16 or more cores) by doing a key solver step in parallel instead of sequentially. However, the parallel reordering results can sometimes be worse than the default reordering scheme, thus having an adverse effect on the factorization times and resulting in worse overall performance. It is best to only use this option when trying to maximize performance.

*Memory\_Option*

Memory allocation option:

**DEFAULT**

Use the default memory allocation strategy for the distributed sparse solver. The default strategy attempts to run in the INCORE memory mode. If there is not enough physical memory available when the solver starts to run in the INCORE memory mode, the solver will then attempt to run in the OPTIMAL memory mode.

**INCORE**

Use a memory allocation strategy in the distributed sparse solver that will attempt to obtain enough memory to run with the entire factorized matrix in memory. This option uses the most amount of memory and should avoid doing any I/O. By avoiding I/O, this option often achieves the best performance of all the memory modes. However, a significant amount of memory is required to run in this mode, and it is only recommended on machines with a large amount of memory. If the allocation for in-core memory fails, ANSYS will automatically revert to OPTIMAL out-of-core mode.

**OPTIMAL**

Use a memory allocation strategy in the distributed sparse solver that will attempt to allocate enough work space to keep the largest front matrix in memory during factorization. As a result, this memory mode often achieves an optimal balance between I/O and memory usage.

**FORCE**

This option, when used in conjunction with the *Memory\_Size* option, allows you to force the distributed sparse solver to run with a specific amount of memory. This option is only recommended for the advanced user who understands distributed sparse solver memory requirements for the problem being solved, understands the physical memory on the system, and wants to control the distributed sparse solver memory usage.

***Memory\_Size***

Initial memory size allocation for the sparse solver in MB. The *Memory\_Size* setting should always be well within the physical memory available, but not so small as to cause the distributed sparse solver to run out of memory. Warnings and/or errors from the distributed sparse solver will appear if this value is set too low. If the FORCE memory option is used, this value is the amount of memory allocated for the entire duration of the distributed sparse solver solution.

--,--

Unused fields

***Solve\_Info***

Solver output option:

**OFF**

Turns off additional output printing from the distributed sparse solver (default).

**PERFORMANCE**

Turns on additional output printing from the distributed sparse solver, including a performance summary and a summary of file I/O for the distributed sparse solver. Information on memory usage during assembly of the global matrix (that is, creation of the `Jobname.FULL` file) is also printed with this option.

***Command Default***

Automatic memory allocation is used.

**Notes**

This command controls options related to the distributed sparse solver in all analysis types where the distributed sparse solver can be used.

The amount of memory required for the distributed sparse solver is unknown until the matrix structure is preprocessed, including equation reordering. The amount allocated for the distributed sparse solver is then dynamically adjusted using the ANSYS memory manager.

If you have a large memory system, you may want to try selecting the INCORE memory mode for larger jobs to improve performance. Also, when running the distributed sparse solver with many processors on the same machine or on a machine with very slow I/O performance (e.g., slow hard drive speed), you may want to try using the INCORE memory mode to achieve better performance. However, doing so may require much more memory compared to running in the OPTIMAL memory mode.

Running with the INCORE memory mode is best for jobs which comfortably fit within the limits of the physical memory on a given system. If the distributed sparse solver workspace exceeds physical memory size, the system will be forced to use virtual memory (or the system page/swap file). In this case, it is typically more efficient to run with the OPTIMAL memory mode (**DSPOPTION,,OPTIMAL**).

## Menu Paths

**This command cannot be accessed from a menu.**

**DSUM**, *SIGNIF*, *Label*, *TD*

**Specifies the double sum mode combination method.**

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *SIGNIF*

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multipoint, or DDAM response (**SPOPT**, SPRS, MPRS, or DDAM), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**, PSD.)

### *Label*

Label identifying the combined mode solution output.

### **DISP**

Displacement solution (default). Displacements, stresses, forces, etc., are available.

### **VELO**

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

### **ACEL**

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

### *TD*

Time duration for earthquake or shock spectrum. *TD* defaults to 10.

## Notes

This command is also valid for PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine**  
**Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine**  
**Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine**  
**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine**

## DSURF, KCN, XSURF, YSURF, ZSURF

Defines the surface upon which digitized nodes lie.

[PREP7: Digitizing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### KCN

Surface is located in coordinate system *KCN*. *KCN* may be 0,1,2 or any previously defined local coordinate system number.

### XSURF, YSURF, ZSURF

Input one value to define the surface constant. Input 999 in the other two fields. Interpret fields as R,  $\theta$ , Z for cylindrical or R,  $\theta$ ,  $\Phi$  for spherical or toroidal coordinate systems. *XSURF* and *YSURF* default to 999 if *KCN* = 0.

## Command Default

Surface associated with **DIG** command is the global Cartesian X-Y plane with Z = 0.

## Notes

Defines the surface upon which the nodes to be digitized (with the **DIG** command) actually lie. Surfaces are defined by a coordinate system number and a coordinate constant [**MOVE**]. Two coordinates are determined from the drawing and converted to surface coordinates. The third coordinate is defined from the input surface constant. If nodes lie on warped or undefined surfaces, use the **DMOVE** command.

## Menu Paths

**Main Menu>Preprocessor>Create>Nodes>Digitize Nodes>Define Surface**

## DSYM, Lab, Normal, KCN

Specifies symmetry or antisymmetry degree-of-freedom constraints on nodes.

[SOLUTION: FE Constraints](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### Lab

Symmetry label:

**SYMM**

Generate symmetry constraints as described below (default).

**ASYM**

Generate antisymmetry constraints as described below.

**Normal**

Surface orientation label to determine the constraint set (surface is assumed to be perpendicular to this coordinate direction in coordinate system *KCN*):

**X**

Surface is normal to coordinate X direction (default). Interpreted as R direction for non-Cartesian coordinate systems.

**Y**

Surface is normal to coordinate Y direction.  $\theta$  direction for non-Cartesian coordinate systems.

**Z**

Surface is normal to coordinate Z direction.  $\Phi$  direction for spherical or toroidal coordinate systems.

**KCN**

Reference number of global or local coordinate system used to define surface orientation.

**Notes**

Specifies symmetry or antisymmetry degree-of-freedom constraints on the selected nodes. The nodes are first automatically rotated (any previously defined rotations on these nodes are redefined) into coordinate system *KCN*, then zero-valued constraints are generated, as described below, on the selected degree-of-freedom set (limited to displacement, velocity, and magnetic degrees of freedom) [**DOFSEL**]. Constraints are defined in the (rotated) nodal coordinate system, as usual. See the **D** and **NROTAT** commands for additional details about constraints and nodal rotations.

This command is also valid in PREP7.

**Symmetry and Antisymmetry Constraints:**

Symmetry or antisymmetry constraint generations are based upon the valid degrees of freedom in the model, i.e., the degrees of freedom associated with the elements attached to the nodes. The labels for degrees of freedom used in the generation depend on the *Normal* label.

For displacement degrees of freedom, the constraints generated are:

<i>Normal</i>	<b>SYMM</b>		<b>ASYM</b>	
	<b>2-D</b>	<b>3-D</b>	<b>2-D</b>	<b>3-D</b>
X	UX, ROTZ	UX, ROTZ, ROTY	UY	UY, UZ, ROTX
Y	UY, ROTZ	UY, ROTZ, ROTX	UX	UX, UZ, ROTY
Z	--	UZ, ROTX, ROTY	--	UX, UY, ROTZ

For velocity degrees of freedom, the constraints generated are:

	SYMM		ASYM	
<i>Normal</i>	2-D	3-D	2-D	3-D
X	VX	VX	VY	VY,VZ
Y	VY	VY	VX	VX,VZ
Z	--	VZ	--	VX,VY

For magnetic degrees of freedom, the SYMM label generates flux normal conditions (flux flows normal to the surface). Where no constraints are generated, the flux normal condition is "naturally" satisfied. The ASYM label generates flux parallel conditions (flux flows parallel to the surface).

	SYMM		ASYM	
<i>Normal</i>	2-D	3-D	2-D	3-D
X	--	AX	AZ	AY,AZ
Y	--	AY	AZ	AX,AZ
Z	--	AZ	--	AX,AY

### Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Nodes**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Nodes**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Nodes**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes**

**Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Nodes**

**Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Nodes**

### DSYS, KCN

**Activates a display coordinate system for geometry listings and plots.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### KCN

Coordinate system reference number. KCN may be 0,1,2 or any previously defined local coordinate system number.

#### Note

If a cylinder is displayed in its cylindrical coordinate system (with a 1,0,0 view), it will be unrolled (developed) into a flat plane (with theta along the Y direction).

## Command Default

Global Cartesian ( $KCN = 0$ ) display coordinate system.

## Notes

Boundary condition symbols, vector arrows, and element coordinate system triads are not transformed to the display coordinate system. The display system orientation (for the default view) is X horizontal to the right, Y vertical upward, and Z out of the screen (normal).

Line directions and area directions (**/PSYMB,LDIR** and **/PSYMB,ADIR**) are not plotted for  $DSYS > 0$ .

When you create ANSYS 3-D annotation, the coordinates are stored to the database in the  $DSYS$  that was active at the time of creation. Changing the  $DSYS$  does not change the annotation coordinate data in the database.

This command is valid in any processor.

## Menu Paths

Utility Menu>WorkPlane>Change Display CS to>Global Cartesian  
Utility Menu>WorkPlane>Change Display CS to>Global Cylindrical  
Utility Menu>WorkPlane>Change Display CS to>Global Spherical  
Utility Menu>WorkPlane>Change Display CS to>Specified Coord Sys

---

## DTRAN

Transfers solid model DOF constraints to the finite element model.

[SOLUTION: Solid Constraints](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

## Notes

Constraints are transferred only from selected solid model entities to selected nodes. The **DTRAN** operation is also done if the **SBCTRAN** command is issued, and is automatically done upon initiation of the solution calculations [**SOLVE**].

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Constraints  
Main Menu>Solution>Define Loads>Operate>Transfer to FE>Constraints

---



**DUMP**, *NSTRT*, *NSTOP*

Dumps the contents of a binary file.

[AUX2: Binary Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NSTRT***, ***NSTOP***

Dump file from record *NSTRT* (defaults to 1) to *NSTOP* (defaults to *NSTRT*). If *NSTRT* = HEAD, dump only record 1 of the file (*NSTOP* and the format specification are ignored). If *NSTRT* = ALL, dump the entire file.

**Notes**

Dumps the file named on the AUX2 **FILEAUX2** command according the format specified on the **FORM** command.

**Menu Paths**

Utility Menu>File>List>Binary Files

Utility Menu>List>Files>Binary Files

**/DV3D**, *Lab*, *Key*

Sets 3-D device option modes.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Lab***

Mode label:

**ACCU**

Allows ANSYS to use the accumulation buffer for OpenGL graphics. Activating this feature will provide faster model rotation when shaded backgrounds are in use. This feature is off by default.

**ACTR**

Label term to designate the cursor position as the center for automatic dynamic rotational center capability . The subsequent *Key* value (see below) turns this capability on and off. This feature is on by default. (Available for OpenGL displays only)

**ANIM**

Animation mode. The ANIM option allows you to create animation frames in pixmap mode instead of display list mode. This may improve large model performance, but it eliminates local manipulation while animation is in progress. This feature is on by default.

**ANTI**

Label term to control Anti-aliasing, a smoothing technique for your graph plots. (see below) The subsequent *Key* value turns this capability on and off. The default for this feature is off. (Available for OpenGL displays only).

**CNTR**

Switches banded contours on (1) or off (0) for your 3-D contour display. The default is 1 (ON). Other contour parameters such as number of contours or the increment and range are defined using the

**/CONTOUR** command. When either 9 or 128 contours are specified via **/CONTOUR**, this command is ignored and a smooth contour is always displayed.

### DGEN

Local manipulation degenerate mode. You access the DGEN option to set wire-frame local manipulation mode for 3-D devices (device dependent). This feature is off by default.

### DLIST

With DLIST, you can specify whether screen updates and redraws will be performed using the ANSYS Display List (off), or the 3-D device's Display List (on). DLIST is on by default for Windows systems, but off for UNIX. .

### DELS

You use DELS to suppress contour display screen overwrites when **/NOERASE** is active. This prevents the bleed-through that occurs when you overlay contour plots.

### TRIS

Triangle strip mode. Tri-stripping provides faster 3-D display capabilities and is on by default. Some display enhancements, such as texturing, are adversely affected by tri-stripping. You can turn off tri-stripping in order to improve these display functions. Be sure to turn tri-stripping on after the desired output is obtained.

### Key

The following key options apply to  $Lab = ACCU$ :

- 0**  
(OFF) The accumulation buffer is not accessed. (default)
- 1**  
(ON) Access to the buffer is enabled.

The following key options apply to  $Lab = ACTR$ :

- 0**  
(OFF) The cursor position has no effect on the existing rotational center for dynamic operations.
- 1**  
(ON) The rotational center for dynamic rotations in OpenGL is determined by the position of the mouse cursor on (or within 15 pixels of) the model. Any rotations that are initiated with the cursor more than 15 pixels from the model will occur about the midpoint of the Z-axis at that point in space. If the Z-buffer has not been refreshed the Z-axis will have an infinite value, and rotations will appear to occur about an extremely long Z-axis. This behavior stops when the graphics window is refreshed or replotted. (default)

Note that when using the GUI in 3-D mode, when  $ACTR = 1$ , the **Rotational Center** option is grayed out under **Utility Menu > PlotCtrls > View Setting** because the rotational center is determined strictly by the position of the mouse cursor.

The following key options apply to  $Lab = ANIM$ :

- 0**  
Display list animation. The object can be dynamically manipulated while animating. No legend, countour or annotation items are displayed. (see Notes, below)
- 1**  
On UNIX, device-dependent pixmap animation is used. On the PC, bitmap animation is provided (default). When you animate in this mode, you cannot dynamically manipulate your model (see Notes, below)..

**2**

On the PC only, this option provides AVI animation which uses the AVI movie player .

Although you can create animations of multiple ANSYS window schemes, animations created with OpenGL display lists (/DV3D, ANIM, 0) do not retain the windowing scheme information. You CAN save multiple windows via the X11/WIN32 drivers, or via the OpenGL driver with /DV3D, ANIM, KEY in effect (where KEY is not zero).

The following key options apply to *Lab* = ANTI

**0**

(OFF) Anti-aliasing is not active (default).

**1**

(ON) The anti-aliasing technique will be applied to smooth the lines in your displays (valid for OpenGL only).

The following key options apply to *Lab* = DGEN:

**0**

Normal manipulation.

**1**

Wireframe Manipulation.

The following key options apply to *Lab* = DLIST:

**0**

(OFF) The ANSYS Display List is used for plotting and dynamic graphics manipulation (UNIX default).

**1**

(ON) The local (3-D Device) Display List is used for plotting and dynamic rotation (Windows default).

The following key options apply to *Lab* = TRIS:

**0**

(OFF) Tri-stripping is off.

**1**

(ON) Tri-stripping is on (default).

This command is valid in any processor.

## Notes

ANSYS uses display list animation for its 3-D models. This memory resident array method interfaces with the OpenGL model information to allow the program to efficiently pan, zoom, rotate and dynamically manipulate your model during animation. The logo, legend, contour and other annotation items are produced in 2-D and will not appear when /DV3D, ANIM, 0 is in effect. To display these items, use /DV3D, ANIM, 1. All screen data will be displayed, but manipulation of the model will not be possible.

## Menu Paths

**Utility Menu>PlotCtrls>Device Options**

**DVAL**, *BaseID*, *Lab*, *VALUE*, *VALUE2*, *KeyCal*

**Defines values at enforced base.**

SOLUTION: FE Constraints

MP ME ST PR PRN <> <> FL EM <> <> PP <> EME MFS

**BaseID**

The identification number of the defined enforced base (from the **D** command in the modal analysis).

**Lab**

**U**

Enforced displacement.

**ACC**

Enforced acceleration.

**VALUE**

The value or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent (%) signs (for example: *DVAL*, *BaseID*, *U*, %tablename%). Use the **\*DIM** command to define a table.

**VALUE2**

The value of the second degree of freedom (if present). If the analysis type and the degree of freedom allow a complex input, *VALUE* is the real component and *VALUE2* is the imaginary component.

**KeyCal**

Calculation Key:

**ON**

Automatically calculate enforced displacement or acceleration based on the input enforced acceleration or displacement (default).

**OFF**

Do not automatically calculate enforced displacement.

## Notes

In modal superposition harmonic or transient analysis, you can apply enforced displacement or acceleration load when *KeyCal* is *ON*. If multiple loads are specified, the last applied load overrides the previous ones. For example, the following commands apply displacement to the base with identification number = 1:

**DVAL**,1,*U*,*VALUE*

**DVAL**,1,*ACC*,*VALUE*

In this case, the acceleration (*ACC*) applied in the last command will override the displacement (*U*).

## Menu Paths

**This command cannot be accessed from a menu.**

---

**DVMORPH**, *VOLU*, *XAREA*, *RMSHKY*

**Move nodes in selected volumes to conform to structural displacements.**

PREP7: Morphing

MP <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

***VOLU***

Non-structural volume to which mesh movement (morph) applies. If ALL, apply morphing to all selected volumes [VSEL]. If *VOLU* = P, graphical picking is enabled. A component may be substituted for *VOLU*.

***XAREA***

Areas to be excluded from morphing. If ALL, exclude all selected areas [ASEL]. If *XAREA* = P, graphical picking is enabled. A component may be substituted for *XAREA*. If *XAREA* is blank (default), allow morphing of nodes attached to areas of the selected volumes (*VOLU*) which are not shared by unselected volumes. (See Notes for clarification).

***RMSHKY***

Remesh flag option:

- 0** Remesh the selected non-structural volumes only if mesh morphing fails.
- 1** Remesh the selected non-structural volumes and bypass mesh morphing.
- 2** Perform mesh morphing only and do not remesh.

**Notes**

The selected volumes should include only non-structural regions adjacent to structural regions. DVMORPH will morph the non-structural volumes to coincide with the deflections of the structural regions.

Nodes in the structural regions move in accordance with computed displacements. Displacements from a structural analysis must be in the database prior to issuing DVMORPH.

By default, nodes attached to areas can move along the areas. You can use *XAREA* to restrain nodes on certain areas.

By default (*RMSHKY* = 0), DVMORPH will remesh the selected non-structural volumes entirely if a satisfactory morphed mesh cannot be provided.

If boundary conditions and loads are applied directly to nodes and elements, the DVMORPH command requires that these be removed before remeshing can take place.

Exercise care with initial conditions defined by the **IC** command. Before a structural analysis is performed for a sequentially coupled analysis, the DVMORPH command requires that initial conditions be removed from all null element type nodes in the non-structural regions. Use **ICDELE** to delete the initial conditions.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Modify Mesh>Phys Morphing>Volumes**

## DYNOPT

Specifies "Dynamic analysis options" as the subsequent status topic.

SOLUTION: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu > List > Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Utility Menu > List > Status > Solution > Dynamics Options**

---

---

## E Commands

**E**, *I, J, K, L, M, N, O, P*

**Defines an element by node connectivity.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*I*

Number of node assigned to first nodal position (node I). If  $I = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

*J, K, L, M, N, O, P*

Number assigned to second (node J) through eighth (node P) nodal position, if any.

### Notes

Defines an element by its nodes and attribute values. Up to 8 nodes may be specified with the **E** command. If more nodes are needed for the element, use the **EMORE** command. The number of nodes required and the order in which they should be specified are described in Chapter 4 of the *Element Reference* for each element type. Elements are automatically assigned a number [**NUMSTR**] as generated. The current (or default) MAT, TYPE, REAL, SECNUM and ESYS attribute values are also assigned to the element.

When creating elements with more than 8 nodes using this command and the **EMORE** command, it may be necessary to turn off shape checking using the **SHPP** command before issuing this command. If a valid element type can be created without using the additional nodes on the **EMORE** command, this command will create that element. The **EMORE** command will then modify the element to include the additional nodes. If shape checking is active, it will be performed before the **EMORE** command is issued. Therefore, if the shape checking limits are exceeded, element creation may fail before the **EMORE** command modifies the element into an acceptable shape.

### Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>Thru Nodes**

---

**EALIVE**, *ELEM*

**Reactivates an element (for the birth and death capability).**

[SOLUTION: Birth and Death](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***ELEM***

Element to be reactivated:

**ALL**

Reactivates all selected elements (**ESEL**).

**P**

Enables graphical picking of elements. All remaining command fields are ignored. (Valid only in the ANSYS GUI.)

**Comp**

Specifies a component name.

## Notes

Reactivates the specified element when the birth and death capability is being used. An element can be reactivated only *after* it has been deactivated (**EKILL**).

Reactivated elements have a zero strain (or thermal heat storage, etc.) state.

ANSYS recommends using the element deactivation/reactivation procedure for analyses involving linear elastic materials only. Do not use element deactivation/reactivation in analyses involving time-dependent materials, such as viscoelasticity, viscoplasticity, and creep analysis.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Birth & Death>Activate Elem**

**Main Menu>Solution>Load Step Opts>Other>Birth & Death>Activate Elem**

---

## EDADAPT, PART, Key

**Activates adaptive meshing in an explicit dynamic analysis.**

**SOLUTION: Explicit Dynamics**

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**PART**

Part ID (number) for which adaptive meshing is to be turned on (or off). Use *PART* = STAT to list the current adaptive meshing definitions.

**Key**

Adaptivity key:

**OFF**

Do not use adaptive meshing for the specified part ID (default).

**ON**

Use adaptive meshing for the specified part ID.

## Command Default

Adaptive meshing is off for all parts in the model.

## Notes

When adaptive meshing (adaptivity) is turned on, the mesh will automatically be regenerated to ensure adequate element aspect ratios. Adaptive meshing is most commonly used in the analysis of large deformation processes such as metal forming, in which the blank would need to be adaptively meshed.



Adaptive meshing is only valid for parts consisting of [SHELL163](#) elements. By default, adaptive meshing is OFF for all parts in the model. To specify adaptive meshing for more than one part in the model, you must issue the **EDADAPT** command for each part ID. Use the **EDPART** command to create and list valid part IDs. Use the **EDCADAPT** command to define additional adaptive meshing parameters.

The **EDADAPT** command is not supported in an explicit dynamic full restart analysis (**EDSTART,3**). In addition, a full restart cannot be performed successfully if adaptive meshing was used in the previous analysis.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Analysis Options>Adaptive Meshing>Apply to Part**  
**Main Menu>Solution>Analysis Options>Adaptive Meshing>Status**

---

**EDALE**, *Option*, --, *AFAC*, *BFAC*, --, *DFAC*, *EFAC*, *START*, *END*

**Assigns mesh smoothing to explicit dynamic elements that use the ALE formulation.**

[SOLUTION: Explicit Dynamics](#)

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### *Option*

Label identifying the option to be performed:

#### **ADD**

Add smoothing controls (default).

#### **DELETE**

Delete smoothing controls.

#### **LIST**

List smoothing controls.

--

Unused field.

#### **AFAC**

Simple average smoothing weight factor (default = 0).

#### **BFAC**

Volume weighted smoothing weight factor (default = 0).

--

Unused field.

#### **DFAC**

Equipotential smoothing weight factor (default = 0).

#### **EFAC**

Equilibrium smoothing weight factor (default = 0). *EFAC* is only applicable to [PLANE162](#) elements.

#### **START**

Start time for ALE smoothing (default = 0).

**END**

End time for ALE smoothing (default = 1e20).

**Command Default**

The Lagrangian formulation is used for all elements by default.

**Notes**

Mesh smoothing specified by the **EDALE** command is only applicable to **PLANE162** and **SOLID164** elements that are flagged to use the ALE formulation (**KEYOPT(5) = 1**). To activate the ALE formulation, you must specify at least one smoothing weight factor on this command and the number of cycles between advection (**NADV**) on the **EDGCALE** command. See [Arbitrary Lagrangian-Eulerian Formulation](#) in the *ANSYS LS-DYNA User's Guide* for more information.

The **EDALE** command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Solution>Analysis Options>ALE Options>Define**  
**Main Menu>Solution>Analysis Options>ALE Options>Delete All**  
**Main Menu>Solution>Analysis Options>ALE Options>List All**

**EDASMP**, *Option*, *ASMID*, *PART1*, *PART2*, *PART3*, *PART4*, *PART5*, *PART6*, *PART7*, *PART8*, *PART9*, *PART10*, *PART11*, *PART12*, *PART13*, *PART14*, *PART15*, *PART16*

**Creates a part assembly to be used in an explicit dynamic analysis.**

[PREP7: Explicit Dynamics](#)

<> <> <> <> <> <> <> <> <> <> <> <> <> <> <> <>

**Option**

Label identifying the part assembly option to be performed.

**ADD**

Adds a part assembly (default).

**DELETE**

Deletes a part assembly.

**LIST**

Lists each part assembly number, and the part numbers that make up each part assembly.

**ASMID**

User defined part assembly ID number. The part assembly number cannot be the same as any currently defined part ID number.

**PART1, PART2, PART3, ..., PART16**

Part numbers to be included in the assembly (up to 16 different parts).

## Command Default

Default for *Option* is ADD. You must specify *ASMPID* to avoid an error message.

## Notes

Several ANSYS LS-DYNA commands (such as **EDCGEN**, **EDPVEL**, and **EDIS**) refer to assembly ID numbers. If you intend to use assembly ID numbers with these commands, you must first define the assembly ID numbers using **EDASMP**.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Assembly Options

**EDBOUND**, *Option*, *Lab*, *Cname*, *XC*, *YC*, *ZC*, *Cname2*, *COPT*

**Defines a boundary plane for sliding or cyclic symmetry.**

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> <> <> <> <>

### *Option*

Label identifying the symmetry plane option to be performed.

#### **ADD**

Define a sliding or cyclic symmetry plane.

#### **DELE**

Delete a specified sliding or cyclic symmetry plane.

#### **LIST**

List defined sliding or cyclic symmetry planes.

### *Lab*

Valid boundary options for defining a symmetry plane. A valid label must always be specified for adding, deleting, or listing boundary planes.

#### **SLIDE**

Sliding symmetry plane.

#### **CYCL**

Cyclic symmetry plane.

### *Cname*

Name of existing component [**CM**] to which boundary symmetry is to be applied or deleted. Component must consist of nodes. For *Option* = LIST, a component is not required because all defined symmetry planes are listed for the specified *Lab*. For *Option* = DELE, use *Cname* = ALL to delete all symmetry planes currently defined for the specified *Lab*.

### *XC*, *YC*, *ZC*

X, Y, and Z coordinates of the head of the vector defining normal (*Lab* = SLIDE) or axis of rotation (*Lab* = CYCL). The tail of the vector is at the global origin.

**Cname2**

Name of existing nodal component [**CM**] for which second cyclic boundary plane is to be applied. Each node in *Cname2* component is constrained to a corresponding node in the first component set. Therefore, component *Cname2* must have the same number of nodes as the *Cname* component. *Cname2* is valid only for *Lab* = CYCL.

**COPT**

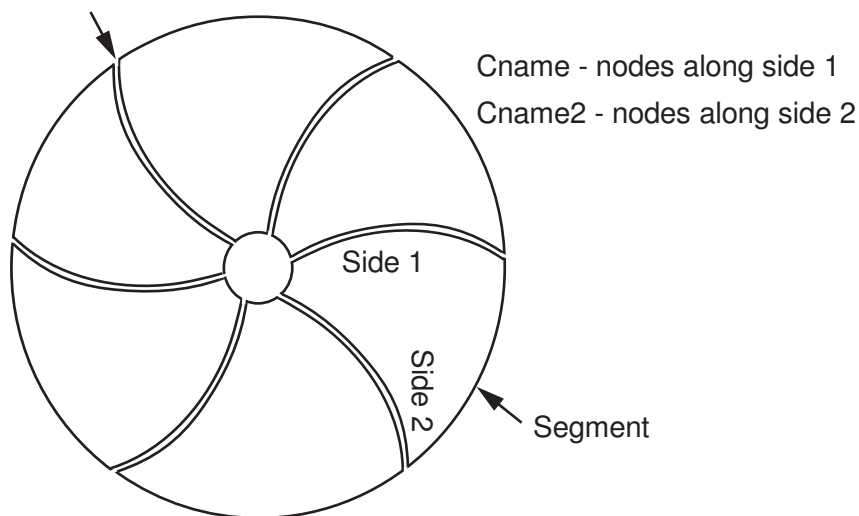
Specified constraint option for sliding plane symmetry. *COPT* is valid only for *Lab* = SLIDE. Valid *COPT* options are:

- 0**  
Nodes move on normal plane (default).
- 1**  
Nodes move only in vector direction.

**Notes**

For cyclic symmetry, the node numbers in component *Cname2* must differ from the node numbers in *Cname* by a constant offset value. In addition, the nodes in *Cname2* must have locations which, if given in cylindrical coordinates, all differ by the same angle from the nodes in *Cname*. The following figure shows how you would define components for a cyclic symmetry plane.

Conformable Interface



This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Symm Bndry Plane**  
**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Symm Bndry Plane>Delete All**  
**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Symm Bndry Plane>Delete Individ**  
**Main Menu>Solution>Constraints>Apply>Symm Bndry Plane**  
**Main Menu>Solution>Constraints>Delete>Symm Bndry Plane>Delete All**  
**Main Menu>Solution>Constraints>Delete>Symm Bndry Plane>Delete Individ**

---

**EDBX**, *Option*, *BOXID*, *XMIN*, *XMAX*, *YMIN*, *YMAX*, *ZMIN*, *ZMAX*

**Creates a box shaped volume to be used in a contact definition for explicit dynamics.**

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> DY <> <> <> <>

***Option***

Label identifying the contact box definition option to be performed.

**ADD**

Adds a contact box definition (default).

**DELETE**

Deletes a contact box definition.

**LIST**

Lists each box ID number, and the coordinates that make up each box shaped volume.

***BOXID***

User defined list ID number.

***XMIN***

Minimum x-coordinate.

***XMAX***

Maximum x-coordinate.

***YMIN***

Minimum y-coordinate.

***YMAX***

Maximum y-coordinate.

***ZMIN***

Minimum z-coordinate.

***ZMAX***

Maximum z-coordinate.

## ***Command Default***

Default for *Option* is ADD. You must specify *BOXID* to avoid an error message.

## **Notes**

The ANSYS LS-DYNA command **EDCGEN** allows you to define contact and target volumes using box ID numbers *BOXID1* and *BOXID2*, respectively. If you use these arguments to define contact volumes, you must first define their coordinates using the EDBX command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## **Menu Paths**

**Main Menu>Preprocessor>LS-DYNA Options>Contact>Define Box**

**EDBVIS**, *QVCO*, *LVCO*

**Specifies global bulk viscosity coefficients for an explicit dynamics analysis.**

**SOLUTION:** Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**QVCO**

Quadratic viscosity coefficient (defaults to 1.5).

**LVCO**

Linear viscosity coefficient (defaults to 0.06).

**Notes**

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Bulk Viscosity**

**Main Menu>Preprocessor>Material Props>Bulk Viscosity**

**Main Menu>Solution>Analysis Options>Bulk Viscosity**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Bulk Viscosity**

**EDCADAPT**, *FREQ*, *TOL*, *OPT*, *MAXLVL*, *BTIME*, *DTIME*, *LCID*, *ADPSIZE*, *ADPASS*, *IREFLG*, *ADPENE*, *ADPTH*, *MAXEL*

**Specifies adaptive meshing controls for an explicit dynamic analysis.**

**SOLUTION:** Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**FREQ**

Time interval between adaptive mesh refinements (default = 0.0). Use *FREQ* = STAT to list the current adaptive meshing control settings.

**TOL**

Adaptive angle tolerance (in degrees) for which adaptive meshing will occur (default = 1e31). If the relative angle change between elements exceeds the specified tolerance value, the elements will be refined.

**OPT**

Adaptivity option:

**1**

Angle change (in degrees) of elements is based on original mesh configuration (default).

**2**

Angle change (in degrees) of elements is incrementally based on previously refined mesh.

**MAXLVL**

Maximum number of mesh refinement levels (default = 3). This parameter controls the number of times an element can be remeshed. Values of 1, 2, 3, 4, etc. allow a maximum of 1, 4, 16, 64, etc. elements, respectively, to be created for each original element.

**BTIME**

Birth time to begin adaptive meshing (default = 0.0).

**DTIME**

Death time to end adaptive meshing (default = 1e31).

**LCID**

Data curve number (previously defined on the **EDCURVE** command) identifying the interval of remeshing (no default). The abscissa of the data curve is time, and the ordinate is the varied adaptive time interval. If LCID is nonzero, the adaptive frequency (*FREQ*) is replaced by this load curve. Note that a nonzero *FREQ* value is still required to initiate the first adaptive loop.

**ADPSIZE**

Minimum element size to be adapted based on element edge length (default = 0.0).

**ADPASS**

One or two pass adaptivity option.

**0**

Two pass adaptivity (default).

**1**

One pass adaptivity.

**IREFLG**

Uniform refinement level flag (no default). Values of 1, 2, 3, etc. allow 4, 16, 64, etc. elements, respectively, to be created uniformly for each original element.

**ADPENE**

Adaptive mesh flag for starting adaptivity when approaching (positive *ADPENE* value) or penetrating (negative *ADPENE* value) the tooling surface (default = 0.0).

**ADPTH**

Absolute shell thickness level below which adaptivity should begin. This option works only if the adaptive angle tolerance (*TOL*) is nonzero. If thickness based adaptive remeshing is desired without angle change, set *TOL* to a large angle. The default is *ADPTH* = 0.0, which means this option is not used.

**MAXEL**

Maximum number of elements at which adaptivity will be terminated (no default). Adaptivity is stopped if this number of elements is exceeded.

**Command Default**

No adaptive meshing.

**Notes**

The **EDCADAPT** command globally sets the control options for all part IDs that are to be adaptively meshed (see the **EDADAPT** command). Because *FREQ* defaults to zero, you must input a nonzero value in this field in order to activate adaptive meshing. You must also specify a reasonable value for *TOL* since the default adaptive angle tolerance (1e31) will not allow adaptive meshing to occur.

The **EDCADAPT** command is not supported in an explicit dynamic full restart analysis (**EDSTART,3**).

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>Analysis Options>Adaptive Meshing>Global Settings  
 Main Menu>Solution>Analysis Options>Adaptive Meshing>Status

**EDCGEN**, *Option, Cont, Targ, FS, FD, DC, VC, VDC, V1, V2, V3, V4, BTIME, DTIME, BOXID1, BOXID2*

**Specifies contact parameters for an explicit dynamics analysis.**

[PREP7: Explicit Dynamics](#)

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### *Option*

Label identifying the contact behavior (dictates the meaning of V1 through V4).

#### **AG**

Automatic general contact.

#### **ANTS**

Automatic nodes-to-surface contact.

#### **ASSC**

Automatic single surface contact.

#### **ASS2D**

Automatic 2-D single surface contact.

#### **ASTS**

Automatic surface-to-surface contact.

#### **DRAWBEAD**

Drawbead contact

#### **ENTS**

Eroding nodes-to-surface contact.

#### **ESS**

Eroding single surface contact.

#### **ESTS**

Eroding surface-to-surface contact.

#### **FNTS**

Forming nodes-to-surface contact.

#### **FOSS**

Forming one way surface-to-surface contact.

#### **FSTS**

Forming surface-to-surface contact.

#### **NTS**

Nodes-to-surface contact.

#### **OSTS**

One way surface-to-surface contact.

#### **RNTR**

Rigid nodes to rigid body contact.



**ROTR**

Rigid body to rigid body (one way) contact.

**SE**

Single edge contact.

**SS**

Single surface contact.

**STS**

Surface-to-surface contact.

**TDNS**

Tied nodes-to-surface contact.

**TSES**

Tied shell edge-to-surface contact.

**TDSS**

Tied surface-to-surface contact.

**TNTS**

Tiebreak nodes-to-surface contact

**TSTS**

Tiebreak surface-to-surface contact.

**Cont**

Contact surface identified by a component name [**CM**], a part ID number [**EDPART**], or an assembly ID number [**EDASMP**]. If a component name is input, the component must contain nodes that represent the contact surface (assemblies are not valid for a component name). Alternatively, a part number may be input that identifies a group of elements as the contact surface, or an assembly number may be input containing a maximum of 16 parts. The assembly ID number must be greater than the highest number used for the part ID. *Cont* is not required for automatic general contact, single edge contact, and single surface contact options (*Option* = AG, SE, ASSC, ESS, and SS). For automatic 2-D single surface contact (ASS2D), *Cont* must be defined as a part assembly. For eroding node-to-surface contact (ENTS), *Cont* must be defined as a nodal component. For eroding single surface contact (ESS) and eroding surface-to-surface contact (ESTS), *Cont* must be defined as a part ID or part assembly.

**Targ**

Target surface identified by a component name [**CM**], a part ID number [**EDPART**], or an assembly ID number [**EDASMP**]. If a component name is input, the component must contain nodes that represent the target surface (assemblies are not valid for a component name). Alternatively, a part number may be input that identifies a group of elements as the target surface, or an assembly number may be input containing a maximum of 16 parts. The assembly ID number must be greater than the highest number used for the part ID. *Targ* is not defined for automatic general contact, single edge contact, automatic single surface contact, eroding single surface contact, single surface contact, and automatic 2-D single surface contact options (*Option* = AG, SE, ASSC, ESS, SS, and ASS2D). For eroding node-to-surface contact (ENTS) and eroding surface-to-surface contact (ESTS), *Targ* must be defined as a part ID or part assembly.

**FS**

Static friction coefficient (defaults to 0).

**FD**

Dynamic friction coefficient (defaults to 0).

**DC**

Exponential decay coefficient (defaults to 0).

**VC**

Coefficient for viscous friction (defaults to 0).

**VDC**

Viscous damping coefficient in percent of critical damping (defaults to 0).

**V1, V2, V3, V4**

Additional input for drawbead, eroding, rigid, and tiebreak contact. The meanings of V1-V4 will vary, depending on *Option*. See the table below for V1-V4 definitions.

Additional input for drawbead contact (*Option* = DRAWBEAD):

**V1**

Load curve ID giving the bending component of the restraining force per unit draw bead length as a function of draw bead displacement. V1 must be specified.

**V2**

Load curve ID giving the normal force per unit draw bead length as a function of draw bead displacement. V2 is optional.

**V3**

Draw bead depth.

**V4**

Number of equally spaced integration points along the draw bead (default = 0, in which case ANSYS LS-DYNA calculates this value based on the size of the elements that interact with the draw bead).

Additional input for eroding contact (*Option* = ENTS, ESS, or ESTS):

**V1**

Symmetry plane option. The purpose of this option is to retain the correct boundary conditions in a model with symmetry.

**0**

Off (default).

**1**

Do not include faces with normal boundary constraints (e.g., segments of brick elements on a symmetry plane).

**V2**

Erosion/interior node option.

**0**

Erosion occurs only at exterior boundaries.

**1**

Interior eroding contact can occur (default).

**V3**

Adjacent material treatment for solid elements.

**0**

Solid element faces are included only for free boundaries (default).

**1**

Solid element faces are included if they are on the boundary of the material subset. This option also allows erosion within a body and the consequent treatment of contact.

Additional input for rigid contact (*Option* = RNTR or ROTR):

**V1**

Data curve id for force versus deflection behavior [**EDCURVE**]. Also specify *V2*. (No default.)

**V2**

Force calculation method for rigid contact. (No default.)

**1**

Data curve gives total normal force on surface versus maximum penetration of any node (only applicable for *Option* = ROTR).

**2**

Data curve gives normal force on each node versus penetration of node through the surface (*Option* = RNTR or ROTR).

**3**

Data curve gives normal pressure versus penetration of node through the surface (only applicable for *Option* = RNTR).

**V3**

Unloading stiffness for rigid contact. This should not be larger than the maximum value used in the data curve. The default is to unload along the data curve (specified on *V1*).

Additional input for tiebreak surface-to-surface contact (*Option* = TSTS). *V1* and *V2* are used to calculate the failure criterion:

**V1**

Normal failure stress. (No default.)

**V2**

Shear failure stress. (No default.)

Additional input for tiebreak nodes-to-surface contact (*Option* = TNTS). *V1* through *V4* are used to calculate the failure criterion:

**V1**

Normal failure force. Only tensile failure (i.e., tensile normal forces) will be considered in the failure criterion. (No default.)

**V2**

Shear failure force. (No default.)

**V3**

Exponent for normal force. (Defaults to 2.)

**V4**

Exponent for shear force. (Defaults to 2.)

***BTIME***

Birth time for which contact definition will become active (defaults to 0.0).

***DTIME***

Death time for which contact definition will become inactive (defaults to 1e21).

***BOXID1***

Contact volume as defined using the **EDBX** command (valid only when defining contact with parts or assemblies).

***BOXID2***

Target volume as defined using the **EDBX** command (valid only when defining contact with parts or assemblies).

## Command Default

No contact defined.

## Notes

The frictional coefficient used for contact is determined from  $FS$ ,  $FD$ , and  $DC$ , and is assumed to be dependent on the relative velocity of the surfaces in contact:

$$\mu_c = FD + (FS - FD) e^{-DC(v_{rel})}$$

The coefficient for viscous friction,  $VC$ , is necessary to limit the friction force to a maximum. A limiting force is computed:

$$F_{lim} = VC(A_{cont})$$

where  $A_{cont}$  is the area of the segment contacted by the node in contact. The suggested value for  $VC$  is to use the yield stress in shear:

$$VC = \frac{\sigma_o}{\sqrt{3}}$$

where  $\sigma_o$  is the yield stress of the contacted material.

If a part number is input for  $Cont$  or  $Targ$ , it must represent a valid explicit dynamics part definition. For example, an elastic material for explicit dynamics requires  $EX$ ,  $NUXY$ , and  $DENS$ . If any part of the material definition is missing for the part, the **EDCGEN** command will be ignored.

In addition to the contact parameters on this command, you can specify penalty scale factors for the contact (slave) and target (master) surfaces by using the **EDCMORE** command.

Duplicate definitions of the same contact type on the same components or parts will cause only one contact to be defined (previous definitions will be ignored). Duplicate definitions of different contact types on the same components or parts will cause multiple contact types to be defined.

Use the **EDCLIST** and **EDDC** commands to list and delete contact surface specifications. Use the **EDPC** command to select and plot contact entities.

The **EDCGEN** command is not supported in an explicit dynamic full restart analysis (**EDSTART,3**). Thus, you cannot add new contact specifications in a full restart. You can use the **EDCLIST** command to list any contact specifications that were defined in the previous analysis.

**EDCGEN** is also valid in **SOLUTION**.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Contact>Define Contact**

---

**EDCLIST**, *NUM*

**Lists contact entity specifications in an explicit dynamics analysis.**

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**NUM**

Number identifying contact entity to be listed. Use *NUM* = ALL to list all contact entities (ALL is the default).

**Notes**

Lists contact entity specifications previously defined with the **EDCGEN** command. The listing will include any contact parameters defined using the **EDCMORE** command.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>Preprocessor>LS-DYNA Options>Contact>List Entities

**EDCMORE**, *Option*, *NUM*, --, *VAL1*, *VAL2*

**Specifies additional contact parameters for a given contact definition in an explicit dynamic analysis.**

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**Option**

Label identifying the option to be performed.

**ADD**

Define contact parameters for the contact entity specified by *NUM* (default).

**DELE**

Delete contact parameters (*VAL1* and *VAL2*) for the contact entity specified by *NUM*. If *NUM* = ALL, all contact parameters previously defined by **EDCMORE** are deleted.

**NUM**

Contact entity number. This contact entity must have been previously defined with the **EDCGEN** command. Use **EDCLIST** to obtain a list of contact entity numbers.

**--**

Unused field.

**VAL1**

Penalty scale factor for slave (contact) surface (SFS); default = 1.

**VAL2**

Penalty scale factor for master (target) surface (SFM); default = 1.

## Command Default

For all contract definitions, SFS = 1 and SFM = 1.

## Notes

You can use the **EDCMORE** command to specify two additional contact parameters (SFS and SFM) for a specific contact definition. These parameters will apply only to the contact entity number entered on the *NUM* field. Use the **EDCLIST** command to obtain a list of contact definitions and their corresponding contact entity numbers. The listing produced by **EDCLIST** will include any contact parameters specified with the **EDCMORE** command.

When you use the **EDDC** command to delete a contact definition, any parameters you specified with **EDCMORE** for that contact definition will also be deleted. To delete only the parameters specified by **EDCMORE** for a given contact definition, use the command **EDCMORE,DELE,NUM**.

---

### Note

When you delete a contact definition with the **EDDC** command, the contact entity numbers will be renumbered for the remaining contact definitions. Therefore, you should always issue **EDCLIST** to obtain a current list of contact entity numbers before adding or deleting contact parameters with the **EDCMORE** command.

The **EDCMORE** command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Contact>Additional Parm

---

**EDCNSTR**, *Option*, *Ctype*, *Comp1*, *Comp2*, *VAL1*

Defines various types of constraints for an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <>

### *Option*

Label identifying the option to be performed.

### ADD

Define a constraint (default).

### DELE

Delete the constraint specified by *Ctype*, *Comp1*, and *Comp2*. If *Ctype* = ALL, all constraints are deleted.

### LIST

List all of the constraints previously defined by the **EDCNSTR** command.

### *Ctype*

Constraint type. The command format will vary, depending on the *Ctype* value.

**ENS**

Extra node set added to an existing rigid body.

**NRB**

Nodal rigid body.

**STS**

Tie between a shell edge and solid elements.

**RIVET**

Massless rivet between two noncoincident nodes.

If *Ctype* = ENS, the command format is **EDCNSTR,Option,ENS,Comp1,Comp2**

**Comp1**

Part number of the existing rigid body to which nodes will be added. The rigid body must be previously defined with the **EDMP** command. **EDMP** defines a rigid body based on material reference number (MAT). You must determine the corresponding part number (**EDPART**) for input in this field.

**Comp2**

Component name identifying extra nodes to be added to the rigid body specified by *Comp1*. *Comp2* must be a nodal component and must not be attached to any other rigid body.

If *Ctype* = NRB, the command format is **EDCNSTR,Option,NRB,Comp1,--,VAL1**

**Comp1**

Component name identifying a set of nodes that are to be defined as a rigid body. The component may consist of nodes from several different deformable parts.

--

This field is not used for *Ctype* = NRB.

**VAL1**

Coordinate system ID number (CID) to be used for output of data. The coordinate system must have been previously defined with the **EDLCS** command.

If *Ctype* = STS, the command format is **EDCNSTR,Option,STS,Comp1,Comp2**

**Comp1**

Node number of the shell element node that will be tied to solid element nodes that are specified by *Comp2*. The *Comp1* node must lie along the edge of a shell element and be coincident to at least one node included in *Comp2*.

**Comp2**

Component name consisting of solid element nodes (up to nine nodes) to which the shell element node will be tied. *Comp2* must consist of nodes that are on solid elements, and the nodes must define a line that will remain linear throughout the analysis. At least one of the nodes in *Comp2* must be coincident with the shell node specified in *Comp1*.

If *Ctype* = RIVET, the command format is **EDCNSTR,Option,RIVET,Comp1,Comp2**

**Comp1**

Node number of the first node on the rivet.

**Comp2**

Node number of the second node on the rivet. This node cannot have the same coordinates as the first node specified by *Comp1*.

## Notes

The **EDCNSTR** command allows you to define several types of constraints in an explicit dynamic analysis. A brief description of each constraint type is given below. See [Constraints and Initial Conditions](#) in the *ANSYS LS-DYNA User's Guide* for more information.

### Extra Node Set Added to a Rigid Body (*Ctype* = ENS)

The ability to add extra nodes to an existing rigid body has many potential applications, including placing nodes where joints will be attached between rigid bodies, defining nodes where point loads will be applied, and defining a lumped mass at a specific location. The extra nodes specified by *Comp2* may be located anywhere in the model and may have coordinates outside those of the original rigid body specified by *Comp1*.

### Nodal Rigid Body (*Ctype* = NRB)

Unlike typical rigid bodies that are defined with the **EDMP** command, nodal rigid bodies defined with the **EDCNSTR** command are not associated with a part number. This can be advantageous for modeling rigid (welded) joints in a model. For a rigid joint, portions of different flexible components (having different MAT IDs) act together as a rigid body. It is difficult to define this type of rigid body with a unique MAT ID (and corresponding part number). However, the rigid joint can be easily defined using a nodal rigid body.

### Shell Edge to Solid Tie (*Ctype* = STS)

The STS option ties regions of solid elements to regions of shell elements. A single shell node may be tied to up to nine brick element nodes that define a "fiber" vector. Solid element nodes constrained in this way remain linear throughout the analysis but can move relative to each other in the fiber direction.

### Rivet between Two Nodes (*Ctype* = RIVET)

The RIVET option defines a massless rigid constraint between two nodes, similar to spotwelds defined with the **EDWELD** command. Unlike a spotweld, however, rivets contain nodes that are noncoincident, and failure cannot be specified. When a rivet is defined, the distance between the nodes is kept constant throughout any motion that occurs during a simulation. Nodes connected by a rivet cannot be part of any other constraints specified in the model.

The **EDCNSTR** command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Additional Nodal**  
**Main Menu>Solution>Constraints>Apply>Additional Nodal**

---



**EDCONTACT**, *SFSI, RWPN, IPCK, SHTK, PENO, STCC, ORIE, CSPC, PENCHK***Specifies contact surface controls for an explicit dynamics analysis.**

PREP7: Explicit Dynamics

&lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; DY &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt;

**SFSI**

Scale factor for sliding interface penalties. Defaults to 0.1.

**RWPN**Scale factor for rigid wall penalties (defaults to 0). If *RWPN* = 0, rigid bodies interacting with rigid walls are not considered. If *RWPN* > 0, rigid bodies interact with fixed rigid walls. A value of 1.0 should be optimal; however, this may be problem dependent.**IPCK**

Initial contact surface penetration checking option:

**1**

No checking.

**2**

Full check of initial penetration is performed (default).

**SHTK**

Shell thickness contact option for surface-to-surface and nodes-to-surface contact (see Notes below):

**0**

Thickness is not considered (default).

**1**

Thickness is considered, except in rigid bodies.

**2**

Thickness is considered, including rigid bodies.

**PENO**

Penalty stiffness option (options 4 and 5 are useful for metal forming calculations):

**1**

Minimum of master segment and slave node (default).

**2**

Use master segment stiffness.

**3**

Use slave node value.

**4**

Use area or mass weighted slave node value.

**5**

Use slave node value inversely proportional to shell thickness. (This may require special scaling and is not generally recommended.)

**STCC**

Shell thickness change option for single surface contact:

**1**

Shell thickness changes are not considered (default).

- 2** Shell thickness changes are included.

**ORIE**

Option for automatic reorientation of contact surface segments during initialization:

- 1** Activate for automated (part ID) input only (default).
- 2** Activate for manual (nodal component) and automated (part ID) input.
- 3** Do not activate.

**CSPC**

Contact surface penetration check multiplier, used if small penetration checking is on (*PENCHK* = 1 or 2). Defaults to 4.

**PENCHK**

Small penetration check, used only for contact types STS, NTS, OSTs, TNTs, and TSTs. If the contact surface node penetrates more than the target thickness times *CSPC*, the penetration is ignored and the contacting node is set free. The target thickness is the element thickness for shell elements, or 1/20 of the shortest diagonal for solid elements.

- 0** Penetration checking is off (default).
- 1** Penetration checking is on.
- 2** Penetration checking is on, but shortest diagonal is used.

## Notes

The thickness offsets are always included in single surface, automatic surface-to-surface, and automatic nodes-to-surface contact. The shell thickness change option must be used [**EDSHELL**,,,1] and a nonzero value must be specified for *SHTK* before the shell thickness changes can be included in the surface-to-surface contact type. Additionally, *STCC* must be set to 2 if thickness changes are to be included in the single surface contact algorithms.

To reset the contact options to default values, issue the **EDCONTACT** command with no fields specified.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Contact>Advanced Controls**

---

**EDCPU**, *CPUTIME***Specifies CPU time limit for an explicit dynamics analysis.**[SOLUTION: Explicit Dynamics](#)

&lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; DY &lt;&gt; &lt;&gt; &lt;&gt;

**CPUTIME**

CPU time limit (in seconds) for the current phase of the analysis (defaults to 0). If *CPUTIME* = 0, no CPU time limit is set. *CPUTIME* values below 0 are not allowed.

**Notes**

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Solution>Analysis Options>CPU Limit**

**EDCRB**, *Option*, *NEQN*, *PARTM*, *PARTS***Constrains two rigid bodies to act as one in an explicit dynamics analysis.**[PREP7: Explicit Dynamics](#)

&lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; DY &lt;&gt; &lt;&gt; &lt;&gt;

**Option**

Label identifying the option to be performed:

**ADD**

Define an equation to constrain two rigid bodies (default).

**DELE**

Delete the equation (specified by *NEQN*) that constrains two rigid bodies. If *NEQN* is blank, all equations constraining rigid bodies are deleted.

**LIST**

List constrained rigid bodies specified by *NEQN*. If *NEQN* is blank, all constrained rigid bodies are listed.

**NEQN**

Equation reference number. Defaults to *PARTS*. *NEQN* should be a unique number for each pair of *PARTM* and *PARTS*. If it is not unique, the equation reference number defined last will overwrite any previously defined *NEQN* with the same number.

**PARTM**

PART number [**EDPART**] identifying the master rigid body. This value is ignored if the DELE or LIST labels are specified. No default; you must enter a value.

**PARTS**

PART number [**EDPART**] identifying the slave rigid body. This value is ignored if the DELE or LIST labels are specified. No default; you must enter a value.

## Notes

**EDCRB** is valid only for materials defined as rigid bodies with the **EDMP,RIGID** command. **EDCRB** automatically generates a constraint equation to force the specified rigid bodies to behave as a single rigid body. The slave rigid body takes on the material properties and loading of the master rigid body. Any loads [**EDLOAD**] existing on the slave rigid body are ignored.

To create a single large rigid body from several smaller bodies, use a series of **EDCRB** commands. With the first command, specify a master and slave to create the first combined rigid body. Then, using that body as the master, specify another slave to create a larger rigid body. Continue the process, using the expanding rigid body as the master and adding slave bodies until you have defined the desired large rigid body. All slave rigid bodies will take on the material properties and loading of the original master rigid body. Note that you will need to use different *NEQN* values for each pair of *PARTM* and *PARTS*. This command will be ignored if you specify the previously-defined master rigid body as a slave rigid body in the same analysis. To change the master and slave definitions, first use the **DELE** option to delete all master and slave definitions, and then use the **ADD** option to redefine them.

The equation number, *NEQN*, is a reference number by which the constrained bodies can be identified for listing and deleting purposes on the **EDCRB** command. For any other reference to the constrained bodies (loading, contact definitions, etc.), use the master body part number (*PARTM*).

This command is also valid in **SOLUTION**.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Rigid Body CE

---

## EDCSC, Key

Specifies whether to use subcycling in an explicit dynamics analysis.

**SOLUTION: Explicit Dynamics**

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <>

### Key

Subcycling key:

#### OFF

Do not use subcycling (default).

#### ON

Use subcycling.

## Command Default

No subcycling.

## Notes

Subcycling can be used to speed up an analysis when element sizes within a model vary significantly. Relatively small elements will result in a small time step size. When subcycling is on, the minimum time step size is increased for the smallest elements.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Time Controls>Subcycling**

## EDCTS, DTMS, TSSFAC

**Specifies mass scaling and scale factor of computed time step for an explicit dynamics analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### DTMS

Time step size for mass scaled solutions (defaults to 0).

### TSSFAC

Scale factor for computed time step. Defaults to 0.9; if high explosives are used, the default is lowered to 0.67.

## Command Default

No mass scaling; scale factor for computed time step = 0.9.

## Notes

If *DTMS* is positive, the same time step size will be used for all elements and mass scaling will be done for all elements. Therefore, positive values should only be used if inertial effects are insignificant.

If *DTMS* is negative, mass scaling is applied only to elements whose calculated time step size is smaller than *DTMS*. Negative values should only be used in transient analyses if the mass increases are insignificant.

In order to use mass scaling in an explicit dynamic small restart analysis (**EDSTART,2**) or full restart analysis (**EDSTART,3**), mass scaling must have been active in the original analysis. The time step and scale factor used in the original analysis will be used by default in the restart. You can issue **EDCTS** in the restart analysis to change these settings.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Time Controls>Time Step Ctrl**

**EDCURVE**, *Option*, *LCID*, *Par1*, *Par2*

**Specifies data curves for an explicit dynamic analysis.**

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> <> <> <> <>

**Option**

Label identifying the option to be performed.

**ADD**

Define a data curve (default). If *Option* = ADD, *Par1* and *Par2* must be previously defined array parameters.

**DELE**

Delete the specified data curve (*LCID*). If *LCID* is blank, all data curves are deleted. *Par1* and *Par2* are ignored for this option.

**LIST**

List defined data curve (*LCID*). If *LCID* is blank, all data curves are listed. *Par1* and *Par2* are ignored for this option.

**PLOT**

Plot defined data curve (*LCID*). If *Option* = PLOT, *LCID* must be previously defined with an **EDCURVE** command. Otherwise a warning message will report that *LCID* has not been defined. *Par1* and *Par2* are ignored for this option.

**LCID**

Data curve ID number (no default). Must be a positive integer.

**Par1**

Name of user-defined array parameter that contains the abscissa values of the curve data (e.g., time, effective plastic strain, effective strain rate, displacement, etc.).

**Par2**

Name of user-defined array parameter that contains the ordinate values of the curve data (e.g., damping coefficients, initial yield stress, elastic modulus, force, etc.) corresponding to the abscissa values in *Par1*.

---

**Note**

If the length of *Par1* and *Par2* are different, the shortest length will be used.

**Notes**

**EDCURVE** can be used to define material data curves (e.g., stress-strain) and load data curves (force-deflection) associated with material models in an explicit dynamics analysis. Material data specified by this command is typically required to define a particular material behavior (e.g., **TB,HONEY**), and the *LCID* number is used as input on the **TBDATA** command.

**EDCURVE** can also be used to define load curves that represent time dependent loads (force, displacement, velocity, etc.). *Par1* must contain the time values, and *Par2* must contain the corresponding load values. The *LCID* number assigned to the load curve can be used as input on the **EDLOAD** command.

## Note

You cannot update a previously defined data curve by changing the array parameters that were input as *Par1* and *Par2*. The data curve definition is written to the database at the time **EDCURVE** is issued. Therefore, subsequent changes to the array parameters that were used as input on **EDCURVE** will not affect the load curve definition. If you need to change the load curve definition, you must delete the load curve (**EDCURVE,DELE,LCID**) and define it again.

*LCID* identifies the data curve. If the value input for *LCID* is the same as the ID number for a data curve previously defined by **EDCURVE**, the previous data will be overwritten. Use **EDCURVE,LIST** and **EDCURVE,PLOT** to check existing data curves.

A starting array element number must be specified for *Par1* and *Par2*. The input for these fields must be a single column array parameter, or a specific column from a multi-column array parameter. When using the GUI with multi-column parameters, you must specify the parameter name and starting position for *Par1* and *Par2* by typing the **EDCURVE** command in the Input Window. This is because only the parameter name is available through the dialog box, which pulls in the first position of a single-column array parameter.

If you need to change a curve definition in an explicit dynamic small restart analysis, issue **EDSTART,2** first (to specify the restart), then issue the **EDCURVE** command. The revised curve must contain the same number of points as the curve it replaces. This limitation does not apply to a full restart analysis (**EDSTART,3**).

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Curve Options>Add Curve**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Curve Options>Delete Curve**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Curve Options>List Curve**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Curve Options>Plot Curve**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Curve Options>Add Curve**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Curve Options>Delete Curve**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Curve Options>List Curve**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Curve Options>Plot Curve**  
**Main Menu>Preprocessor>Material Props>Curve Options>Add Curve**  
**Main Menu>Preprocessor>Material Props>Curve Options>Delete Curve**  
**Main Menu>Preprocessor>Material Props>Curve Options>List Curve**  
**Main Menu>Preprocessor>Material Props>Curve Options>Plot Curve**  
**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Curve Options>Add Curve**  
**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Curve Options>Delete Curve**  
**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Curve Options>List Curve**  
**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Curve Options>Plot Curve**  
**Main Menu>Solution>Loading Options>Curve Options>Add Curve**  
**Main Menu>Solution>Loading Options>Curve Options>Delete Curve**  
**Main Menu>Solution>Loading Options>Curve Options>List Curve**

**EDDAMP**, *PART*, *LCID*, *VALDMP*

Defines mass weighted (Alpha) or stiffness weighted (Beta) damping for an explicit dynamics model.

*SOLUTION*: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <>

***PART***

*PART* number [EDPART] identifying the group of elements to which damping should be applied. If *PART* = ALL (default), damping is applied to the entire model.

***LCID***

Load curve ID (previously defined with the **EDCURVE** command) identifying the damping coefficient versus time curve. If time-dependent damping is defined, an *LCID* is required.

***VALDMP***

Constant system damping coefficient or a scale factor applied to the curve defining damping coefficient versus time.

**Notes**

Mass-weighted (Alpha) or stiffness-weighted (Beta) damping can be defined with the **EDDAMP** command. Generally, stiffness proportional or beta damping is effective for oscillatory motion at high frequencies. This type of damping is orthogonal to rigid body motion and so will not damp out rigid body motion. On the other hand, mass proportional or alpha damping is more effective for low frequencies and will damp out rigid body motion. The different possibilities are described below:

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

***Global Damping****Mass-weighted or Alpha damping*

When *PART* = (blank) or ALL (default), mass-weighted global damping can be defined in the following two ways. In this case, the same damping is applied for the entire structure.

- When the damping coefficient versus time curve (*LCID*) is specified using the **EDCURVE** command, *VALDMP* is ignored by LS-DYNA (although it is written in the LS-DYNA input file Jobname.K). The damping force applied to each node in the model is given by  $f_d = d(t)mv$ , where  $d(t)$  is the damping coefficient as a function of time defined by the **EDCURVE** command,  $m$  is the mass, and  $v$  is the velocity.
- When the *LCID* is 0 or blank (default), a constant mass-weighted system damping coefficient can be specified using *VALDMP*.

The constant and time-dependent damping, described above, cannot be defined simultaneously. The last defined global damping will overwrite any previously defined global damping.

***Damping defined for a PART****Mass-weighted or Alpha damping*



When *both* a valid *PART* number is specified and the damping coefficient versus time curve (*LCID*) is specified using the **EDCURVE** command, mass-weighted time-dependent damping will be defined for the particular *PART*. In this case, *VALDMP* will act as a scaling factor for the damping versus time curve (if *VALDMP* is not specified, it will default to 1). A valid *PART* number must be specified to define this type of damping. For example, use *PART* =1 (and not blank) when the entire model consists of only one *PART*. Issue the command repeatedly with different *PART* numbers in order to specify alpha damping for different *PARTS*.

#### *Stiffness-weighted or Beta damping*

When a valid *PART* number is specified with *LCID* = 0 or (blank) (default), a stiffness-weighted (Beta) constant damping coefficient for this particular *PART* can be defined by *VALDMP*. The stiffness-weighted value corresponds to the percentage of damping in the high frequency domain. For example, 0.1 roughly corresponds to 10% damping in the high frequency domain. Recommended values range from 0.01 to 0.25. Values lower than 0.01 may have little effect. If a value larger than 0.25 is used, it may be necessary to lower the time step size significantly (see the **EDCTS** command). Issue the command repeatedly with different *PART* numbers in order to specify beta damping for different *PARTS*. Time-dependent stiffness-weighted damping is not available in ANSYS LS-DYNA.

The mass-weighted and stiffness-weighted damping, described above, cannot be defined simultaneously for a particular *PART* number. The last defined damping for the particular *PART* number will overwrite any previously defined mass-weighted or stiffness-weighted damping for this *PART*.

In order to define the mass-weighted and stiffness-weighted damping simultaneously, you can use the **MP,DAMP** command (instead of the **EDDAMP,PART,VALDMP** command) to define stiffness-weighted (Beta) constant damping coefficient. However, do *not* use both of these commands together to define stiffness-weighted (Beta) constant damping coefficient for a particular *PART*. If you do, duplicate stiffness-weighted (Beta) constant damping coefficients for this *PART* will be written to the LS-DYNA input file *Jobname.K*. The last defined value will be used by LS-DYNA. Also, note that the **MP,DAMP** command is applied on the *MAT* number, and not on the *PART* number. Since a group of elements having the same *MAT* ID may belong to more than one *PART* (the opposite is not true), you need to issue the **MP,DAMP** command only once for this *MAT* ID and the stiffness-weighted (Beta) damping coefficients will be automatically defined for all the *PARTS* with that *MAT* ID.

Mass-weighted and stiffness-weighted damping can be defined simultaneously using the **EDDAMP** command only when mass-weighted damping (constant or time-dependent) is defined as global damping (**EDDAMP,ALL,LCID,VALDMP**) and stiffness-weighted damping is defined for all necessary *PARTS* (**EDDAMP,PART,VALDMP**).

To remove defined *global* damping, reissue the **EDDAMP,ALL** command with *LCID* and *VALDMP* set to 0. To remove damping defined for a particular *PART*, reissue **EDDAMP,PART**, where *PART* is the *PART* number, with *LCID* and *VALDMP* set to 0. There is no default for the **EDDAMP** command, i.e., issuing the **EDDAMP** command with *PART* = *LCID* = *VALDMP* = 0 will result in an error. Stiffness-weighted damping defined by the **MP,DAMP** command can be deleted using **MPDELE,DAMP,MAT**.

In an explicit dynamic small restart (**EDSTART,2**) or full restart analysis (**EDSTART,3**), you can only specify global alpha damping. This damping will overwrite any alpha damping input in the original analysis. If you do not input global alpha damping in the restart, the damping properties input in the original analysis will carry over to the restart.

Damping specified by the **EDDAMP** command can be listed, along with other explicit dynamics specifications, by typing the command string **EDSOLV\$STAT** into the ANSYS input window. Beta damping specified by the **MP,DAMP** command can be listed by **MPLIST,MAT** command.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Damping**

**Main Menu>Preprocessor>Material Props>Damping**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Damping**

## EDDBL, KEY

Selects a numerical precision type of the explicit dynamics analysis.

[PREP7: Explicit Dynamics](#)

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <>

### KEY

Number or name identifying numerical precision to be used.

#### 0 or SINGLE

Select single precision version of LS-DYNA (default).

#### 1 or DOUBLE

Select double precision version of LS-DYNA.

#### STAT

Check the status of the numerical precision in effect.

## Notes

Sets the single or double precision version of LS-DYNA into effect. Please check the availability of the double precision version of LS-DYNA on your system before using the command. If it is not available, use the command default.

The double precision version may be up to 20% slower than the single precision version. The results may also vary based on problem specifications.

In addition to **EDDBL,STAT**, you can use the GUI dialog box to verify which precision version is currently chosen. The GUI is based on the database and is updated to reflect changes.

See [Double Precision LS-DYNA](#) for more information.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Analysis Options>Double Precision**



**EDDRELAX**, *Option*, *NRCYCK*, *DRTOL*, *DFFCTR*, *DRTERM*, *TSSFDR*, *IRELAL*, *EDTTL*

**Activates initialization to a prescribed geometry or dynamic relaxation for the explicit analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

***Option***

Specifies when dynamic relaxation is activated.

**ANSYS**

Stresses are initialized in ANSYS LS-DYNA to a prescribed geometry for small strains, according to the solution of an ANSYS (implicit) run. The explicit solution is based on the implicit X,Y,Z displacements and rotations contained in the `dre lax` file (created with the **REXPOR T** command).

**DYNA**

Dynamic relaxation is on. When you use this option, you can specify some or all of the parameters *NRCYCK*, *DRTOL*, *DFFCTR*, *DRTERM*, *TSSFDR*, *IRELAL*, and *EDTTL*. Any parameters that you do not specify are set to their default values.

**OFF**

Turn off initialization to a prescribed geometry (*Option* = ANSYS) or dynamic relaxation (*Option* = DYNA).

***NRCYCK***

Number of iterations between convergence checks for dynamic relaxation option. Default = 250.

***DRTOL***

Convergence tolerance for dynamic relaxation option. Default = 0.001.

***DFFCTR***

Dynamic relaxation factor. Default = 0.995.

***DRTERM***

Optional termination time for dynamic relaxation. Termination occurs at this time, or when convergence is attained, whichever comes first. Default = infinity.

***TSSFDR***

Scale factor for computed time step during dynamic relaxation. If zero, the value is set to *TSSFAC* (defined on the **EDCTS** command). After converging, the scale factor is reset to *TSSFAC*.

***IRELAL***

Automatic control for dynamic relaxation option based on algorithm of Papadrakakis.

**0**

Not active (default).

**1**

Active.

***EDTTL***

Convergence tolerance on automatic control of dynamic relaxation (default = 0.04).

**Notes**

Use *Option* = ANSYS when running an implicit-to-explicit sequential solution to initialize the structure to a static solution performed earlier by the ANSYS implicit solver. Use *Option* = DYNA to perform dynamic relaxation within the LS-DYNA program. Use *Option* = OFF to turn off previously specified stress initialization or dynamic relaxation. You must specify the *Option* you want; there is no default.

In LS-DYNA, the dynamic relaxation is performed before the regular transient analysis. The convergence process of the dynamic relaxation is not written to the ANSYS history file. The ANSYS results files only include the converged result of the dynamic relaxation, which is the result at time zero in the `Jobname.HIS` and `Jobname.RST` files.

You can restart a dynamic relaxation analysis (**EDSTART,2** or **EDSTART,3**) from a previous transient analysis or a previous dynamic relaxation analysis. In the restart, you can change or set the convergence criteria with the **EDDRELAX** command. Only the load curves that are flagged for dynamic relaxation (`PHASE = 1` or `2` on **EDLOAD**) are applied after restarting. If you restart the explicit portion of an implicit-to-explicit sequential solution, you do not need to reissue the **REXPORT** command because displacement information contained in the `drelax` file is already included in the LS-DYNA restart file. If the dynamic relaxation is activated from a regular transient analysis, LS-DYNA continues the output of data to ANSYS results files. This is unlike the dynamic relaxation phase at the beginning of the calculation for which only the converged solution is written.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>Analysis Options>Dynamic Relax

### EDDUMP, NUM, DT

Specifies output frequency for the explicit dynamic restart file (d3dump).

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <>

#### NUM

Number of d3dump (restart) files written during the analysis (defaults to 1). When you specify *NUM*, the time interval between restart files is  $TIME / NUM$ , where *TIME* is the analysis end-time specified on the **TIME** command.

#### DT

Time interval at which the d3dump (restart) files are written. If *NUM* is input, *DT* is ignored.

### Command Default

One restart file is written at the end of the analysis.

### Notes

You can use *NUM* or *DT* to specify the time interval at which d3dump restart files will be written. You should not specify both quantities; if both are input, *NUM* will be used. The restart files are written sequentially as `d3dump01`, `d3dump02`, etc.

In LS-DYNA, the restart file output is specified in terms of number of time steps. Because the total number of time steps is not known until the LS-DYNA solution finishes, ANSYS calculates an approximate number of time steps for the solution, and then uses *NUM* or *DT* to calculate the required LS-DYNA input. This approximated number of time steps may be different from the total number reached in LS-DYNA after the solution finishes. Therefore, the number of restart dump files or the output interval may differ slightly from what you requested using *NUM* or *DT*.

In an explicit dynamic small restart (**EDSTART,2**) or full restart analysis (**EDSTART,3**), the **EDDUMP** setting will default to the *NUM* or *DT* value used in the original analysis. You can issue **EDDUMP** in the restart to change this setting.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Output Controls>File Output Freq>Number of Steps**  
**Main Menu>Solution>Output Controls>File Output Freq>Time Step Size**

## EDELE, IEL1, IEL2, INC

**Deletes selected elements from the model.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *IEL1, IEL2, INC*

Delete elements from *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *INC* (defaults to 1). If *IEL1* = ALL, *IEL2* and *INC* are ignored and all selected elements [**ESEL**] are deleted. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *INC* are ignored).

## Notes

Deleted elements are replaced by null or "blank" elements. Null elements are used only for retaining the element numbers so that the element numbering sequence for the rest of the model is not changed by deleting elements. Null elements may be removed (although this is not necessary) with the **NUMCMP** command. If related element data (pressures, etc.) are also to be deleted, delete that data before deleting the elements. **EDELE** is for unattached elements only. You can use the **xCLEAR** family of commands to remove any attached elements from the database.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements**  
**Main Menu>Preprocessor>Modeling>Delete>Elements**  
**Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts**

## EDENERGY, HGEN, SWEN, SIEN, RLEN

**Specifies energy dissipation controls for an explicit dynamics analysis.**

[SOLUTION: Explicit Dynamics](#)

<> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### *HGEN*

Hourglass energy control key:

**OFF or 0**

Hourglass energy is not computed.

**ON or 1**

Hourglass energy is computed and included in the energy balance (default).

**SWEN**

Stonewall energy dissipation control key:

**OFF or 0**

Stonewall energy dissipation is not computed.

**ON or 1**

Stonewall energy dissipation is computed and included in the energy balance (default).

**SIEN**

Sliding interface energy dissipation control key:

**OFF or 0**

Sliding interface energy dissipation is not computed.

**ON or 1**

Sliding interface energy dissipation is computed and included in the energy balance (default).

**RLEN**

Rayleigh (damping) energy dissipation control key:

**OFF or 0**

Rayleigh energy dissipation is not computed.

**ON or 1**

Rayleigh energy dissipation is computed and included in the energy balance (default).

## Notes

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Analysis Options>Energy Options**

## EDFPLOT, Key

**Allows plotting of explicit dynamics forces and other load symbols.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**Key**

Load symbol plotting key.

**ON or 1**

Turn display of load symbols on (default).

**OFF or 0**

Turn display of load symbols off.

## Command Default

Display of load symbols on.

## Notes

You must issue **EDFPLOT,ON** to display explicit dynamics load symbols. The explicit load symbols are erased automatically upon a subsequent plot command.

An explicit load symbol always indicates a positive load direction (e.g., positive X direction for FX load), even if the load value is negative. The load symbol does not reflect the load magnitude. You can use standard ANSYS symbol controls to control the appearance of the load symbol. No load symbol is displayed for temperature loads.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Show Forces**  
**Main Menu>Solution>Loading Options>Show Forces**  
**Utility Menu>PlotCtrls>Symbols**

---

## EDGCALE, NADV, METH

Defines global ALE controls for an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <>

### NADV

Number of cycles between advection (default = 0).

### METH

Advection method.

**0**

Donor cell + Half Index Shift (first order accurate) (default).

**1**

Van Leer + Half Index Shift (second order accurate).

## Command Default

The Lagrangian formulation is used for all elements by default.

## Notes

This command sets global ALE controls in an explicit dynamic analysis. These ALE controls apply to all [PLANE162](#) or [SOLID164](#) elements in the model that are flagged to use the ALE formulation (KEYOPT(5) = 1). To activate the ALE formulation, you must specify the number of cycles between advection on this command



and at least one smoothing weight factor on the **EDALE** command. See [Arbitrary Lagrangian-Eulerian Formulation](#) in the *ANSYS LS-DYNA User's Guide* for more information.

To see the current **EDGCALE** settings, issue the command **EDALE,LIST**.

The **EDGCALE** command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Analysis Options>ALE Options>Define**

**/EDGE**, *WN, KEY, ANGLE*

**Displays only the "edges" of an object.**

GRAPHICS:Style

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **WN**

Window number (or ALL) to which command applies (defaults to 1).

### **KEY**

Edge key:

#### **Elements Plots**

**0**

Display common lines between all adjacent element faces.

**1**

Display only the common lines between non-coplanar faces (that is, show only the edges).

#### **Contour Plots**

**0**

Display only the common lines between non-coplanar faces.

**1**

Display common lines between all element faces.

### **ANGLE**

Largest angle between two faces for which the faces are considered to be coplanar (0° to 180°). Defaults to 45°. A smaller angle produces more edges, a larger angle produces fewer edges.

## Command Default

For element plots, display common lines between all adjacent element faces. For contour plots, display only the common lines between non-coplanar faces.

## Notes

The *ANGLE* field is used in PowerGraphics to determine geometric discontinuities. It is a tolerance measure for the differences between the normals of the surfaces being considered. Values within the tolerance are accepted as coplanar (geometrically continuous).

A surface can be displayed as an edge outline without interior detail. This is useful for both geometry and postprocessing displays. Element outlines are normally shown as solid lines for geometry and displacement displays. Lines common to adjacent "coplanar" element faces are removed from the display. Midside nodes of elements are ignored. The **/SHRINK** option is ignored with the edge option. **/EDGE** is not supported for **PLESOL** and **/ESHAPE** displays when in PowerGraphics mode [**/GRAPHICS,POWER**].

The **/EDGE** command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Edge Options

---

### EDHGLS, HGCO

Specifies the hourglass coefficient for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

#### HGCO

Hourglass coefficient value (defaults to 0.1). Values greater than 0.15 may cause instabilities.

## Notes

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Hourglass Ctrls>Global

Main Menu>Preprocessor>Material Props>Hourglass Ctrls>Global

Main Menu>Solution>Analysis Options>Hourglass Ctrls>Global

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Hourglass Ctrls>Global

---

### EDHIST, Comp

Specifies time-history output for an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

#### Comp

Name of the component containing nodes or elements for which output is desired. *Comp* is required.

## Command Default

No time-history output is written.

## Notes

The time-history output is written to the file `Jobname.HIS`. Output is written only for the nodes or elements contained in `Comp`. The data is written at time intervals specified on the **EDHTIME** command. If no time interval is specified, output is written at 1000 steps over the analysis. (See also the **EDOUT** command which controls time-history output in ascii form for an explicit dynamics analysis.)

Use **EDHIST,LIST** to list the time-history output specification. (The listing will include output requested with the **EDOUT** command.) Use **EDHIST,DELE** to delete the time-history output specification.

`Jobname.HIS` is a binary file that is read by the ANSYS time-history postprocessor (POST26). If LS-DYNA output has been requested on the **EDWRITE** command [**EDWRITE,LSDYNA** or **EDWRITE,BOTH**], the file `D3THDT` will also be written. `D3THDT` is a binary file that is read by the LS-POST postprocessor.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>Output Controls>Select Component

## EDHTIME, *NSTEP*, *DT*

Specifies the time-history output interval for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <>

### *NSTEP*

Number of steps at which output is written to the time-history file, `Jobname.HIS`, and the ASCII output files. Defaults to 1000. The time increment between output is  $TIME / NSTEP$ , where  $TIME$  is the analysis end-time specified on the **TIME** command.

### *DT*

Time interval at which output is written to the time-history file, `Jobname.HIS`, and the ASCII output files. If *NSTEP* is input, *DT* is ignored.

## Command Default

Time-history output is written at 1000 steps over the analysis.

## Notes

**EDHTIME** controls the number of steps at which output will be written to the time-history file, `Jobname.HIS` (see the **EDHIST** command), and any ASCII files requested on the **EDOUT** command. You can use *NSTEP* or *DT* to specify the output interval. You should not specify both quantities; if both are input, *NSTEP* will be used.

In an explicit dynamic small restart (**EDSTART**,2) or full restart analysis (**EDSTART**,3), the **EDHTIME** setting will default to the *NSTEP* or *DT* value used in the original analysis. You can issue **EDHTIME** in the restart to change this setting.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>Output Controls>File Output Freq>Number of Steps  
Main Menu>Solution>Output Controls>File Output Freq>Time Step Size

## EDINT, SHELLIP, BEAMIP

**Specifies number of integration points for explicit shell and beam output.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### SHELLIP

Number of shell integration points used for output (defaults to 3). For element **SHELL163**, each integration point is associated with a layer. *SHELLIP* must be  $\geq 3$ . If *SHELLIP* = 3, results are written for the shell top, middle, and bottom. If *SHELLIP* > 3, then the results for the first *SHELLIP* layers are written.

### BEAMIP

Number of beam integration points used for stress output for **BEAM161** (defaults to 4).

## Command Default

For **SHELL163**, output is available for the top, middle and bottom layers. For **BEAM161**, stress is available at 4 integration points (top-front, top-back, bottom-front, bottom-back). For the resultant beam formulation (KEYOPT(1) = 2), there is no stress output.

## Notes

The number of integration points is defined by the element real constant NIP for both the beam elements (in the cross section) and the shell elements (through the thickness).

For shell elements that have only 1 or 2 integration points (NIP = 1 or 2), use the default of *SHELLIP* = 3. If NIP = 1, the same results are reported at the top, middle, and bottom layers. If the NIP = 2, the results at the bottom correspond to integration point 1, the results at the top correspond to integration point 2, and the results at the middle are an average of the top and bottom results.

For shell elements with 2 x 2 integration points in the plane, the data from the four points are averaged, and there is a single output value for each layer.

If you set *BEAMIP* = 0, no stress output is written for **BEAM161** elements. In this case, the beams will not appear in any POST1 plots because the program assumes they are failed elements.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>Output Controls>Integ Pt Storage

**EDIPART**, *PART*, *Option*, *Cvect*, *TM*, *IRCS*, *Ivect*, *Vvect*, *CID*

**Defines inertia for rigid parts in an explicit dynamics analysis.**

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### **PART**

Part number for which the inertia is defined (the part number must have been previously generated using the **EDPART** command). The part should be composed of a rigid material (**EDMP**, RIGID). For *Option* = ADD, you must input a value; there is no default. For *Option* = DELE or LIST, *PART* defaults to all parts.

### **Option**

#### **ADD**

Define inertia for the specified *PART* (default).

#### **DELE**

Delete the inertia properties for the specified *PART*. The remaining fields are ignored. If *PART* is blank, inertia properties previously specified using **EDIPART** are deleted for all rigid parts.

#### **LIST**

List the inertia properties for the specified *PART*. The remaining fields are ignored. If *PART* is blank, inertia properties are listed for all rigid parts.

### **Cvect**

The vector containing the global Cartesian coordinates of the center of mass for the part. This vector must have been previously defined with a dimension of three (**\*DIM** command) and filled in as shown below. If *Cvect* is blank, the global Cartesian origin (0,0,0) is used as the center of mass.

*Cvect*(1) -- X-coordinate of the center of mass

*Cvect*(2) -- Y-coordinate of the center of mass

*Cvect*(3) -- Z-coordinate of the center of mass

### **TM**

Translation mass (no default, must be defined).

### **IRCS**

Flag for inertia tensor reference coordinate system.

#### **0 (or blank)**

Global inertia tensor (default). You must supply all six inertia tensor components (see *Ivect*).

#### **1**

Principal moments of inertia with orientation vectors. You must supply IXX, IYY, IZZ (see *Ivect*) and *CID*.

**Ivect**

The name of a vector containing the components of the inertia tensor. This vector must have been previously defined (**\*DIM** command) with a dimension of six and filled in as shown below. Vector entries 2, 3, and 5 are ignored if  $IRCS = 1$ . There is no default for this vector; it must be specified.

*Ivect*(1) -- IXX component of the inertia tensor  
*Ivect*(2) -- IXY (set this entry to zero if  $IRCS = 1$ )  
*Ivect*(3) -- IXZ (set this entry to zero if  $IRCS = 1$ )  
*Ivect*(4) -- IYY component of the inertia tensor  
*Ivect*(5) -- IYZ (set this entry to zero if  $IRCS = 1$ )  
*Ivect*(6) -- IZZ component of the inertia tensor

**Vvect**

The name of a vector containing the initial velocity (relative to the global Cartesian coordinate system) of the rigid part. This vector must have been previously defined (**\*DIM** command) with a dimension of six and filled in as shown below. If *Vvect* is blank, the initial velocity defaults to zero.

*Vvect*(1) -- Initial translational velocity of rigid body in X-direction  
*Vvect*(2) -- Initial translational velocity of rigid body in Y-direction  
*Vvect*(3) -- Initial translational velocity of rigid body in Z-direction  
*Vvect*(4) -- Initial rotational velocity of rigid body about the X-axis  
*Vvect*(5) -- Initial rotational velocity of rigid body about the Y-axis  
*Vvect*(6) -- Initial rotational velocity of rigid body about the Z-axis

**CID**

Local coordinate system ID. This coordinate system must have been previously defined with the **EDLCS** command. You must input *CID* if  $IRCS = 1$  (no default).

**Command Default**

Inertia properties are calculated by the program for all rigid parts.

**Notes**

The **EDIPART** command applies only to rigid parts (**EDMP**,RIGID). It allows you to input the inertia properties for the rigid part rather than having the program calculate the properties from the finite element mesh.

This command is also valid in Solution.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>LS-DYNA Options>Inertia Options>Define Inertia**  
**Main Menu>Preprocessor>LS-DYNA Options>Inertia Options>Delete Inertia**  
**Main Menu>Preprocessor>LS-DYNA Options>Inertia Options>List Inertia**

---

**EDIS**, *Option*, *PIDN*, *PIDO***Specifies stress initialization in an explicit dynamic full restart analysis.**SOLUTION: [Explicit Dynamics](#)

&lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; DY &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt;

**Option**

Label identifying the option to be performed.

**ADD**

Define stress initialization between parts (default).

**DELE**

Delete stress initialization between parts.

**LIST**

List stress initialization between parts.

**PIDN**

New part ID or part assembly ID in the full restart analysis (defaults to all parts in the model).

**PIDO**Old part ID or part assembly ID in the previous analysis, (default to *PIDN*).**Command Default**

No stress initialization is performed.

**Notes**

The **EDIS** command is only valid in an explicit dynamic full restart analysis (**EDSTART**,3). (**EDIS** is ignored if it is not preceded by the **EDSTART**,3 command.) Use **EDIS** to specify which parts and/or part assemblies should undergo stress initialization in the restart based on the stresses from the previous analysis. You can specify stress initialization for multiple parts (or part assemblies) by issuing **EDIS** multiple times. If you issue **EDIS** with no arguments, stress initialization is performed for all parts in the restart analysis that have a corresponding part (having the same part ID) in the previous analysis.

In a full restart analysis, the complete database is written as an LS-DYNA input file, *Jobname\_nn.K*. When the LS-DYNA solution begins, LS-DYNA performs the stress initialization using file *Jobname\_nn.K* and the restart dump file (*d3dumpnn* specified on the **EDSTART** command) from the previous analysis. At the end of initialization, all the parts that were specified by the **EDIS** commands are initialized from the data saved in the restart dump file. In order for the stress initialization to be performed successfully, the new parts in the full restart analysis and the old parts in the previous analysis must have the same number of elements, same element order, and same element topology. (The parts may have different identifying numbers.) If this is not the case, the stresses cannot be initialized. If part assemblies are used, the part assemblies must contain the same number of parts. (See [A Full Restart](#) in the *ANSYS LS-DYNA User's Guide* for more details).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.**Menu Paths****Main Menu>Solution>Analysis Options>Initial Stress**

---

**EDLCS**, *Option, CID, X1, Y1, Z1, X2, Y2, Z2, X3, Y3, Z3*

Defines a local coordinate system for use in explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**Option**

Label identifying the option to be performed:

**ADD**

Define a coordinate system (default).

**DELE**

Delete a coordinate system. If *CID* is blank, all coordinate systems are deleted.

**LIST**

List defined coordinate systems. If *CID* is blank, all coordinate systems are listed.

**CID**

Coordinate system ID.

**X1, Y1, Z1**

X, Y, and Z coordinates of a point on the local x-axis.

**X2, Y2, Z2**

X, Y, and Z coordinates of a point on the local x-y plane.

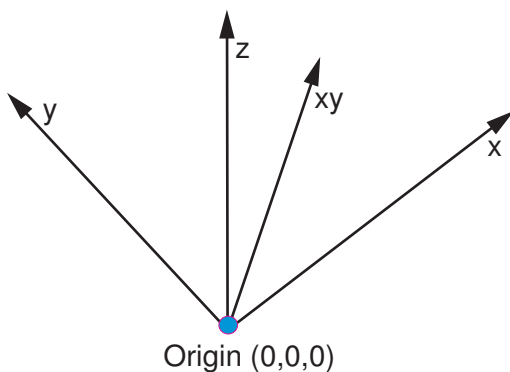
**X3, Y3, Z3**

X, Y, and Z coordinates of the origin. *X3*, *Y3*, and *Z3* all default to zero.

**Notes**

Local coordinate systems defined by this command are used in an explicit dynamic analysis. For example, a local coordinate system may be used when defining orthotropic material properties (see **EDMP**).

The coordinate system is defined by 2 vectors, one from the origin ( $X3, Y3, Z3$ ) to a point on the x-axis ( $X1, Y1, Z1$ ), and one from the origin to a point on the x-y plane ( $X2, Y2, Z2$ ). The cross product of these two vectors determines the z-axis, and the cross product of the z-axis vector and x-axis vector determines the y-axis. If  $X3, Y3$ , and  $Z3$  are not specified, the global origin (0,0,0) is used by default (as shown in the figure below).



The x-axis vector and the xy vector should be separated by a reasonable angle to avoid numerical inaccuracies.

When you use the local coordinate system (defined by the **EDLCS** command) to define a load (**EDLOAD** command), the direction of the load will depend on the load type. For force and moment loads ( $L_{ab} = FX$ ,





**FX, FY, FZ**

Forces.

**MX, MY, MZ**

Moments.

**UX, UY, UZ**

Displacements.

**ROTX, ROTY, ROTZ**

Rotations.

**VX, VY, VZ**

Velocities.

**OMGX, OMGY, OMGZ**

Angular velocities.

**AX, AY, AZ**

Accelerations (on nodes).

**ACLX, ACLY, ACLZ**

Base accelerations.

**TEMP**

Temperature.

Valid load labels for loads applied to elements:

**PRESS**

Pressure (applied to an element).

Valid load labels for loads applied to rigid bodies:

**RBFX, RBFY, RBFZ**

Forces on rigid bodies.

**RBMX, RBMY, RBMZ**

Moments on rigid bodies.

**RBUX, RBUY, RBUZ**

Displacements on rigid bodies.

**RBRX, RBRY, RBRZ**

Rotations on rigid bodies.

**RBVX, RBVY, RBVZ**

Velocities on rigid bodies.

**RBOX, RBOY, RBOZ**

Angular velocities on rigid bodies.

**KEY**

When  $Lab = PRESS$ ,  $KEY =$  Load key (face number) associated with a surface pressure load. Load keys (1,2,3, etc.) are listed under "Surface Loads" in the input data tables for each element type in the [Element Reference](#).

For most other values of  $Lab$ ,  $KEY$  is a coordinate system identification number, CID from the **EDLCS** command. The CID will represent either a local coordinate system (used for loads labels FX, MX, etc.) or a direction vector (used for prescribed motion labels UX, ROTX, VX, AX, etc.). See the Notes section for more information on how the CID is used. If the load is in the global coordinate system, set  $KEY$  equal

to zero, or leave it blank. Some load types do not support the CID key; see [Table 146: Birth Time, Death Time, and CID Support \(p. 473\)](#) in the Notes section for more information.

For *Lab* = PRESS, *KEY* defaults to 1 when *KEY* is blank or when *KEY* = zero. For all other values of *Lab*, *KEY* defaults to zero.

**Cname**

Name of existing component [**CM**] or PART number [**EDPART**] to which this load is to be applied. For all load labels except the pressure load (*Lab* = PRESS) and the rigid body loads (*Lab* = RB<sub>xx</sub>), the component must consist of nodes. For pressure loads, the component must consist of elements. For rigid body loads, a part number must be input instead of a component name. The part number must correspond to a set of elements that has been identified as a rigid body [**EDMP**,RIGID,*MAT*].

**Par1**

Name of user-defined array parameter that contains the time values of the load.

**Par2**

Name of user-defined array parameter that contains the "data" values of the load corresponding to the time values in *Par1*.

---

**Note**

If the length of *Par1* and *Par2* are different, the shortest length will be used.

**PHASE**

Phase of the analysis in which the load curve is to be used.

**0**

Curve is used in transient analysis only (default).

**1**

Curve is used in stress initialization or dynamic relaxation only.

**2**

Curve is used in both stress initialization (or dynamic relaxation) and transient analysis.

**LCID**

Data curve ID number representing the load curve to be applied. The load curve must have been previously defined using the **EDCURVE** command. If *LCID* is specified, *Par1* and *Par2* must be left blank (in the GUI, select "None" for *Par1* and *Par2*).

**SCALE**

Load curve scale factor applied to the specified load curve. The scale value is applied to the data in *Par2* or to the ordinate data in the load curve specified by *LCID*.

**BTIME**

Birth time, or time when imposed motion is activated. The default is 0.0. Some load types do not support birth and death time; see [Table 146: Birth Time, Death Time, and CID Support \(p. 473\)](#) in the Notes section for more information.

**DTIME**

Death time, or time when imposed motion is removed. The default is  $1 \times 10^{38}$ . Some load types do not support birth and death time; see [Table 146: Birth Time, Death Time, and CID Support \(p. 473\)](#) in the Notes section for more information.

## Notes

If a component name is input (*Cname*) and the specified component definition is changed before the **SOLVE** command, the last definition will be used.

You can specify the load data by inputting *LCID* (the ID number of a previously defined load curve) or by inputting the two array parameters *Par1* and *Par2* (which contain time and load values, respectively). The input for *Par1* and *Par2* may be a single column array parameter, or a specific column from a multi-column array parameter. A starting array element number can be specified for *Par1* and *Par2*; if none is specified, array element 1 is used by default.

---

### Note

You cannot update a previously defined load by changing the array parameters that were input as *Par1* and *Par2*. The load definition is written to the database at the time **EDLOAD** is issued. Therefore, subsequent changes to the array parameters that were used as input on **EDLOAD** will not affect the load curve definition. If you need to change the load definition, you must delete the load (**EDLOAD,DELE**) and define it again.

**EDLOAD** automatically assigns a load number to each defined load. Use **EDLOAD,LIST** to obtain a list of loads and their corresponding load numbers. You can plot a load curve by inputting the load number on the **EDPL** command. The load numbers may change when loads are deleted (**EDLOAD,DELE**). Therefore, you should obtain a current list of load numbers (**EDLOAD,LIST**) before plotting a load curve.

For prescribed motion, we recommend that you specify velocity time histories instead of displacement time histories. Also, you should not specify nonzero initial displacements. A piecewise linear displacement time history may lead to discontinuous velocities and infinite accelerations.

By default, the load will be applied in the global Cartesian direction. You can define the load in a different direction by inputting a CID (coordinate system ID) value in the *KEY* field. The CID must be previously defined using the **EDLCS** command. For load labels (*Lab* = FX, FY, FZ, MX, MY, MZ, RBFX, RBFY, RBFZ, RBMX, RBMY, RBMZ), the load will be applied in the direction of the local coordinate system defined by **EDLCS**. For prescribed motion degrees of freedom labels (*Lab* = UX, UY, UZ, ROTX, ROTY, ROTZ, VX, VY, VZ, AX, AY, AZ, RBUX, RBUY, RBUZ, RBRX, RBRY, RBRZ, RBVX, RBVY, RBVZ, RBOX, RBOY, RBOZ), the motion will act in the direction of a vector defined by two points input on the **EDLCS** command. The origin and terminus ends of the vector are defined by the *X1, Y1, Z1* and *X2, Y2, Z2* fields, respectively, of **EDLCS**.

For *Lab* = OMGX, OMGY, and OMGZ, you may need to specify the origin location of the acceleration coordinate system [**CGLOC**].

When applying a temperature load (*Lab* = TEMP), you may also need to define a reference temperature via the **TREF** command. The thermal loading is defined as the difference between the applied temperature and the reference temperature. Note that **EDLOAD,LIST** will list only the temperature values specified on **EDLOAD**, not the temperature difference.

When applying loads to axisymmetric **PLANE162** elements, the load may be interpreted differently depending on whether you use the area weighted or volume weighted option (**KEYOPT(2)**). See the **PLANE162** element description in the *Element Reference* for details.

Use *PHASE* = 0 when you are using the LS-DYNA solver to conduct a transient explicit analysis only or when you are conducting a sequential implicit/explicit analysis, in which the ANSYS (implicit) resulting displacements

(stored in the `drelax` file from the **REXPORT** command) are used to preload the explicit model [**EDDRELAX,ANSYS**]

Use *PHASE* = 1 or 2 when you need to use LS-DYNA to preload the model (as opposed to ANSYS) before running the transient portion of the analysis. *PHASE* = 1 applies the load initially and then immediately removes the load. As a result, the load is removed, and the structure vibrates freely. *PHASE* = 2 applies the load and then continues to apply the load over the course of the transient analysis, so that the transient analysis includes the effect of the initial loading and continues to account for the initial loading.

Birth and Death times, as well as the CID key are supported only for the **EDLOAD** labels specified with a Yes in the following table.

**Table 146 Birth Time, Death Time, and CID Support**

<b>EDLOAD Label</b>	<b>Birth Time</b>	<b>Death Time</b>	<b>KEY = CID</b>
FX, FY, FZ	No	No	Yes
MX, MY, MZ	No	No	Yes
UX, UY, UZ	Yes	Yes	Yes
ROTX, ROTY, ROTZ	Yes	Yes	Yes
VX, VY, VZ	Yes	Yes	Yes
OMGX, OMGY, OMGZ	No	No	No
AX, AY, AZ	Yes	Yes	Yes
ACLX, ACLY, ACLZ	No	No	No
TEMP	No	No	No
PRESS	Yes	No	No
RBFX, RBFY, RBFZ	No	No	Yes
RBMX, RBMY, RBMZ	No	No	Yes
RBUX, RBUY, RBUZ	Yes	Yes	Yes
RBRX, RBRY, RBRZ	Yes	Yes	Yes
RBVX, RBVY, RBVZ	Yes	Yes	Yes
RBOX, RBOY, RBOZ	Yes	Yes	Yes

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Delete Loads>Delete All**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Delete Loads>Delete Individ**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Specify Loads**  
**Main Menu>Solution>Loading Options>Delete Loads>Delete All**  
**Main Menu>Solution>Loading Options>Delete Loads>Delete Individ**  
**Main Menu>Solution>Loading Options>Specify Loads**



5

Flanagan-Belytschko stiffness form with exact volume integration for solid elements.

**VAL2**

Hourglass coefficient. (Defaults to 0.1.) Values greater than 0.15 may cause instabilities. The recommended default applies to all options. The stiffness forms can stiffen the response (especially if deformations are large) and, therefore, should be used with care. For the shell and membrane elements, the value input for *VAL1* is the membrane hourglass coefficient. *VAL5* and *VAL6* can also be input, but generally  $VAL2 = VAL5 = VAL6$  is adequate.

**VAL3**

Quadratic bulk viscosity coefficient. (Defaults to 1.5.)

**VAL4**

Linear bulk viscosity coefficient. (Defaults to 0.06.)

**VAL5**

Hourglass coefficient for shell bending. (Defaults to *VAL2*.)

**VAL6**

Hourglass coefficient for shell warping. (Defaults to *VAL2*.)

Additional input for rigid body constraint (*Lab* = RIGID).

**VAL1**

Translational constraint parameter (relative to global Cartesian coordinates).

0

No constraints (default).

1

Constrained X displacement.

2

Constrained Y displacement.

3

Constrained Z displacement.

4

Constrained X and Y displacements.

5

Constrained Y and Z displacements.

6

Constrained Z and X displacements.

7

Constrained X, Y, and Z displacements.

**VAL2**

Rotational constraint parameter (relative to global Cartesian coordinates).

0

No constraints (default).

1

Constrained X rotation.

2

Constrained Y rotation.

- 3** Constrained Z rotation.
- 4** Constrained X and Y rotations.
- 5** Constrained Y and Z rotations.
- 6** Constrained Z and X rotations.
- 7** Constrained X, Y, and Z rotations.

Additional input for cable properties ( $L_{ab} = \text{CABLE}$ ).

**VAL1**

Load curve ID defining engineering stress versus engineering strain (i.e., change in length over the initial length). If  $VAL1$  and Young's modulus [**MP**,EX] are input, the load curve corresponding to  $VAL1$  will be used and Young's modulus will be ignored.

Additional input for material coordinate system ( $L_{ab} = \text{ORTHO}$ ).

**VAL1**

Coordinate system ID number from the **EDLCS** command. This coordinate system will be used to orient the orthotropic or anisotropic materials associated with the material number,  $MAT$ .

Additional input for fluid material properties ( $L_{ab} = \text{FLUID}$ ).

**VAL1**

Bulk modulus of fluid. If  $VAL1$  is not input, the bulk modulus will be calculated from the elastic modulus (EX) and Poisson's ratio (NUXY).

## Notes

For  $L_{ab} = \text{RIGID}$ , you must specify elastic modulus (EX), density (DENS), and Poisson's ratio (NUXY) [**MP** command]. For  $L_{ab} = \text{CABLE}$ , you must specify density (DENS) and one of the following: Young's modulus (EX) or an engineering stress-strain curve ( $VAL1$ ). For  $L_{ab} = \text{FLUID}$ , you must specify either the bulk modulus ( $VAL1$ ) or both Young's modulus (EX) and Poisson's ratio (NUXY) (if all three are specified, only  $VAL1$  will be used).

After you define a rigid body using **EDMP**,RIGID, you may assign inertia properties to that rigid body using the **EDIPART** command.

**EDMP**,ORTHO is required for orthotropic or anisotropic material properties that are not locally orthotropic with material axes determined by element nodes (see [Orthotropic Elastic Model](#) and [Anisotropic Elastic Model](#) in the *ANSYS LS-DYNA User's Guide*).

Via the GUI,  $L_{ab} = \text{RIGID}$ ,  $L_{ab} = \text{CABLE}$ ,  $L_{ab} = \text{ORTHO}$ , and  $L_{ab} = \text{FLUID}$  are available through the material model interface. See [Defining Explicit Dynamics Material Models](#) in the *ANSYS LS-DYNA User's Guide* for more information.

Use the **MPLIST** and **MPDELE** commands to list and delete materials defined by the **EDMP** command.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.



## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Hourglass Ctrls>Local

Main Menu>Preprocessor>Material Props>Hourglass Ctrls>Local

Main Menu>Solution>Analysis Options>Hourglass Ctrls>Local

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Hourglass Ctrls>Local

**EDNB**, *Option*, *Cname*, *AD*, *AS*

Defines a nonreflecting boundary in an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### *Option*

Label identifying the nonreflecting boundary option to be performed.

#### **ADD**

Define a nonreflecting boundary (default).

#### **DELE**

Delete a nonreflecting boundary.

#### **LIST**

List all defined nonreflecting boundaries (remaining fields are ignored).

### *Cname*

Name of existing nodal component to which the nonreflecting boundary is to be added or deleted. For *Option* = DELE, use *Cname* = ALL to delete all defined nonreflecting boundaries.

### *AD*

Activation flag for dilatational waves (dampers normal to waves).

**0**

Dilatational activation flag is off (default).

**1**

Dilatational activation flag is on.

### *AS*

Activation flag for shear waves (dampers tangent to waves).

**0**

Shear activation flag is off (default).

**1**

Shear activation flag is on.

## Notes

Nonreflecting boundaries can be defined on the external surfaces of [SOLID164](#) and [SOLID168](#) elements that are being used to model an infinite domain. They are typically used in geomechanical applications to limit the size of the model. For example, when a half space is being modeled with a finite geometry, the nonreflecting boundary option can be used to prevent artificial stress wave reflections generated at the boundary from reentering the model and contaminating the results.

When using nonreflecting boundaries, you should not constrain the nodes at the boundary; doing so would negate the presence of the dampers. Usually, the large mass of the finite domain is sufficient to resist motion.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Non-Refl Bndry

Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Non-Refl Bndry>Delete All

Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Non-Refl Bndry>Delete Individ

Main Menu>Solution>Constraints>Apply>Non-Refl Bndry

Main Menu>Solution>Constraints>Delete>Non-Refl Bndry>Delete All

Main Menu>Solution>Constraints>Delete>Non-Refl Bndry>Delete Individ

---

## EDNDTSD, Vect1, Vect2, DATAP, FITPT, Vect3, Vect4, DISP

**Allows smoothing of noisy data for explicit dynamics analyses and provides a graphical representation of the data.**

[PREP7: Explicit Dynamics](#)

<> <> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### Vect1

Name of the first vector that contains the noisy data set (i.e., independent variable). You must create and fill this vector before issuing **EDNDTSD**.

### Vect2

Name of the second vector that contains the dependent set of data. Must be the same length as the first vector. You must create and fill this vector before issuing **EDNDTSD**.

### DATAP

Number of data points to be fitted, starting from the beginning of the vector. If left blank, the entire vector will be fitted. The maximum number of data points is 100,000 (or greater, depending on the memory of the computer).

### FITPT

Curve fitting order to be used as a smooth representation of the data. This number should be less than or equal to the number of data points. However, because high order polynomial curve fitting can cause numerical difficulties, a polynomial order less than 7 is suggested. The default (blank) is one-half the number of data points or 7, whichever is less. The following values are available:

#### 1

Curve is the absolute average of all of the data points.

#### 2

Curve is the least square average of all of the data points.

#### 3 or more

Curve is a polynomial of the order (n-1), where n is the number of data fitting order points.

### Vect3

Name of the vector that contains the smoothed data of the independent variable. This vector should have a length equal to or greater than the number of smoothed data points. In batch (command) mode,

you must create this vector before issuing the **EDNDTSD** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector `smth_ind`.

#### ***Vect4***

Name of the vector that contains the smoothed data of the dependent variable. This vector must be the same length as `Vect3`. In batch (command) mode, you must create this vector before issuing the **EDNDTSD** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector `smth_dep`.

#### ***DISP***

Specifies how you want to display data. No default; you must specify an option.

- 1** Unsmoothed data only
- 2** Smoothed data only
- 3** Both smoothed and unsmoothed data

## Notes

You can control the attributes of the graph using standard ANSYS controls (**/GRID**, **/GTHK**, **/COLOR**, etc.). If working interactively, these controls appear in this dialog box for convenience, as well as in their standard dialog boxes. You must always create `Vect1` and `Vect2` (using **\*DIM**) and fill these vectors before smoothing the data. If you're working interactively, ANSYS automatically creates `Vect3` and `Vect4`, but if you're working in batch (command) mode, you must create `Vect3` and `Vect4` (using **\*DIM**) before issuing **EDNDTSD**. `Vect3` and `Vect4` are then filled automatically by ANSYS. In addition, ANSYS creates an additional TABLE type array that contains the smoothed array and the unsmoothed data to allow for plotting later with **\*VPLOT**. Column 1 in this table corresponds to `Vect1`, column 2 to `Vect2`, and column 3 to `Vect4`. This array is named `Vect3_SMOOTH`, up to a limit of 32 characters. For example, if the array name is X1, the table name is X1\_SMOOTH.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Smooth Data**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Smooth Data**  
**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Smooth Data**  
**Main Menu>Solution>Loading Options>Smooth Data**  
**Main Menu>TimeHist Postpro>Smooth Data**

---

**EDNROT**, *Option*, *CID*, *Cname*, *DOF1*, *DOF2*, *DOF3*, *DOF4*, *DOF5*, *DOF6*

**Applies a rotated coordinate nodal constraint in an explicit dynamics analysis.**

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

#### ***Option***

Label identifying the option to be performed:

**ADD**

Add a rotated nodal coordinate constraint (default).

**DELE**

Delete specified rotated nodal coordinate constraints.

**LIST**

List all rotated nodal coordinate constraints.

**CID**

Coordinate system ID for which rotated nodal constraints will be added or deleted. The *CID* must have been previously defined with the **EDLCS** command. If *Option* = DELE, use *CID* = ALL to delete all previously specified nodal constraints.

**Cname**

Nodal component set to which the rotated coordinate constraint will be applied. *Cname* must be previously specified using the **CM** command.

**DOF1, DOF2, DOF3, . . . , DOF6**

Degrees of freedom for which the rotated nodal constraint will be applied. Valid degree of freedom labels include UX, UY, UZ, ROTX, ROTY, and ROTZ. If *DOF1* = ALL, rotated nodal constraints will be applied to all degrees of freedom.

## Notes

Constraints applied with **EDNROT** are zero displacement constraints.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Rotated Nodal**

**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Rotated Nodal>Delete All**

**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Rotated Nodal>Delete Individ**

**Main Menu>Solution>Constraints>Apply>Rotated Nodal**

**Main Menu>Solution>Constraints>Delete>Rotated Nodal>Delete All**

**Main Menu>Solution>Constraints>Delete>Rotated Nodal>Delete Individ**

---

## EDOPT, *Option*, --, *Value*

**Specifies the type of output for an explicit dynamics analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**Option**

Label identifying the option to be performed:

**ADD**

Define an output type specification (default).

**DELE**

Delete an output type specification.

**LIST**

List the current output type specification.

--

Unused field.

**Value**

Label identifying the type of output that the LS-DYNA solver should produce:

**ANSYS**

Write results files for the ANSYS postprocessors (default). The files that will be written are Job-name .RST and Jobname .HIS (see "Notes" (p. 481) below).

**LSDYNA**

Write results files for the LS-DYNA postprocessor (LS-POST). The files that will be written are D3PLOT, and files specified by **EDOUT** and **EDHIST** (see "Notes" (p. 481) below).

**BOTH**

Write results files for both ANSYS and LS-DYNA postprocessors.

**Command Default**

Output is written for the ANSYS postprocessors only.

**Notes**

By default, LS-DYNA will write the ANSYS results file Jobname .RST (see the **EDRST** command.) If Job-name .HIS is desired, you must also issue **EDHIST**.

*Value* = LSDYNA or BOTH will cause LS-DYNA to write results files for the LS-POST postprocessor. The D3PLOT file is always written for these two options. If other LS-POST files are desired, you must issue the appropriate **EDHIST** and **EDOUT** commands.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Solution>Output Controls>Output File Types**

**EDOUT**, *Option*

**Specifies time-history output (ASCII format) for an explicit dynamics analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <>

*Option*

Output data option. Each option corresponds to a separate file that is written by the LS-DYNA solver. If *Option* = ALL, all files except NODOUT and ELOUT are written. Valid options are:

**GLSTAT**

Global data (default).

**BNDOUT**

Boundary condition forces and energy.

**RWFORC**

Wall force.

**DEFORC**

Discrete element data.

**MATSUM**

Material energies data.

**NCFORC**

Nodal interface forces.

**RCFORC**

Resultant interface force data.

**DEFGEO**

Deformed geometry data.

**SPCFORC**

SPC reaction force data.

**SWFORC**

Nodal constraint reaction force data (spotwelds and rivets).

**RBDOUT**

Rigid body data.

**GCEOUT**

Geometry contact entities.

**SLEOUT**

Sliding interface energy.

**JNTFORC**

Joint force data.

**NODOUT**

Nodal data.

**ELOUT**

Element data.

## **Command Default**

None of the above output is written.

## **Notes**

This command specifies output to be written during an explicit dynamics solution. The data corresponding to each *Option* is written to a separate ASCII file having the same name as the *Option* label. The data is written for the entire model at time intervals specified by the **EDHTIME** command. If no time interval is specified, output is written at 1000 steps over the analysis. (See also the **EDHIST** command which specifies time-history output for a portion of the model.) The data written to the MATSUM file is actually for each PART number (**EDPART**) at time intervals specified by the **EDHTIME** command, but the data is listed following the Mat no. in the file.

For *Option* = NODOUT and ELOUT, you must specify a component; you must issue **EDHIST** before issuing **EDOUT,NODOUT** or **EDOUT,ELOUT**.

Use **EDOUT,LIST** to list the current time-history output specifications. (The listing will include output requested with the **EDHIST** command.) Use **EDOUT,DELE** to delete all output specifications that have been defined with the **EDOUT** command.

In order for the specified output files to be written, you must also request that explicit dynamics results be written to an LS-DYNA output file [**EDWRITE,LSDYNA** or **EDWRITE,BOTH**].

In an explicit dynamic small restart analysis (**EDSTART,2**) or full restart analysis (**EDSTART,3**), the same ASCII files that were requested for the original analysis are written by default for the restart. You can request different files by issuing the appropriate **EDOUT** commands in the restart analysis.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>Output Controls>ASCII Output

**EDPART**, *Option*, *PARTID*, *Cname*

Configures parts for an explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <>

### *Option*

Option used to organize parts. (No default; *Option* must be specified.)

#### CREATE

Creates new PART IDs assigned to groups of elements with unique combinations of MAT, TYPE, and REAL set numbers. If this option is issued repeatedly, the part list is overwritten, except for PART IDs created with the ADD option. Remaining fields are ignored for this option.

#### UPDATE

Updates the PART IDs for the element groups without changing the order of the existing part list. If elements are redefined (or new elements are created) with different MAT, TYPE, or REAL set numbers, then use this option to create an updated list of PART IDs. Remaining fields are ignored for this option.

#### ADD

Assigns a user-specified PART ID (*PARTID*) to the elements contained in the element component *Cname*, or to the currently selected set of elements if *Cname* = ALL. Use this option to assign a specific PART ID to an element group that has the same combination of MAT, TYPE, and REAL set numbers. An UPDATE operation is automatically performed on the currently selected set of elements immediately following the ADD operation.

#### DELE

Deletes a PART ID assigned by the ADD option. *PARTID* is also required. An UPDATE operation is automatically performed on the currently selected set of elements immediately following the DELE operation.

## LIST

Lists the PART IDs for the element groups. The part list consists of five columns of numbers, one each for PART, MAT, TYPE, and REAL numbers, and one to indicate if the PART ID is used (including how many elements use it). The part list is based on the last CREATE or UPDATE operation. Remaining fields are ignored for this option.

## PARTID

A positive integer to be used as PART ID for the elements specified by *Cname* (no default). The number input must not be currently used for an existing part (except when *Option* = DELE). Any previously defined PART IDs for the elements, whether assigned by the user or created by ANSYS LS-DYNA, will be overwritten. The user-specified PART ID will not be changed by subsequent **EDPART,CREATE** or **EDPART,UPDATE** commands.

## Cname

Element component name for user-specified PART ID definition (*Option* = ADD). If *Cname* = ALL (default), all currently selected elements are considered for the part. The elements in the element component (or the currently selected set of elements if *Cname* = ALL or blank) must have the same combination of MAT, TYPE, and REAL set numbers, or the ADD option will be ignored.

## Notes

Certain ANSYS LS-DYNA commands (such as **EDCGEN**, **EDLOAD**, **EDREAD**, etc.) refer to PART IDs. You must define PART IDs (**EDPART,CREATE** or **EDPART,ADD**) before using these commands.

If parts are repeatedly created using *Option* = CREATE, the part list is continuously overwritten. This may cause problems for previously defined commands that reference a part number that has changed. To avoid this problem, the part list should be updated (*Option* = UPDATE) rather than recreated to obtain the current part list.

**EDPART,ADD** allows you to assign a specific part number to a group of elements instead of a number generated by the ANSYS LS-DYNA program. The user-specified PART IDs will not be changed by subsequent **EDPART,CREATE** or **EDPART,UPDATE** commands. Thus, you can use **EDPART,ADD** to specify PART IDs for some element groups, and use **EDPART,CREATE** or **EDPART,UPDATE** to assign PART IDs for the remaining element groups. Use **EDPART,DELE** to delete a PART ID generated by the ADD option. In this case, ANSYS LS-DYNA will generate a new PART ID for those elements associated with the deleted PART ID.

After creating or updating the part list, use **EDPART,LIST** to list the PART IDs and choose the correct one for use with other ANSYS LS-DYNA commands. For a detailed discussion on PART IDs, see [The Definition of Part](#) in the *ANSYS LS-DYNA User's Guide*.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Parts Options

---



**EDPC**, *MIN*, *MAX*, *INC***Selects and plots explicit dynamic contact entities.**

PREP7: Explicit Dynamics

&lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; DY &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt;

**MIN**

Minimum contact entity number to be selected and plotted (default = 1).

**MAX**Maximum contact entity number to be selected and plotted (default = *MIN*).**INC**

Contact entity number increment (default = 1).

**Notes**

**EDPC** invokes an ANSYS macro which selects and plots explicit dynamic contact entities. The plot will consist of nodes or elements, depending on the method (node components or parts) that was used to define the contact surfaces (see the **EDCGEN** command). For single surface contact definitions, all external surfaces within the model are plotted.

**Note**

**EDPC** changes the selected set of nodes and elements. After plotting contact entities, you must reselect all nodes and elements (**NSEL** and **ESEL**) required for subsequent operations, such as **SOLVE**

Use the **EDCLIST** command to list the contact entity numbers for all defined contact.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>Preprocessor>LS-DYNA Options>Contact>Select and Plot

**EDPL**, *LDNUM***Plots a time dependent load curve in an explicit dynamic analysis.**

SOLUTION: Explicit Dynamics

&lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; DY &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt;

**LDNUM**

Load number.

## Notes

**EDPL** invokes an ANSYS macro which produces a load vs. time graph for a load defined with the **EDLOAD** command. Only one load curve can be plotted at a time. Use **EDLOAD,LIST** to obtain a list of loads and corresponding load numbers.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Plot Load Curve

Main Menu>Solution>Loading Options>Plot Load Curve

**EDPVEL**, *Option*, *PID*, *VX*, *VY*, *VZ*, *OMEGAX*, *OMEGAY*, *OMEGAZ*, *XC*, *YC*, *ZC*, *ANGX*, *ANGY*, *ANGZ*

**Applies initial velocities to parts or part assemblies in an explicit dynamic analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> <> <> <> <> <> <> <> <> <> <> <>

### *Option*

Label identifying the option to be performed.

#### **VGEN**

Define initial velocities for the part or part assembly based on translational velocities (relative to global Cartesian) and the rotational velocity about an arbitrary axis. For this option, use the fields *VX*, *VY*, *VZ* to specify the translational velocities, and use *OMEGAX*, *XC*, *YC*, *ZC*, *ANGX*, *ANGY*, *ANGZ* to specify the rotational velocity and the axis of rotation.

#### **VELO**

Define initial velocity for the part or part assembly based on translational velocities and nodal rotational velocities input relative to the global Cartesian axes. For this option, use the following fields to define the initial velocity: *VX*, *VY*, *VZ*, *OMEGAX*, *OMEGAY*, *OMEGAZ*.

#### **LIST**

List initial velocity for the part or part assembly specified by *PID*. If *PID* is blank, all initial velocities defined on parts and part assemblies are listed. Remaining fields are ignored for this option.

#### **DELE**

Delete initial velocity defined for the part or part assembly specified by *PID*. If *PID* is blank, all initial velocities defined on parts and part assemblies are deleted. Remaining fields are ignored for this option.

### *PID*

Part ID or part assembly ID to which the initial velocity is to be applied. The part or assembly ID must be defined (**EDPART** or **EDASMP**) before issuing this command.

### *VX*

Initial velocity in X direction. Defaults to 0.

### *VY*

Initial velocity in Y direction. Defaults to 0.

**VZ**

Initial velocity in Z direction. Defaults to 0.

**OMEGAX**

For *Option* = VGEN, *OMEGAX* is the initial rotational velocity of the part or part assembly about the specified rotational axis. For *Option* = VELO, *OMEGAX* is the initial nodal rotational velocity about the X-axis. *OMEGAX* defaults to 0.

**OMEGAY**

Initial nodal rotational velocity about the Y-axis (used only if *Option* = VELO). Defaults to 0.

**OMEGAZ**

Initial nodal rotational velocity about the Z-axis (used only if *Option* = VELO). Defaults to 0.

The remaining fields are used only if *Option* = VGEN.

**XC**

X coordinate on rotational axis. Defaults to 0.

**YC**

Y coordinate on rotational axis. Defaults to 0.

**ZC**

Z coordinate on rotational axis. Defaults to 0.

**ANGX**

Angle (in degrees) of the rotational axis relative to the global X-axis. Defaults to 0.

**ANGY**

Angle (in degrees) of the rotational axis relative to the global Y-axis. Defaults to 0.

**ANGZ**

Angle (in degrees) of the rotational axis relative to the global Z-axis. Defaults to 0.

## Notes

You cannot mix the two methods of initial velocity input (*Option* = VELO and *Option* = VGEN) in the same analysis. You must use only one method for all initial velocity definitions.

The VGEN and VELO methods differ in how the rotational velocity is defined. Use *Option* = VGEN to input the initial velocities of a rotating part or part assembly. Use *Option* = VELO to apply the rotations directly to the nodes' rotation degrees of freedom. Since only shell and beam elements have rotation degrees of freedom, the rotations input with *Option* = VELO are only applicable to [SHELL163](#) and [BEAM161](#) elements. The rotational velocities input with *Option* = VELO are ignored for nodes not having rotational degrees of freedom (such as nodes attached to a [SOLID164](#) or [SOLID168](#) element).

It is normally acceptable to mix nodes belonging to deformable bodies and rigid bodies in the part assembly used in an initial velocity definition. However, when defining initial velocities in an implicit-to-explicit sequential solution, this is not an acceptable practice. In order for the initial velocities to be defined correctly in this type of analysis, you must define the initial velocities on the deformable body nodes separately from the initial velocities on the rigid body nodes.

Issuing the **EDPVEL** command again for the same part or part assembly (*PID*) will overwrite previous initial velocities defined for that part or part assembly.

To set the initial velocities to zero, issue the **EDPVEL** command with only the *Option* (use VELO or VGEN) and *PID* fields specified.

In a small restart analysis (**EDSTART,2**), you can only use the *Option* = VELO method to change initial velocities. When used in a small restart, the command **EDPVEL,VELO** changes the velocity of the specified part or part assembly. If you don't change the velocity of the parts and assemblies, their velocity at the beginning of the restart will be the same as the velocity at the end of the previous analysis.

Except for the LIST option, the **EDPVEL** command is not supported in a full restart analysis (**EDSTART,3**). You can list initial velocities defined in the previous analysis with the command **EDPVEL,LIST**. However, you cannot change initial velocities for parts that existed in the previous analysis; their velocity at the beginning of the analysis will be the same as the velocity at the end of the previous analysis. In addition, you cannot define initial velocities for any parts that are added in the full restart; the velocity of new parts will be zero.

To apply initial velocities to node components or nodes, use the **EDVEL** command.

You can use **EDPVEL** and **EDVEL** in the same analysis. If a node or node component input on the **EDVEL** command shares common nodes with a part or part assembly input on the **EDPVEL** command, the initial velocities defined on the common nodes will be determined by the last command input.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Parts>Delete**

**Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Parts>List**

**Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Parts>w/Axial Rotate**

**Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Parts>w/Nodal Rotate**

**Main Menu>Solution>Initial Velocity>On Parts>Delete**

**Main Menu>Solution>Initial Velocity>On Parts>List**

**Main Menu>Solution>Initial Velocity>On Parts>w/Axial Rotate**

**Main Menu>Solution>Initial Velocity>On Parts>w/Nodal Rotate**

---

**EDRC**, *Option*, *NRBF*, *NCSF*, *--*, *DTMAX*

**Specifies rigid/deformable switch controls in an explicit dynamic analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### *Option*

Label identifying option to be performed.

#### **ADD**

Define rigid/deformable controls (default).

#### **DELE**

Delete rigid/deformable controls.

#### **LIST**

List rigid/deformable controls.

**NRBF**

Flag to delete/activate nodal rigid bodies. If nodal rigid bodies or generalized weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities.

**0**

No change from previous status (default).

**1**

Delete.

**2**

Activate.

**NCSF**

Flag to delete/activate nodal constraint set. If nodal constraint/spotweld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities.

**0**

No change from previous status (default).

**1**

Delete.

**2**

Activate.

**--**

Unused field.

**TDMAX**

Maximum allowed time step after restart (no default).

**Command Default**

No rigid/deformable controls are defined.

**Notes**

This command is only valid in an explicit dynamic small restart analysis (**EDSTART**,2). Use this command when you do a rigid/deformable switch (**EDRD** command) and you want to control constraints defined by other means for the deformable body (such as nodal constraints or a weld). For example, if a deformable body has nodal constraints defined and it is switched to a rigid body, the nodal constraints should be deleted since they are invalid for the rigid body. Later on, if you want to switch the rigid body to deformable again and retain the nodal constraints, you can use **EDRC** to activate the constraints previously defined for the deformable body. Otherwise, the nodal constraints remain deactivated.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Solution>Rigid-Deformable>Controls**

**EDRD**, *Option*, *PART*, *MRB*

**Switches a part from deformable to rigid or from rigid to deformable in an explicit dynamic analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**Option**

Label identifying the option to be performed.

**D2R**

Change specified part from deformable to rigid (default).

**R2D**

Change specified part from rigid to deformable. Use this option to switch a part back to a deformable state after it has been changed to rigid using **EDRD**,**D2R**.

**LIST**

List parts that are flagged to change from deformable to rigid or rigid to deformable.

**PART**

Part number for part to be changed (no default).

**MRB**

Part number of the master rigid body to which the part is merged. *MRB* is used only if *Option* = **D2R**. If *MRB* = 0 (which is the default), the part becomes an independent rigid body.

**Command Default**

No parts are switched.

**Notes**

This command is valid in a new explicit dynamic analysis or in a restart. It is only possible to switch parts (**D2R** or **R2D**) in a restart if part switching is first activated in the original analysis. If part switching is not required in the original analysis but will be used in the restart, you must issue **EDRD**,**D2R** with no further arguments in the original analysis. You can use the **EDRI** command to define inertia properties for newly created rigid bodies (**D2R**).

Parts that are defined as rigid using **EDMP**,**RIGID** are permanently rigid and cannot be changed to deformable.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Solution>Rigid-Deformable>Switch**

**EDREAD**, *NSTART*, *Label*, *NUM*, *STEP1*, *STEP2*

**Reads explicit dynamics output into variables for time-history postprocessing.**

POST26: Set Up

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

***NSTART***

Starting reference number assigned to the first variable. Allowed range is 2 (the default) to *NV* [**NUMVAR**]. (*NV* defaults to 30 for an explicit dynamics analysis.)

***Label***

Label identifying the output file to be read. No default.

**GLSTAT**

Read data from the GLSTAT file.

**MATSUM**

Read data from the MATSUM file.

**SPCFORC**

Read data from the SPCFORC file.

**RCFORC**

Read data from the RCFORC file.

**SLEOUT**

Read data from the SLEOUT file.

**NODOUT**

Read data from the NODOUT file.

**RBDOUT**

Read data from the RBDOUT file.

***NUM***

Number identifying the data set to be read in (defaults to 1). If *Label* = GLSTAT, *NUM* is ignored. If *Label* = MATSUM or RBDOUT, *NUM* is the PART number [**EDPART**] for which output is desired. If *Label* = SPCFORC or NODOUT, *NUM* is the node number for which output is desired. If *Label* = SLEOUT or RCFORC, *NUM* is the number of the contact entity for which output is desired.

***STEP1*, *STEP2***

Load step range of data to be read in. If *STEP1* and *STEP2* are blank, all load steps are read in.

**Notes**

**EDREAD** reads data from the specified ascii output file so that it may be used during postprocessing. After **EDREAD**, you must issue the **STORE** command to store the data in time history variables. Once stored, the variables can be viewed as plots of output item versus time.

The number of variables stored depends on the file specified. The following table shows the items in each file and the order in which they are stored. If data items were previously stored in variables *NSTART* to *NSTART*+15, they will be overwritten. If more variables are needed, change *NV* on the **NUMVAR** command. (Note that hourglass energy will not be available if it was not specified for the model [**EDENERGY**,1].)

The following items under MATSUM are listed in the MATSUM ASCII file (in the Mat no. field) for each part number at time intervals specified by the **EDHTIME** command. Use **EDREAD**,MATSUM,NUM to specify the part number that corresponds to the mat number in the MATSUM file.

Resultant contact forces and sliding interface energies are available from the RCFORC and SLEOUT files, respectively. The RCFORC file is written for surface based contact types that include target and contact (master and slave) definitions. You should ensure that this file contains valid force results before issuing **EDREAD,,RCFORC**. Only the resultant contact forces on the *master* surface are available for time-history postprocessing.

Variable Number	GLSTAT	MATSUM	SP-CFORC	RCFORC	SLEOUT
NSTART	Time step	Internal energy	X force	X force	Slave energy
NSTART+1	Kinetic energy	Kinetic energy	Y force	Y force	Master energy
NSTART+2	Internal energy	X-momentum	Z force	Z force	Total slave energy
NSTART+3	Spring and damper energy	Y-momentum	X moment	--	Total master energy
NSTART+4	System damping energy	Z-momentum	Y moment	--	Total energy
NSTART+5	Sliding interface energy	X-rigid-body-velocity	Z moment	--	--
NSTART+6	External work	Y-rigid-body-velocity	--	--	--
NSTART+7	Eroded kinetic energy	Z-rigid-body-velocity	--	--	--
NSTART+8	Eroded internal energy	Hourglass energy	--	--	--
NSTART+9	Total energy	--	--	--	--
NSTART+10	Total energy/initial energy	--	--	--	--
NSTART+11	Energy ratio w/o eroded energy	--	--	--	--
NSTART+12	Global X velocity	--	--	--	--
NSTART+13	Global Y velocity	--	--	--	--
NSTART+14	Global Z velocity	--	--	--	--
NSTART+15	Hourglass energy	--	--	--	--

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>TimeHist Postpro>Read LSDYNA Data>GLSTAT file**  
**Main Menu>TimeHist Postpro>Read LSDYNA Data>MATSUM file**  
**Main Menu>TimeHist Postpro>Read LSDYNA Data>NODOUT file**  
**Main Menu>TimeHist Postpro>Read LSDYNA Data>RBDOUT file**  
**Main Menu>TimeHist Postpro>Read LSDYNA Data>RCFORC file**  
**Main Menu>TimeHist Postpro>Read LSDYNA Data>SLEOUT file**  
**Main Menu>TimeHist Postpro>Read LSDYNA Data>SPCFORC file**



**EDRI**, *Option, PART, XC, YC, ZC, TM, IXX, IYY, IZZ, IXY, IYZ, IXZ*

**Defines inertia properties for a new rigid body that is created when a deformable part is switched to rigid in an explicit dynamic analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**Option**

Label identifying the option to be performed.

**ADD**

Define inertia for specified part (default).

**DELE**

Delete inertia definition for specified part.

**LIST**

List inertia definitions.

**PART**

Part number for which inertia is defined (no default).

**XC, YC, ZC**

X, Y, and Z-coordinates of the center of mass (no defaults).

**TM**

Translational mass (no default).

**IXX, IYY, IZZ, IXY, IYZ, IXZ**

Components (xx, yy, etc.) of inertia tensor. *IXX*, *IYY*, and *IZZ* must be input (no defaults). *IXY*, *IYZ*, and *IXZ* default to zero.

## Command Default

Inertia properties are calculated by the program for all parts switched from deformable to rigid.

## Notes

Use this command to define inertia properties for a rigid body that is created when a deformable part is switched to rigid (using the **EDRD**,D2R command) in an explicit dynamic analysis. If these properties are not defined, LS-DYNA will compute the new rigid body properties from the finite element mesh (which requires an accurate mesh representation of the body). When rigid bodies are merged to a master rigid body, the inertia properties defined for the master rigid body apply to all members of the merged set.

**EDRI** can only be issued in a new analysis. Therefore, if you are going to use inertia properties in a subsequent restart analysis, you must issue **EDRI** in the original analysis for the part that will later be switched to rigid in the restart.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>Rigid-Deformable>Inertia Property

**EDRST**, *NSTEP*, *DT*

**Specifies the output interval for an explicit dynamic analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

***NSTEP***

Number of steps at which output is written to the results file (Jobname .RST). Defaults to 100. When you specify *NSTEP*, *NSTEP*+2 results are written to the Jobname .RST file. The time interval between output is  $TIME / NSTEP$ , where *TIME* is the analysis end-time specified on the **TIME** command. Do not specify a value of *NSTEP* = 0.

***DT***

Time interval at which output is written to the results file (Jobname .RST). If *NSTEP* is input, *DT* is ignored.

**Command Default**

Output will be written to Jobname .RST at 100 steps over the analysis.

**Notes**

You can use *NSTEP* or *DT* to specify the output interval to be used for Jobname .RST. You should not specify both quantities; if both are input, *NSTEP* will be used.

In an explicit dynamic small restart (**EDSTART**,2) or full restart analysis (**EDSTART**,3), the **EDRST** setting will default to the *NSTEP* or *DT* value used in the original analysis. You can issue **EDRST** in the restart to change this setting.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>Solution>Output Controls>File Output Freq>Number of Steps

Main Menu>Solution>Output Controls>File Output Freq>Time Step Size

**EDRUN**, *Option*, *Cons*, *Ncpu*

**Specify LS-DYNA serial or parallel processing.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

***Option***

LS-DYNA processing option

**SERIAL**

Use serial processing (default)

**SMP**

Use Shared Memory Parallel processing

**CONS**

Consistency setting (only applicable when *Option* = SMP)

**0**

Result consistency is not required (default)

**1**

Result consistency is required

**NCPU**

Number of processors to use (applicable only with *Option* = SMP)

## Command Default

The command default is serial processing.

## Notes

The **EDRUN** command specifies either serial (one CPU) processing or shared (multiple CPU) memory parallel processing (SMP). When using SMP, the calculations may be executed in a different order, depending on CPU availability and the load on each CPU. You may therefore see slight differences in the results when running the same job multiple times, either with the same number or a different number of processors. Comparing nodal accelerations often shows wider discrepancies. To avoid such differences, you can specify that consistency be maintained by setting *CONS* = 1. Maintaining consistency can result in an increase of up to 15 percent in CPU time.

The parallel processing setting is only effective when you have multiple CPUs *and* licenses for the appropriate number of ANSYS LS-DYNA SMP tasks. If your site does not meet both requirements, the **EDRUN** command sets serial processing, regardless of command settings.

For more information on using SMP, see [Solution Features](#) in the *ANSYS LS-DYNA User's Guide*.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Analysis Options>Parallel Option**

**EDSHELL**, *WPAN*, *SHNU*, *SHTC*, *WPBT*, *SHPL*, *ITRST*

**Specifies shell computation controls for an explicit dynamics analysis.**

**SOLUTION: Explicit Dynamics**

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <>

**WPAN**

Maximum shell element warpage angle in degrees. Defaults to 20.

**SHNU**

Hughes-Liu shell normal update option:

**-2**

Unique nodal fibers. This option is required for **SHELL163** (KEYOPT(1) = 1, 6, or 7) if the real constant NLOC = 1 or -1.

**-1**  
Compute normals each cycle (default). This option is recommended.

**1**  
Compute on restarts.

***n***  
Compute every *n*th substep.

**SHTC**

Shell thickness change option:

**0**  
No change.

**1**  
Membrane straining causes thickness change. Important in sheet metal forming (default).

**WPBT**

Warping stiffness option for Belytschko-Tsay shells:

**1**  
Belytschko-Wong-Chiang warping stiffness added. This option is recommended.

**2**  
Belytschko-Tsay warping stiffness (default).

**SHPL**

Shell plasticity option. This option is only valid for these material models: strain rate independent plastic kinematic, strain rate dependent plasticity, power law plasticity, and piecewise linear plasticity.

**1**  
Iterative plasticity with 3 secant iterations (default).

**2**  
Full iterative plasticity.

**3**  
Radial return noniterative plasticity. (Use this option with caution; it may lead to inaccurate results.)

**ITRST**

Triangular shell sorting option. If sorting is on, degenerate quadrilateral shell elements are treated as triangular shells.

**1**  
Full sorting (default).

**2**  
No sorting.

## Notes

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Shell Elem Ctrl**

---

## EDSOLV

Specifies "explicit dynamics solution" as the subsequent status topic.

[SOLUTION: Explicit Dynamics](#)

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### Notes

This is a status [\[STAT\]](#) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the [STAT](#) command should immediately follow this command.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Menu Paths

**Utility Menu>List>Status>LS-DYNA**

**EDSP**, *Option*, *MIN*, *MAX*, *INC*

Specifies small penetration checking for contact entities in an explicit dynamic analysis.

[PREP7: Explicit Dynamics](#)

<> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

#### *Option*

Label identifying the option to be performed (no default).

#### **ON**

Turn small penetration checking on for specified contact entities.

#### **OFF**

Turn small penetration checking off for specified contact entities.

#### **LIST**

List current setting for penetration checking.

#### **MIN**

Minimum contact entity number for which to turn on/off small penetration check (default = 1).

#### **MAX**

Maximum contact entity number for which to turn on/off small penetration check (defaults to *MIN*).

#### **INC**

Contact entity number increment (default = 1).

### Command Default

Penetration checking is determined by *PENCHK* on the **EDCONTACT** command.



## Notes

**EDSTART** can be issued before the **SOLVE** command to specify a new analysis, a simple restart, a small restart, or a full restart as described below.

*New analysis:* For a new analysis, you do not need to issue **EDSTART** unless you want to change the *MEMORY* or *FSIZE* option. If you do not specify the dump file name, `d3dump01` will be created by default.

*Simple restart:* This option assumes that the database has not been altered. Upon restarting, results will be appended to the existing results files. Issue **EDSTART,1,,,d3dumpnn** to indicate which restart file to use as a starting point. The dump file to be used must have been created in an earlier run and must be available at the time this command is issued. You would typically use a simple restart when you interrupt the LS-DYNA run via CTRL-C and terminate the run prematurely by issuing the "sense switch control" key SW1 (see [Solution Control and Monitoring](#) in the *ANSYS LS-DYNA User's Guide*). At this point you should be able to view the partial solution using ANSYS postprocessors. After you are done viewing the partial solution, you can reenter the solution processor and issue **EDSTART,1,,,d3dumpnn**, followed by **SOLVE** to continue with the analysis. The results will be appended to the results files `Jobname.RST` and `Jobname.HIS`. You can perform multiple simple restarts by issuing **EDSTART,1,,,d3dumpnn** multiple times, as needed. The solutions in the `Jobname.RST` file will all be in load step number 1.

*Small restart:* This option can be used when minor changes in the database are necessary. For example, you can reset the termination time, reset the output interval, add displacement constraints, change initial velocities, switch parts from a deformable to rigid state, etc. (See [A Small Restart](#) in the *ANSYS LS-DYNA User's Guide* for a complete description of database items that can be changed.) Issue **EDSTART,2,,,d3dumpnn** followed by the commands required to change the database, then issue **SOLVE**. The results will be appended to the results files `Jobname.RST` and `Jobname.HIS`. You can perform multiple restarts by issuing **EDSTART,2,,,d3dumpnn** multiple times, as needed. The additional restart solutions will be stored in `Jobname.RST` as load step numbers 2, 3, etc.

*Full restart:* A full restart is appropriate when many modifications to the database are required. For example, you can change the model geometry, apply different loading conditions, etc. Issue **EDSTART,3,,,d3dumpnn** to denote a full restart analysis. The `Jobname` will automatically be changed to `Jobname_nn`, ( $nn = 01$  initially, and will be incremented each time **EDSTART,3** is issued for subsequent full restarts). After the **EDSTART** command, you can input any commands needed to change the database. (Most commands which are applicable to an ANSYS LS-DYNA new analysis are also applicable to full restart analysis. A few commands related to contact specifications, initial velocity definitions, and adaptive meshing are not supported.) Then issue the **EDIS** command to specify which portions of the model should be initialized in the full restart using results data from the `d3dumpnn` file. Finally, issue the **SOLVE** command. At this point, new results files, `Jobname_nn.RST` and `Jobname_nn.HIS`, will be created. Time and output intervals in the new results files are continuous from the previous results files; the time is not reset to zero. (See [A Full Restart](#) in the *ANSYS LS-DYNA User's Guide* for a complete description of a full restart analysis.)

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Analysis Options>Restart Option**

**EDTERM**, *Option*, *Lab*, *NUM*, *STOP*, *MAXC*, *MINC*

**Specifies termination criteria for an explicit dynamic analysis.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

### **Option**

Label identifying the option to be performed.

#### **ADD**

Define termination criteria (default).

#### **DELE**

Delete termination criteria.

#### **LIST**

List termination criteria.

### **Lab**

Label identifying the type of termination (no default).

#### **NODE**

Terminate solution based on nodal point coordinates. The analysis terminates when the current position of the specified node reaches either the maximum or minimum coordinate value (*STOP* = 1, 2, or 3), or when the node picks up force from any contact surface (*STOP* = 4).

#### **PART**

Terminate solution based on rigid body (part) displacements. The analysis terminates when the displacement of the center of mass of the specified rigid body reaches either the maximum or minimum value (*STOP* = 1, 2, or 3), or when the displacement magnitude of the center of mass is exceeded (*STOP* = 4).

### **NUM**

Node number (if *Lab* = NODE) or rigid body Part ID (if *Lab* = PART). (No default.)

### **STOP**

Criterion for stopping the solution (no default).

**1**

Global X-direction.

**2**

Global Y-direction.

**3**

Global Z-direction.

**4**

For *Lab* = NODE, stop the solution if contact occurs. For *Lab* = PART, stop the solution if the displacement magnitude is exceeded for the specified rigid body (use *MAXC* to define the displacement magnitude).

### **MAXC**

Maximum (most positive) coordinate value (*Lab* = NODE) or displacement (*Lab* = PART). *MAXC* defaults to 1.0e21

### **MINC**

Minimum (most negative) coordinate value (*Lab* = NODE) or displacement (*Lab* = PART). *MINC* defaults to -1.0e21.



## Command Default

No termination criteria are defined other than the termination time set on the **TIME** command.

## Notes

You may specify multiple termination criteria using **EDTERM**; the solution will terminate when any one of the criteria is satisfied, or when the solution end time (specified on the **TIME** command) is reached.

In an explicit dynamic small restart analysis (**EDSTART,2**) or full restart analysis (**EDSTART,3**), the termination criteria set in the previous analysis (the original analysis or the previous restart) are carried over to the restart. If the previous analysis terminated due to one of these criteria, that specific criterion must be modified so that it will not cause the restart to terminate prematurely. In particular, if a termination condition based on nodal contact ( $Lab = NODE, STOP = 4$ ) is satisfied, this condition must be deleted and replaced with a condition based on nodal coordinates for that same node. (If a condition based on nodal coordinates already exists for that node, the replacement is not necessary.) In the restart, the number of termination criteria specified using **EDTERM** cannot exceed a maximum of 10 or the number specified in the original analysis.

Note that the termination criteria set by **EDTERM** are not active during dynamic relaxation (**EDDRELAX**).

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Analysis Options>Criteria to Stop>List**

**Main Menu>Solution>Analysis Options>Criteria to Stop>On a Node**

**Main Menu>Solution>Analysis Options>Criteria to Stop>On a Part**

---

## EDTP, OPTION, VALUE1, VALUE2

**Plots explicit elements based on their time step size.**

**SOLUTION: Explicit Dynamics**

<> <> <> <> <> <> <> <> <> <> DY <> <> <>

### OPTION

Plotting option (default = 1).

- 1**  
Plots the *elements* with the smallest time step sizes. The number of elements plotted and listed is equal to *VALUE1* (which defaults to 100). Each element is shaded red or yellow based on its time step value (see "*Notes*" (p. 502) for details).
- 2**  
Produces the same plot as for *OPTION* = 1, and also produces a list of the plotted elements and their corresponding time step values.
- 3**  
Produces a plot similar to *OPTION* = 1, except that all selected elements are plotted. Elements beyond the first *VALUE1* elements are blue and translucent. The amount of translucency is specified by

*VALUE2*. This option also produces a list of the first *VALUE1* elements with their corresponding time step values.

**VALUE1**

Number of elements to be plotted and listed (default = 100). For example, if *VALUE1* = 10, only the elements with the 10 smallest time step sizes are plotted and listed.

**VALUE2**

Translucency level ranging from 0 to 1 (default = 0.9). *VALUE2* is only used when *OPTION* = 3, and only for the elements plotted in blue. To plot these elements as non-translucent, set *VALUE2* = 0.

**Notes**

EDTP invokes an ANSYS macro that plots and lists explicit elements based on their time step size. For *OPTION* = 1 or 2, the number of elements plotted is equal to *VALUE1* (default = 100). For *OPTION* = 3, all selected elements are plotted.

The elements are shaded red, yellow, or blue based on their time step size. Red represents the smallest time step sizes, yellow represents the intermediate time step sizes, and blue represents the largest time step sizes. For example, if you specify *VALUE1* = 30, and if T1 is the smallest critical time step of all elements and T30 is the time step of the 30th smallest element, then the elements are shaded as follows:

Red - time step range is T1 to T1 + [0.05\*(T30--T1)]

Yellow - time step range is T1 + [0.05\*(T30--T1)] to T30

Blue (translucent) - time step range is > T30

Translucent blue elements only appear when *OPTION* = 3.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Solution>Time Controls>Time Step Prediction**

**EDVEL**, *Option*, *Cname*, *VX*, *VY*, *VZ*, *OMEGAX*, *OMEGAY*, *OMEGAZ*, *XC*, *YC*, *ZC*, *ANGX*, *ANGY*, *ANGZ*

**Applies initial velocities to nodes or node components in an explicit dynamic analysis.**

**SOLUTION: Explicit Dynamics**

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <> <>

**Option**

Label identifying the option to be performed.

**VGEN**

Define initial velocities based on translational velocities (relative to global Cartesian) and the rotational velocity about an arbitrary axis. For this option, use the fields *VX*, *VY*, *VZ* to specify the translational velocities, and use *OMEGAX*, *XC*, *YC*, *ZC*, *ANGX*, *ANGY*, *ANGZ* to specify the rotational velocity and the axis of rotation.

**VELO**

Define initial velocity based on translational velocities and nodal rotational velocities input relative to the global Cartesian axes. For this option, use the following fields to define the initial velocity:  
*VX, VY, VZ, OMEGAX, OMEGAY, OMEGAZ.*

**LIST**

List initial velocity for the component or node specified by *Cname*. If *Cname* is blank, all initial velocities defined on nodes and node components are listed. Remaining fields are ignored for this option.

**DELE**

Delete initial velocity defined for the component or node specified by *Cname*. If *Cname* is blank, all initial velocities defined on nodes and node components are deleted. Remaining fields are ignored for this option.

***Cname***

Name of existing component [**CM**] or node number to which the initial velocity is to be applied. If a component is used, it must consist of nodes.

***VX***

Initial velocity in X direction. Defaults to 0.

***VY***

Initial velocity in Y direction. Defaults to 0.

***VZ***

Initial velocity in Z direction. Defaults to 0.

***OMEGAX***

For *Option* = VGEN, *OMEGAX* is the initial rotational velocity of the component (or node) about the specified rotational axis. For *Option* = VELO, *OMEGAX* is the initial nodal rotational velocity about the X-axis. *OMEGAX* defaults to 0.

***OMEGAY***

Initial nodal rotational velocity about the Y-axis (used only if *Option* = VELO). Defaults to 0.

***OMEGAZ***

Initial nodal rotational velocity about the Z-axis (used only if *Option* = VELO). Defaults to 0.

The remaining fields are used only if *Option* = VGEN.

***XC***

X coordinate on rotational axis. Defaults to 0.

***YC***

Y coordinate on rotational axis. Defaults to 0.

***ZC***

Z coordinate on rotational axis. Defaults to 0.

***ANGX***

Angle (in degrees) of the rotational axis relative to the global X-axis. Defaults to 0.

***ANGY***

Angle (in degrees) of the rotational axis relative to the global Y-axis. Defaults to 0.

***ANGZ***

Angle (in degrees) of the rotational axis relative to the global Z-axis. Defaults to 0.

## Notes

You cannot mix the two methods of initial velocity input (*Option* = VELO and *Option* = VGEN) in the same analysis. You must use only one method for all initial velocity definitions.

The VGEN and VELO methods differ in how the rotational velocity is defined. Use *Option* = VGEN to input the initial velocities of a rotating component. Use *Option* = VELO to apply the rotations directly to the nodes' rotation degrees of freedom. Since only shell and beam elements have rotation degrees of freedom, the rotations input with *Option* = VELO are only applicable to [SHELL163](#) and [BEAM161](#) elements. The rotational velocities input with *Option* = VELO are ignored for nodes not having rotational degrees of freedom (such as nodes attached to a [SOLID164](#) or [SOLID168](#) element).

It is normally acceptable to mix nodes belonging to deformable bodies and rigid bodies in the nodal component used in an initial velocity definition. However, when defining initial velocities in an implicit-to-explicit sequential solution, this is not an acceptable practice. In order for the initial velocities to be defined correctly in this type of analysis, you must define the initial velocities on the deformable body nodes separately from the initial velocities on the rigid body nodes.

Issuing the **EDVEL** command again for the same component or node (*Cname*) will overwrite previous initial velocities defined for that component or node.

To set the initial velocities to zero, issue the **EDVEL** command with only the *Option* (use VELO or VGEN) and *Cname* fields specified.

In a small restart analysis ([EDSTART,2](#)), you can only use the *Option* = VELO method to change initial velocities. When used in a small restart, the command **EDVEL,VELO** changes the velocity of the specified nodes. If you don't change the velocity of the nodes, their velocity at the beginning of the restart will be the same as the velocity at the end of the previous analysis.

Except for the LIST option, the **EDVEL** command is not supported in a full restart analysis ([EDSTART,3](#)). You can list initial velocities defined in the previous analysis with the command **EDVEL,LIST**. However, you cannot change initial velocities for nodes or node components that existed in the previous analysis; their velocity at the beginning of the analysis will be the same as the velocity at the end of the previous analysis. In addition, you cannot define initial velocities for any nodes that are added in the full restart; the velocity of new nodes will be zero.

To apply initial velocities to parts or part assemblies, use the **EDPVEL** command.

You can use **EDPVEL** and **EDVEL** in the same analysis. If a node or node component input on the **EDVEL** command shares common nodes with a part or part assembly input on the **EDPVEL** command, the initial velocities defined on the common nodes will be determined by the last command input.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Nodes>Delete**

**Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Nodes>List**

**Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Nodes>w/Axial Rotate**

**Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Nodes>w/Nodal Rotate**

**Main Menu>Solution>Initial Velocity>On Nodes>Delete**

Main Menu>Solution>Initial Velocity>On Nodes>List  
 Main Menu>Solution>Initial Velocity>On Nodes>w/Axial Rotate  
 Main Menu>Solution>Initial Velocity>On Nodes>w/Nodal Rotate

**EDWELD**, *Option, NWELD, N1, N2, SN, SS, EXPN, EXPS, EPSF, TFAIL, NSW, CID*

Defines a massless spotweld or generalized weld for use in an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> DY <> <> <>

### *Option*

Label identifying the option to be performed:

#### **ADD**

Define a weld (default). This weld may be a spotweld between two nodes or a generalized weld. A massless spotweld will be defined if valid node numbers are specified in fields *N1* and *N2*. A generalized weld will be defined if a node component is specified in field *N1*.

#### **DELE**

Delete specified weld. If *NWELD* is blank, all welds are deleted.

#### **LIST**

List specified weld. If *NWELD* is blank, all welds are listed.

### **NWELD**

Reference number identifying the spotweld or generalized weld.

### **N1, N2**

For a spotweld, *N1* and *N2* are the nodes which are connected by the spotweld. For a generalized weld, input a nodal component name in *N1* and leave *N2* blank. The nodal component should contain all nodes that are to be included in the generalized weld.

### **SN**

Normal force at spotweld failure.

### **SS**

Shear force at spotweld failure.

### **EXPN**

Exponent for normal spotweld force.

### **EXPS**

Exponent for shear spotweld force.

### **EPSF**

Effective plastic strain at ductile failure (used only for a generalized weld).

### **TFAIL**

Failure time for constraint set (used only for a generalized weld); default = 1.0e20.

### **NSW**

Number of spot welds for the generalized weld.

### **CID**

Coordinate system ID number (CID) to be used for output data (used only for a generalized weld). The coordinate system must be previously defined with the **EDLCS** command.

## Notes

This command can be used to define a massless spotweld between two nodes or a generalized weld for a group of nodes. For a spotweld, the nodes specified by *N1* and *N2* must not be coincident. For a generalized weld, coincident nodes are permitted, but *CID* must be specified when using coincident nodes. **EDWELD** is not updated after a node merge operation; therefore, node merging [**NUMMRG**,**NODE**] should be done before any **EDWELD** definitions. Nodes connected by a spotweld or generalized weld cannot be constrained in any other way.

Failure of the weld occurs when:

$$\left(\frac{|f_n|}{S_n}\right)^{\text{expn}} + \left(\frac{|f_s|}{S_s}\right)^{\text{exps}} \geq 1$$

where  $f_n$  and  $f_s$  are normal and shear interface forces. Normal interface force  $f_n$  is nonzero for tensile values only.

You can graphically display spotwelds by issuing the command **/PBC,WELD,,1**.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Spotweld>Delete**  
**Main Menu>Preprocessor>LS-DYNA Options>Spotweld>Genrlizd Spotwld**  
**Main Menu>Preprocessor>LS-DYNA Options>Spotweld>List**  
**Main Menu>Preprocessor>LS-DYNA Options>Spotweld>Massless Spotwld**

**EDWRITE**, *Option*, *Fname*, *Ext*, --

**Writes explicit dynamics input to an LS-DYNA input file.**

SOLUTION: Explicit Dynamics

<> <> <> <> <> <> <> <> <> <> <> DY <> <> <>

### Option

Sets a flag in the LS-DYNA input file (*Fname*.*Ext*) to produce desired output.

#### ANSYS

Set a flag to write results files for the ANSYS postprocessors (default). The files that will be written are *Jobname*.*RST* and *Jobname*.*HIS* (see Notes below).

#### LSDYNA

Set a flag to write results files for the LS-DYNA postprocessor (LS-POST). The files that will be written are *D3PLOT*, and files specified by **EDOUT** and **EDHIST** (see Notes below).

#### BOTH

Set a flag to write results files for both ANSYS and LS-DYNA postprocessors.

***Fname***

File name and directory path (80 characters maximum, including directory; this limit is due to an LS-DYNA program limitation). If you do not specify a directory path, it will default to your working directory. The file name defaults to `Jobname`. Previous data on this file, if any, are overwritten.

***Ext***

Filename extension (8 character maximum).

The extension defaults to K in a new analysis and in a full restart analysis, and to R in a small restart analysis.

--

Unused field.

## Notes

This command writes an LS-DYNA input file for the LS-DYNA solver. **EDWRITE** is only valid if explicit dynamic elements have been specified. This command is not necessary if the LS-DYNA solver is invoked from within ANSYS, in which case `Jobname.K` (or `Jobname.R`) is written automatically when the solution is initiated. (If LS-DYNA is invoked from within ANSYS, use **EDOPT** to specify desired output.)

If the analysis is a small restart (**EDSTART,2**), the file that is written will have the name `Jobname.R` (by default) and will only contain changes from the original analysis.

If the analysis is a full restart (**EDSTART,3**), the file that is written will have the name `Jobname_nn.K` (by default) and will contain all the information from the database. In a full restart, the jobname is changed to `Jobname_nn` ( $nn = 01$  initially, and is incremented for each subsequent full restart.)

A command is included in the LS-DYNA input file to instruct the LS-DYNA solver to write the results files indicated by *Option*. By default, LS-DYNA will write the ANSYS results file `Jobname.RST` (see the **EDRST** command). If `Jobname.HIS` is desired, you must also issue **EDHIST**.

*Option* = `LSDYNA` or `BOTH` will cause LS-DYNA to write results files for the LS-POST postprocessor. The `D3PLOT` file is always written for these two options. If other LS-POST files are desired, you must issue the appropriate **EDHIST** and **EDOUT** commands.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>Write Jobname.k

---

## **/EFACET**, *NUM*

**Specifies the number of facets per element edge for PowerGraphics displays.**

POST1:Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NUM***

Number of facets per element edge for element plots.

- 1** Use 1 facet per edge (default for h-elements).
- 2** Use 2 facets per edge.
- 4** Use 4 facets per edge.

## Command Default

As stated above.

## Notes

**/EFACET** is valid only when PowerGraphics is enabled [**/GRAPHICS,POWER**], except that it can be used in FULL graphics mode for element **CONTA174**. (See the **/GRAPHICS** command and element **CONTA174** in the *Element Reference* for more information.) The **/EFACET** command is only applicable to element type displays.

**/EFACET** controls the fineness of the subgrid that is used for element plots. The element is subdivided into smaller portions called *facets*. Facets are piecewise linear surface approximations of the actual element face. In their most general form, facets are warped planes in 3-D space. A greater number of facets will result in a smoother representation of the element surface for element plots. **/EFACET** may affect results averaging. See *Contour Displays* in the *Basic Analysis Guide* for more information.

For midside node elements, use  $NUM = 2$ ; if  $NUM = 1$ , no midside node information is output. For non-midside node elements,  $NUM$  should be set to 1. See the **PLNSOL** and **PRNSOL** commands for more information.

With PowerGraphics active (**/GRAPHICS,POWER**), the averaging scheme for surface data with interior element data included (**AVRES,,FULL**) and multiple facets per edge (**/EFACET,2** or **/EFACET,4**) will yield differing minimum and maximum contour values depending on the Z-Buffering options (**/TYPE,,6** or **/TYPE,,7**). When the Section data is not included in the averaging schemes (**/TYPE,,7**), the resulting absolute value for the midside node is significantly smaller.

---

### Caution

*If you specify **/EFACET,1**, PowerGraphics does not plot midside nodes. You must use **/EFACET,2** to make the nodes visible.*

This command is valid in any processor.

## Menu Paths

**Main Menu>General Postproc>Options for Outp**  
**Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu**  
**Utility Menu>List>Results>Options**  
**Utility Menu>Plot>Results>Contour Plot>Nodal Solution**  
**Utility Menu>PlotCtrls>Multi-Plot Contrls**  
**Utility Menu>PlotCtrls>Style>Size and Shape**



**EGEN**, *ITIME*, *NINC*, *IEL1*, *IEL2*, *IEINC*, *MINC*, *TINC*, *RINC*, *CINC*, *SINC*, *DX*, *DY*, *DZ*

**Generates elements from an existing pattern.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ***ITIME*, *NINC***

Do this generation operation a total of *ITIMES*, incrementing all nodes in the given pattern by *NINC* each time after the first. *ITIME* must be >1 if generation is to occur. *NINC* may be positive, zero, or negative. If *DX*, *DY*, and/or *DZ* is specified, *NINC* should be set so any existing nodes (as on **NGEN**) are not overwritten.

#### ***IEL1*, *IEL2*, *IEINC***

Generate elements from selected pattern beginning with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* is negative, *IEL2* and *IEINC* are ignored and the last  $|IEL1|$  elements (in sequence backward from the maximum element number) are used as the pattern to be repeated. If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and use all selected elements [**ESEL**] as pattern to be repeated. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *INC* are ignored).

#### ***MINC***

Increment material number of all elements in the given pattern by *MINC* each time after the first.

#### ***TINC***

Increment type number by *TINC*.

#### ***RINC***

Increment real constant table number by *RINC*.

#### ***CINC***

Increment element coordinate system number by *CINC*.

#### ***SINC***

Increment section ID number by *SINC*.

#### ***DX*, *DY*, *DZ***

Define nodes that do not already exist but are needed by generated elements (as though the **NGEN**, *ITIME*, *INC*, *NODE1*, , , *DX*, *DY*, *DZ* were issued before **EGEN**). Zero is a valid value. If blank, *DX*, *DY*, and *DZ* are ignored.

## Notes

A pattern may consist of any number of previously defined elements. The MAT, TYPE, REAL, ESYS, and SECNUM numbers of the new elements are based upon the elements in the pattern and not upon the current specification settings.

You can use the **EGEN** command to generate interface elements (**INTER192**, **INTER193**, **INTER194**, and **INTER195**) directly. However, because interface elements require that the element connectivity be started from the bottom surface, you must make sure that you use the correct element node connectivity. See the element descriptions for **INTER192**, **INTER193**, **INTER194**, and **INTER195** for the correct element node definition.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Copy>Elements>Auto Numbered**

**\*EIGEN**, *Kmatrix*, *Mmatrix*, *Cmatrix*, *Evals*, *Evects*

**Performs a modal solution with unsymmetric or damping matrices.**

[APDL: Matrix Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Kmatrix*

Name of the stiffness matrix. May be a real or complex-valued matrix.

### *Mmatrix*

Name of the mass matrix.

### *Cmatrix*

Name of the damping matrix (used only for **MODOPT,DAMP**).

### *Evals*

Name of the output eigenvalues vector. It will be an  $m$ -long **\*VEC** vector of complex values, where  $m$  is the number of eigenvalues requested (**MODOPT**).

### *Evects*

Name of the output eigenvector matrix. It will be a  $n \times m$  **\*DMAT** (dense) matrix of complex values, where  $n$  is the size of the matrix and  $m$  is the number of eigenvalues requested (**MODOPT**).

## Notes

Use the command **ANTYPE,MODAL** and the **MODOPT** command to specify the modal solution options. Only **MODOPT,DAMP** and **MODOPT,UNSYM** are supported.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**EINTF**, *TOLER*, *K*, *TLAB*, *KCN*, *DX*, *DY*, *DZ*, *KNONROT*

**Defines two-node elements between coincident or offset nodes.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *TOLER*

Tolerance for coincidence (based on maximum Cartesian coordinate difference for node locations and on angle differences for node orientations). Defaults to 0.0001. Only nodes within the tolerance are considered to be coincident.

### *K*

Only used when the type of the elements to be generated is **PRETS179**.  $K$  is the pretension node that is common to the pretension section that is being created. If  $K$  is not specified, it will be created by ANSYS automatically and will have an ANSYS-assigned node number. If  $K$  is specified but does not already

exist, it will be created automatically but will have the user-specified node number.  $K$  cannot be connected to any existing element.

**TLAB**

Nodal number ordering. Allowable values are:

**LOW**

The 2-node elements are generated from the lowest numbered node to the highest numbered node.

**HIGH**

The 2-node elements are generated from the highest numbered node to the lowest numbered node.

**REVE**

Reverses the orientation of the selected 2-node element.

**KCN**

In coordinate system  $KCN$ , elements are created between node 1 and node 2 (= node 1 + dx dy dz)..

**DX, DY, DZ**

Node location increments that define the node offset in the active coordinate system (DR, D $\theta$ , DZ for cylindrical and DR, D $\theta$ , D $\Phi$  for spherical or toroidal).

**KNONROT**

When  $KNONROT = 0$ , the nodes coordinate system is not rotated. When  $KNONROT = 1$ , the nodes belonging to the elements created are rotated into coordinate system  $KCN$  (see **NROTAT** command description).

## Notes

Defines 2-node elements (such as gap elements) between coincident or offset nodes (within a tolerance). May be used, for example, to "hook" together elements interfacing at a seam, where the seam consists of a series of node pairs. One element is generated for each set of two coincident nodes. For more than two coincident or offset nodes in a cluster, an element is generated from the lowest numbered node to each of the other nodes in the cluster. If fewer than all nodes are to be checked for coincidence, use the **NSEL** command to select the nodes. Element numbers are incremented by one from the highest previous element number. The element type must be set [**ET**] to a 2-node element before issuing this command. Use the **CPINTF** command to connect nodes by coupling instead of by elements. Use the **CEINTF** command to connect the nodes by constraint equations instead of by elements.

For contact element **CONTA178**, the tolerance is based on the maximum Cartesian coordinate difference for node locations only. The angle differences for node orientations are not checked.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>At Coincid Nd**

**Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>Offset Nodes**

**EKILL**, *ELEM*

**Deactivates an element (for the birth and death capability).**

[SOLUTION: Birth and Death](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***ELEM***

Element to be deactivated. If ALL, deactivate all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*.

**Notes**

Deactivates the specified element when the birth and death capability is being used. A deactivated element remains in the model but contributes a near-zero stiffness (or conductivity, etc.) value (**ESTIF**) to the overall matrix. Any solution-dependent state variables (such as stress, plastic strain, creep strain, etc.) are set to zero. Deactivated elements contribute nothing to the overall mass (or capacitance, etc.) matrix.

The element can be reactivated with the **EALIVE** command.

ANSYS recommends using element deactivation/reactivation (**EKILL/EALIVE**) for linear elastic materials only. For all other materials, validate the results carefully before using them.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Birth & Death>Kill Elements**  
**Main Menu>Solution>Load Step Opts>Other>Birth & Death>Kill Elements**

**ELBOW**, *Transkey, TOL, Dof, Cons1, Cons2, Cons3, Cons4*

**Specifies degrees of freedom to be coupled for end release and applies section constraints to elbow elements.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Transkey***

Pipe-to-elbow transition flag:

**OFF**

Do not automatically transition pipes to elbows. (This behavior is the default.)

**ON**

Automatically convert straight **PIPE289** elements to **ELBOW290** elements where it is beneficial. The program converts elements in transition regions where curved **ELBOW290** elements are connected to straight **PIPE289** elements.

***TOL***

Angle tolerance (in degrees) between adjacent **ELBOW290** elements. The default value is 20. A value of -1 specifies *all* selected **ELBOW290** elements.

**Dof**

Degrees of freedom to couple:

**ALL**

Couple all nodal degrees of freedom (UX, UY, UZ, ROTX, ROTY, and ROTZ). This behavior is the default.

**BALL**

Create ball joints (equivalent to releasing ROTX, ROTY, and ROTZ).

**Cons1, Cons2, Cons3, Cons4**

Section degrees of freedoms to constrain. If *Cons1* through *Cons4* are unspecified, no section constraints are applied:

**SECT**

All section deformation

**SE**

Section radial expansion

**SO**

Section ovalization

**SW**

Section warping

**SRA**

Local shell normal rotation about cylindrical axis t2

**SRT**

Local shell normal rotation about cylindrical axis t1

## Notes

The **ELBOW** command specifies end releases and section constraints for **ELBOW290** elements and converts straight **PIPE289** elements to **ELBOW290** elements. The command works on currently selected nodes and elements. It creates end releases on any two connected elbow elements whose angle at connection exceeds the specified tolerance. From within the ANSYS GUI, the **Picked** node option generates an end release and section constraints at the selected node regardless of the angle of connection (that is, the angle tolerance [*TOL*] is set to -1).

Elbow and pipe elements must share the same section ID in order for the pipe-to-elbow transition to occur.

To list the elements altered by the **ELBOW** command, issue an **ELIST** command.

To list the coupled sets generated by the **ELBOW** command, issue a **CPLIST** command.

To list the section constraints generated by the **ELBOW** command, issue a **DLIST** command.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>End Releases>Elbows>On Selected set**  
**Main Menu>Solution>Load Step Opts>Other>End Releases>Elbows>On Selected set**

## ELEM

Specifies "Elements" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>Preprocessor>Elements**

---

**ELIST**, *IEL1*, *IEL2*, *INC*, *NNKEY*, *RKEY*, *PTKEY*

**Lists the elements and their attributes.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***IEL1*, *IEL2*, *INC***

Lists elements from *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *INC* (defaults to 1). If *IEL1* = ALL (default), *IEL2* and *INC* are ignored and all selected elements [**ESEL**] are listed. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *INC* are ignored).

***NNKEY***

Node listing key:

**0**

List attribute references and nodes.

**1**

List attribute references but not nodes.

***RKEY***

Real constant listing key:

**0**

Do not show real constants for each element.

**1**

Show real constants for each element. This includes default values chosen for the element.

***PTKEY***

LS-DYNA part number listing key (applicable to ANSYS LS-DYNA only):

**0**

Do not show part ID number for each element.

1

Show part ID number for each element.

## Notes

Lists the elements with their nodes and attributes (MAT, TYPE, REAL, ESYS, SECNUM, PART). See also the **LAYLIST** command for listing layered elements.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Elements>Attributes + RealConst  
 Utility Menu>List>Elements>Attributes Only  
 Utility Menu>List>Elements>Nodes + Attributes  
 Utility Menu>List>Elements>Nodes + Attributes + RealConst

## \*ELSE

Separates the final if-then-else block.

APDL: Process Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Optional final block separator within an if-then-else construct. See the **\*IF** for details. If a batch input stream hits an end-of-file during a false **\*IF** condition, the ANSYS run will not terminate normally. You will need to terminate it externally (use either the UNIX "kill" function or the Windows task manager). The **\*ELSE** command must appear on the same file as the **\*IF** command, and all five characters must be input.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

**\*ELSEIF**, *VAL1*, *Oper1*, *VAL2*, *Conj*, *VAL3*, *Oper2*, *VAL4*

Separates an intermediate if-then-else block.

APDL: Process Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *VAL1*

First numerical value (or parameter which evaluates to numerical value) in the conditional comparison operation. *VAL1*, *VAL2*, *VAL3*, and *VAL4* can also be character strings (enclosed in quotes) or parameters for *Oper* = EQ and NE only.

**Oper1**

Operation label. A tolerance of 1.0E-10 is used for comparisons between real numbers:

**EQ --**

Equal (for  $VAL1 = VAL2$ ).

**NE --**

Not equal (for  $VAL1 \neq VAL2$ ).

**LT --**

Less than (for  $VAL1 < VAL2$ ).

**GT --**

Greater than (for  $VAL1 > VAL2$ ).

**LE --**

Less than or equal (for  $VAL1 \leq VAL2$ ).

**GE --**

Greater than or equal (for  $VAL1 \geq VAL2$ ).

**ABLT --**

Absolute values of  $VAL1$  and  $VAL2$  before  $<$  operation.

**ABGT --**

Absolute values of  $VAL1$  and  $VAL2$  before  $>$  operation.

**VAL2**

Second numerical value (or parameter which evaluates to numerical value) in the conditional comparison operation.

**Conj**

(Optional) Connection between two logical clauses.

**AND -**

True if both clauses ( $Oper1$  and  $Oper2$ ) are true.

**OR -**

True if either clause is true.

**XOR -**

True if either (but not both) clause is true.

**VAL3**

(Optional) Third numerical value (or parameter which evaluates to numerical value).

**Oper2**

(Optional) Operation label. This will have the same labels as  $Oper1$ , except it uses  $Val3$  and  $Val4$ . A tolerance of 1.0E-10 is used for comparisons between real numbers.

**VAL4**

(Optional) Fourth numerical value (or parameter value which evaluates to a numerical value).

## Notes

Optional intermediate block separator within an if-then-else construct. All seven characters of the command name (\*ELSEIF) must be input. This command is similar to the \*IF command except that the *Base* field is not used. The \*IF, \*ELSEIF, \*ELSE, and \*ENDIF commands for each if-then-else construct must all be read from the same file (or keyboard).



This command is valid in any processor.

## Menu Paths

This command cannot be accessed from a menu.

## EMAGERR

Calculates the relative error in an electrostatic or electromagnetic field analysis.

POST1: Magnetics Calculations

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

## Notes

The relative error is an approximation of the mesh discretization error associated with a solution. It is based on the discrepancy between the unaveraged, element-nodal field values and the averaged, nodal field values. The calculation is valid within a material boundary and does not consider the error in continuity of fields across dissimilar materials.

For electrostatics, the field values evaluated are the electric field strength (EFSUM) and the electric flux density (DSUM). A relative error norm of each is calculated on a per-element basis and stored in the element table [ETABLE] with the labels EF\_ERR and D\_ERR. Normalized error values EFN\_ERR and DN\_ERR are also calculated and stored in the element table. Corresponding quantities for electromagnetics are H\_ERR, B\_ERR, HN\_ERR, and BN\_ERR, which are calculated from the magnetic field intensity (HSUM) and the magnetic flux density (BSUM). The normalized error value is the relative error norm value divided by the peak element-nodal field value for the currently selected elements.

Use the PLETAB and PRETAB commands to plot and list the error norms and normalized error values.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Error Eval

## EMATWRITE, Key

Forces the writing of all the element matrices to File.EMAT.

SOLUTION: Analysis Options

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

### Key

Write key:

#### YES

Forces the writing of the element matrices to File.EMAT even if not normally done.

#### NO

Element matrices are written only if required. This value is the default.

## Notes

The **EMATWRITE** command forces ANSYS to write the `File.EMAT` file. The file is necessary if you intend to perform a subsequent partial solution (**PSOLVE**) requiring `File.EMAT` data as a prerequisite, or if you intend to follow the initial load step with a subsequent inertia relief calculation (**IRLF**). If used in the solution processor (**/SOLU**), this command is only valid within the first load step.

`File.EMAT` is also required in these cases:

- For the static solution of a prestressed modal analysis when you use stress-stiffening energy to obtain better strain energy calculation accuracy.
- When you perform a QR damp eigensolution with a constant material damping coefficient (**MP,DMPR**).

This command is also valid in PREP7.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## EMF

**Calculates the electromotive force (emf), or voltage drop along a predefined path.**

[POST1: Magnetics Calculations](#)

MP ME <> <> <> <> <> EM <> <> PP <> EME MFS

## Notes

**EMF** invokes an ANSYS macro which calculates the electromotive force (emf), or voltage drop along a predefined path (specified with the **PATH** command). It is valid for both 2-D and 3-D electric field analysis or high-frequency electromagnetic field analysis. The calculated emf value is stored in the parameter EMF.

You must define a line path (via the **PATH** command) before issuing the **EMF** command macro. The macro uses calculated values of the electric field (EF), and uses path operations for the calculations. All path items are cleared when the macro finishes executing.

The **EMF** macro sets the "ACCURATE" mapping method and "MAT" discontinuity option on the **PMAP** command. The ANSYS program retains these settings after executing the macro.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>General Postproc>Elec&Mag Calc>Path Based>EMF**

---

## EMFT

Summarizes electromagnetic forces and torques.

[POST1: Magnetics Calculations](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

## Notes

Use this command to summarize electromagnetic force and torque in both static electric and magnetic problems. To use this command, select the nodes in the region of interest and make sure that all elements are selected. If **RSYS** = 0, the force is reported in the global Cartesian coordinate system. If **RSYS** ≠ 0, force is reported in the specified coordinate system. However, for torque, if **RSYS** ≠ 0, this command will account for the shift and rotation as specified by **RSYS**, but will report only the Cartesian components.

Forces are stored as items **\_FXSUM**, **\_FYSUM**, **\_FZSUM**, and **\_FSSUM**. Torque is stored as items **\_TXSUM**, **\_TYSUM**, **\_TZSUM**, and **\_TSSUM**.

This command is valid only with [SOLID117](#), [PLANE121](#), [SOLID122](#), [SOLID123](#), [PLANE233](#), [SOLID236](#) and [SOLID237](#) elements. For any other elements, you must use **FMAGSUM**.

## Menu Paths

Main Menu > General Postprocessor > Elec&Mag Calc > Summarize Force/Torque

## EMID, *Key, Edges*

Adds or removes midside nodes.

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM <> DY PP <> EME MFS

### Key

Add or remove key:

#### ADD

Add midside node to elements (default).

#### REMOVE

Remove midside nodes from elements.

### Edges

#### ALL

Add (or remove) midside nodes to (from) all edges of all selected elements, independent of which nodes are selected (default).

#### EITHER

Add (or remove) midside nodes only to (from) element edges which have either corner node selected.

#### BOTH

Add (or remove) midside nodes only to (from) element edges which have both corner nodes selected.

## Notes

This command adds midside nodes to (or removes midside nodes from) the selected elements. For this to occur, the selected elements must be midside node capable, the active element type [**TYPE**] must allow midside node capability, and the relationship between the finite element model and the solid model (if any) must first be disassociated [**MODMSH**].

By default, **EMID** generates a midside node wherever a zero (or missing) midside node occurs for that element. You can control this and add (or remove) midside nodes selectively by using the *Edges* argument. Nodes are located midway between the two appropriate corner nodes based on a linear Cartesian interpolation. Nodal coordinate system rotation angles are also linearly interpolated. Connected elements share the same midside node. Node numbers are generated sequentially from the maximum node number.

The **EMID** command is useful for transforming linear element types to quadratic element types having the same corner node connectivity.

**EMID** is also useful for transforming elements created outside of the program.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Add Mid Nodes**  
**Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Remove Mid Nd**

## **EMIS**, *MAT*, *EVALU*

**Specifies emissivity as a material property for the Radiation Matrix method.**

[AUX12: Radiation Matrix Method](#)

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

### **MAT**

Material number associated with this emissivity (500 maximum). Defaults to 1.

### **EVALU**

Emissivity for this material ( $0.0 < EVALU \leq 1.0$ ). Enter a very small number for zero.

## Command Default

Emissivity value of 1.0 is associated with all materials.

## Notes

Specifies emissivity as a material property for the Radiation Matrix method. This material property can then be associated with each element.

## Menu Paths

**Main Menu>Radiation Opt>Matrix Method>Emissivities**

**EMODIF**, *IEL, STLOC, I1, I2, I3, I4, I5, I6, I7, I8***Modifies a previously defined element.**[PREP7: Elements](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***IEL***

Modify nodes and/or attributes for element number *IEL*. If ALL, modify all selected elements [**ESEL**]. If *IEL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL*.

***STLOC***

Starting location (*n*) of first node to be modified or the attribute label. If *n*, modify element node positions *n, n+1*, etc. (*n* = 1 to 20). For example, if *STLOC* = 1, *I1* refers to the first node, *I2*, the second, etc. If *STLOC* = 9, *I1* refers to the ninth node, *I2*, the tenth, etc. Attributes are also modified to the currently specified values (use *-n* to modify only nodes and not attributes). If zero, modify only the attributes to the currently specified values. If MAT, TYPE, REAL, ESYS, or SECNUM, modify only that attribute to the *I1* value.

***I1, I2, I3, ..., I8***

Replace the previous node numbers assigned to this element with these corresponding values. A (blank) retains the previous value (except in the *I1* field, which resets the *STLOC* node number to zero). For attributes, replace the existing value with the *I1* value (or the default if *I1* is zero or blank).

**Notes**

The nodes and/or attributes (MAT, TYPE, REAL, ESYS, and SECNUM values) of an existing element may be changed with this command.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Modify Attrib**  
**Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Modify Nodes**

**EMORE**, *Q, R, S, T, U, V, W, X***Adds more nodes to the just-defined element.**[PREP7: Elements](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***Q, R, S, T, U, V, W, X***

Numbers of nodes typically assigned to ninth (node *Q*) through sixteenth (node *X*) nodal positions, if any. If *Q* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**Notes**

Repeat **EMORE** command for up to 4 additional nodes (20 maximum). Nodes are added after the last *nonzero* node of the element. Node numbers defined with this command may be zeroes.

## Menu Paths

This command cannot be accessed from a menu.

### EMSYM, *NSECT*

Specifies circular symmetry for electromagnetic sources.

[PREP7: Special Purpose](#)

MP ME ST <> <> <> <> <> EM EH DY PP <> EME MFS

#### *NSECT*

The number of circular symmetry sections (defaults to 1).

## Notes

Specifies the number of times to repeat electromagnetic sources for circular symmetry. Applies to [SOURC36](#) elements and to coupled-field elements with electric current conduction results in the database. Sources are assumed to be equally spaced over 360° about the global Cartesian Z axis.

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>Copy Sources**

**Main Menu>Solution>Load Step Opts>Magnetics>Options Only>Copy Sources**

### EMTGEN, *Ncomp, Ecomp, PNcomp, DOF, GAP, GAPMIN, FKN, EPZERO*

Generates a set of [TRANS126](#) elements.

[PREP7: Elements](#)

MP <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

#### *Ncomp*

Component name of the surface nodes of a structure which attach to the [TRANS126](#) elements. You must enclose name-strings in single quotes in the **EMTGEN** command line.

#### *Ecomp*

Component name of the [TRANS126](#) elements generated. You must enclose name-strings in single quotes in the **EMTGEN** command line. Defaults to EMTELM.

#### *PNcomp*

Component name of the plane nodes generated by the command at an offset (*GAP*) from the surface nodes. You must enclose name-strings in single quotes in the **EMTGEN** command line. Defaults to EMTPNO.

#### *DOF*

Active structural degree of freedom (DOF) for [TRANS126](#) elements (UX, UY, or UZ) in the Cartesian coordinate system. You must enclose the DOF in single quotes.

**GAP**

Initial gap distance from the surface nodes to the plane. Be sure to use the correct sign with respect to Ncomp node location.

**GAPMIN**

Minimum gap distance allowed (GAPMIN real constant) for **TRANS126** elements. Defaults to the absolute value of (GAP)\*0.05.

**FKN**

Contact stiffness factor used as a multiplier for a contact stiffness appropriate for bulk deformation. Defaults to 0.1.

**EPZERO**

Free-space permittivity. Defaults to 8.854e-6 (μMKS units).

## Notes

The **EMTGEN** command generates a set of **TRANS126** elements between the surface nodes of a moveable structure and a plane of nodes, typically representing a ground plane. The plane of nodes are created by the command at a specified offset (*GAP*). Each element attaches to a surface node and to a corresponding node representing the plane. The created elements are set to the augmented stiffness method (KEYOPT(6) = 1), which can help convergence. The generated plane nodes should be constrained appropriately for the analysis.

You can use **TRANS126** elements for simulating fully coupled electrostatic structural coupling between a MEMS device and a plane, if the gap distance between the device and the plane is small compared to the overall surface area dimensions of the device. This assumption allows for a point-wise closed-form solution of capacitance between the surface nodes and the plane; i.e.  $CAP = EPZERO * AREA / GAP$ , where EPZERO is the free-space permittivity, AREA is the area associated with the node, and GAP is the gap between the node and the plane. The area for each node is computed using the ARNODE function in ANSYS. See the **\*GET** command description for more information on the ARNODE function.

With a distributed set of **TRANS126** elements attached directly to the structure and a plane (such as a ground plane), you can perform a full range of coupled electrostatic-structural simulations, including:

- Static analysis (due to a DC voltage or a mechanical load)
- Prestressed modal analysis (eigenfrequencies, including frequency-shift effects of a DC bias voltage)
- Prestressed harmonic analysis (system response to a small-signal AC excitation with a DC bias voltage or mechanical load)
- Large signal transient analysis (time-transient solution due to an arbitrary time-varying voltage or mechanical excitation)

The **TRANS126** element also employs a node-to-node gap feature so you can perform contact-type simulations where the structure contacts a plane (such as a ground plane). The contact stiffness factor, FKN, is used to control contact penetration once contact is initiated. A smaller value provides for easier convergence, but with more penetration.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Transducers>Electromechanic>Node to Plane**

**EMUNIT**, *Lab*, *VALUE*

**Specifies the system of units for magnetic field problems.**

[PREP7: Materials](#)

MP ME <> <> <> <> <> EM EH <> PP <> EME MFS

***Lab***

Label specifying the type of units:

**MKS**

Rationalized MKS system of units (meters, amperes, henries, webers, etc.). Free-space permeability is set to  $4 \pi \times 10^{-7}$  henries/meter. Free-space permittivity is set to  $8.85 \times 10^{-12}$  F/m.

**MUZRO**

User defined system of units. Free-space permeability is set to the value input for *VALUE*. Other units must correspond to the permeability units. Relative permeability may be altered to absolute values.

**EPZRO**

User defined system of units. Free-space permittivity is set to the value input for *VALUE*. Other units must correspond to the permittivity units.

***VALUE***

User value of free-space permeability (defaults to 1) if *Lab* = MUZRO, or free-space permittivity (defaults to 1) if *Lab* = EPZRO.

***Command Default***

Rationalized MKS system of units (meters, amperes, henries, webers, etc.). Free-space permeability is set to  $4 \pi \times 10^{-7}$  Henries/meter, free-space permittivity is set to  $8.85 \times 10^{-12}$  Farads/meter.

**Notes**

Specifies the system of units to be used for electric and magnetic field problems. The free-space permeability and permittivity values may be set as desired. These values are used with the relative property values [\[MP\]](#) to establish absolute property values.

---

**Note**

If the magnetic source field strength ( $H_s$ ) has already been calculated [\[BIOT\]](#), switching **EMUNIT** will not change the values.

For micro-electromechanical systems (MEMS), where dimensions are on the order of microns, see the conversion factors in [System of Units](#) in the [Coupled-Field Analysis Guide](#).

This command is also valid in SOLUTION.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Electromag Units**

**Main Menu>Preprocessor>Material Props>Electromag Units**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Electromag Units**

---



**EN**, *IEL, I, J, K, L, M, N, O, P*

**Defines an element by its number and node connectivity.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**IEL**

Number assigned to element being defined. If *IEL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**I**

Number of node assigned to first nodal position (node I).

**J, K, L, M, N, O, P**

Number assigned to second (node J) through eighth (node P) nodal position, if any.

## Notes

Defines an element by its nodes and attribute values. Similar to the **E** command except it allows the element number (*IEL*) to be defined explicitly. Element numbers need not be consecutive. Any existing element already having this number will be redefined.

Up to 8 nodes may be specified with the **EN** command. If more nodes are needed for the element, use the **EMORE** command. The number of nodes required and the order in which they should be specified are described in the *Element Reference* for each element type. The current (or default) MAT, TYPE, REAL, SECNUM, and ESYS attribute values are also assigned to the element.

When creating elements with more than 8 nodes using this command and the **EMORE** command, it may be necessary to turn off shape checking using the **SHPP** command before issuing this command. If a valid element type can be created without using the additional nodes on the **EMORE** command, this command will create that element. The **EMORE** command will then modify the element to include the additional nodes. If shape checking is active, it will be performed before the **EMORE** command is issued. Therefore, if the shape checking limits are exceeded, element creation may fail before the **EMORE** command modifies the element into an acceptable shape.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Elements>User Numbered>Thru Nodes**

**\*END**

**Closes a macro file.**

[APDL: Macro Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Closes a file opened with **\*CREATE**. The **\*END** command is an 8-character command (to differentiate it from **\*ENDIF**). If you add commented text on that same line but do not allow enough spaces between **\*END** and the "!" that indicates the comment text, the **\*END** will attempt to interpret the "!" as the 8th character and will fail.

\*ENDDO

---

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## \*ENDDO

**Ends a do-loop and starts the looping action.**

[APDL: Process Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

One **\*ENDDO** is required for each nested do-loop. The **\*ENDDO** command must appear on the same file as the **\*DO** command, and all six characters must be input.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## \*ENDIF

**Ends an if-then-else.**

[APDL: Process Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Required terminator for the if-then-else construct. See the **\*IF** for details. If a batch input stream hits an end-of-file during a false **\*IF** condition, the ANSYS run will not terminate normally. You will need to terminate it externally (use either the UNIX “kill” function or the Windows task manager). The **\*ENDIF** command must appear on the same file as the **\*IF** command, and all six characters must be input.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**ENDRELEASE**, --, *TOLERANCE*, *Dof1*, *Dof2*, *Dof3*, *Dof4*

**Specifies degrees of freedom to be decoupled for end release.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

--

Unused field.

**TOLERANCE**

Angle tolerance (in degrees) between adjacent elements. Defaults to 20°. Set *TOLERANCE* to -1 to indicate all selected elements.

***Dof1*, *Dof2*, *Dof3*, *Dof4***

Degrees of freedom to release. If *Dof1* is blank, WARP is assumed and *Dof2*, *Dof3*, and *Dof4* are ignored.

**WARP**

Release the warping degree of freedom (default).

**ROTX**

Release rotations in the X direction.

**ROTY**

Release rotations in the Y direction.

**ROTZ**

Release rotations in the Z direction.

**UX**

Release displacements in the X direction.

**UY**

Release displacements in the Y direction.

**UZ**

Release displacements in the Z direction.

**BALL**

Create ball joints (equivalent to releasing WARP, ROTX, ROTY, and ROTZ).

## Notes

This command specifies end releases for the [BEAM188](#), [BEAM189](#), [PIPE288](#), and [PIPE289](#) elements. The command works on currently selected nodes and elements. It creates end releases on any two connected beam elements whose angle at connection exceeds the specified tolerance. From within the GUI, the **Picked node** option generates an end release at the selected node regardless of the angle of connection (angle tolerance is set to -1).

Use the [CPLIST](#) command to list the coupled sets generated by the **ENDRELEASE** command.

---

### Note

You should exercise due engineering judgement when using this command, as improper use may result in mechanics that render a solution impossible.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>End Releases>Beams>On Selected set  
Main Menu>Solution>Load Step Opts>Other>End Releases>Beams>On Selected set

**ENERSOL**, *NVAR*, *Item*, --, *Name*

Specifies the total energies to be stored.

[POST26: Set Up](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *NVAR*

Arbitrary reference number assigned to this variable (2 to NV).

### *Item*

#### **SENE**

Stiffness energy

#### **KENE**

Kinetic energy

#### **AENE**

Artificial energy due to hourglass control/drill stiffness

#### **STEN**

Artificial energy due to nonlinear stabilization

--

Unused field.

### *NAME*

A 32-character name identifying the item on printouts and displays. Defaults to a 4-character label formed by the four characters of the *Item* value.

## Menu Paths

This command cannot be accessed from a menu.

**ENGEN**, *IINC*, *ITIME*, *NINC*, *IEL1*, *IEL2*, *IEINC*, *MINC*, *TINC*, *RINC*, *CINC*, *SINC*, *DX*, *DY*, *DZ*

Generates elements from an existing pattern.

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *IINC*

Increment to be added to element numbers in pattern.

### *ITIME*, *NINC*

Do this generation operation a total of *ITIMES*, incrementing all nodes in the given pattern by *NINC* each time after the first. *ITIME* must be > 1 if generation is to occur. *NINC* may be positive, zero, or negative.

***IEL1, IEL2, IEINC***

Generate elements from the pattern that begins with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* is negative, *IEL2* and *IEINC* are ignored and use the last  $|IEL1|$  elements (in sequence backward from the maximum element number) as the pattern to be repeated. If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and all selected elements [ESEL] are used as the pattern to be repeated. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *IEINC* are ignored).

***MINC***

Increment material number of all elements in the given pattern by *MINC* each time after the first.

***TINC***

Increment type number by *TINC*.

***RINC***

Increment real constant table number by *RINC*.

***CINC***

Increment element coordinate system number by *CINC*.

***SINC***

Increment section ID number by *SINC*.

***DX, DY, DZ***

Define nodes that do not already exist but are needed by generated elements (**NGEN**, *ITIME*, *INC*, *NODE1*, , *DX*, *DY*, *DZ*). Zero is a valid value. If blank, *DX*, *DY*, and *DZ* are ignored.

**Notes**

Same as the **EGEN** command except it allows element numbers to be explicitly incremented (*IINC*) from the generated set. Any existing elements already having these numbers will be redefined.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Copy>Elements>User Numbered**

**ENORM, ENUM**

**Reorients shell element normals or line element node connectivity.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

***ENUM***

Element number having the normal direction that the reoriented elements are to match. If *ENUM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**Notes**

Reorients shell elements so that their outward normals are consistent with that of a specified element. **ENORM** can also be used to reorder nodal connectivity of line elements so that their nodal ordering is consistent with that of a specified element.

For shell elements, the operation reorients the element by reversing and shifting the node connectivity pattern. For example, for a 4-node shell element, the nodes in positions I, J, K and L of the original element are placed in positions J, I, L and K of the reoriented element. All 3-D shell elements in the selected set are considered for reorientation, and no element is reoriented more than once during the operation. Only shell elements adjacent to the lateral (side) faces are considered.

The command reorients the shell element normals on the same panel as the specified shell element. A *panel* is the geometry defined by a subset of shell elements bounded by free edges or T-junctions (anywhere three or more shell edges share common nodes).

Reorientation progresses within the selected set until either of the following conditions is true:

- The edge of the model is reached.
- More than two elements (whether selected or unselected) are adjacent to a lateral face.

In situations where unselected elements might undesirably cause case b to control, consider using **ENSYM,0,,0,ALL** instead of **ENORM**. It is recommended that reoriented elements be displayed and graphically reviewed.

You cannot use the **ENORM** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Real constant values are not reoriented and may be invalidated by an element reversal.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Shell Normals**

**ENSYM**, *IINC*, *--*, *NINC*, *IEL1*, *IEL2*, *IEINC*

**Generates elements by symmetry reflection.**

[PREP7:Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***IINC***

Increment to be added to element numbers in existing set.

***--***

Unused field.

***NINC***

Increment nodes in the given pattern by *NINC*.

***IEL1*, *IEL2*, *IEINC***

Reflect elements from pattern beginning with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and pattern is all selected elements [**ESEL**]. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *IEINC* are ignored).

## Notes

This command is the same as the **ESYM** command except it allows explicitly assigning element numbers to the generated set (in terms of an increment *IINC*). Any existing elements already having these numbers will be redefined.

The operation generates a new element by incrementing the nodes on the original element, and reversing and shifting the node connectivity pattern. For example, for a 4-node 2-D element, the nodes in positions I, J, K and L of the original element are placed in positions J, I, L and K of the reflected element.

Similar permutations occur for all other element types. For line elements, the nodes in positions I and J of the original element are placed in positions J and I of the reflected element. *In releases prior to ANSYS 5.5, no node pattern reversing and shifting occurred for line elements generated by **ENSYM**. To achieve the same results as you did in releases prior to ANSYS 5.5, use the **ENGEN** command instead.*

See the **ESYM** command for additional information about symmetry elements.

The **ENSYM** command also provides a convenient way to reverse shell element normals. If the *IINC* and *NINC* argument fields are left blank, the effect of the reflection is to reverse the direction of the outward normal of the specified elements. You cannot use the **ENSYM** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable. Also note that real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal. See [Revising Your Model](#) in the *Modeling and Meshing Guide* for more information about controlling element normals.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Reverse Normals>of Shell Elems**  
**Main Menu>Preprocessor>Modeling>Reflect>Elements>User Numbered**

## /EOF

**Exits the file being read.**

SESSION:Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Causes an end-of-file exit when encountered on a switched file (see **/INPUT**, **\*USE**, etc.). Commands are then read continuing from the file that contained the file switching command (or from the terminal if the switch was made from the terminal). Use the **/EXIT** command to terminate an ANSYS run. This command cannot be used in a do-loop or if-then-else construct.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

**EORIENT**, *Etype*, *Dir*, *TOLER***Reorients solid element normals.**PREP7: Meshing  
PREP7: Elements

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***Etype***

Specifies which elements to orient.

**LYSL**Specifies that certain solid elements (such as [SOLID185](#) with KEYOPT(3) = 1, [SOLID186](#) with KEYOPT(3) = 1, and [SOLSH190](#)) will be oriented. This value is the default.***Dir***The axis and direction for orientation, or an element number. If *Dir* is set to a positive number (*n*), then all eligible elements are oriented as similarly as possible to element *n*.**NEGX**

The element face with the outward normal most nearly parallel to the element coordinate system's negative x-axis is designated (reoriented) as face 1.

**POSX**

The element face with the outward normal most nearly parallel to the element coordinate system's positive x-axis is designated (reoriented) as face 1.

**NEGY**

The element face with the outward normal most nearly parallel to the element coordinate system's negative y-axis is designated (reoriented) as face 1.

**POSY**

The element face with the outward normal most nearly parallel to the element coordinate system's positive y-axis is designated (reoriented) as face 1.

**NEGZ**

(Default) The element face with the outward normal most nearly parallel to the element coordinate system's negative z-axis is designated (reoriented) as face 1.

**POSZ**

The element face with the outward normal most nearly parallel to the element coordinate system's positive z-axis is designated (reoriented) as face 1.

***TOLER***The maximum angle (in degrees) between the outward normal face and the target axis. Default is 90.0. Lower *TOLER* values will reduce the number of faces that are considered as the basis of element reorientation.**Notes****EORIENT** renumbers the element faces, designating the face most parallel to the XY plane of the element coordinate system (set with **ESYS**) as face 1 (nodes I-J-K-L, parallel to the layers in layered elements). It calculates the outward normal of each face and changes the node designation of the elements so the face with a normal most nearly parallel with and in the same general direction as the target axis becomes face 1.



The target axis, defined by *Dir*, is either the negative or positive indicated axis or the outward normal of face 1 of that element.

All **SOLID185** Layered Structural Solid, **SOLID186** Layered Structural Solid, and **SOLSH190** solid shell elements in the selected set are considered for reorientation.

After reorienting elements, you should always display and graphically review results using the **/ESHAPE** command. When plotting models with many or symmetric layers, it may be useful to temporarily reduce the number of layers to two, with one layer being much thicker than the other.

You cannot use **EORIENT** to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Prisms and tetrahedrals are also supported, within the current limitations of the **SOLID185**, **SOLID186**, and **SOLSH190** elements. (Layers parallel to the four-node face of the prism are not supported.)

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Orient Normal**

## EPLLOT

**Produces an element display.**

**PREP7:Elements**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Produces an element display of the selected elements. In full graphics, only those elements faces with all of their corresponding nodes selected are plotted. In PowerGraphics, all element faces of the selected element set are plotted irrespective of the nodes selected. However, for both full graphics and PowerGraphics, adjacent or otherwise duplicated faces of 3-D solid elements will not be displayed in an attempt to eliminate plotting of interior facets. See the **DSYS** command for display coordinate system issues.

This command will display curvature in midside node elements when PowerGraphics is activated [**/GRAPH-ICS,POWER**] and **/EFACET,2** or **/EFACET,4** are enabled. (To display curvature, two facets per edge is recommended [**/EFACET,2**]). When you specify **/EFACET,1**, PowerGraphics does not display midside nodes. **/EFACET** has no effect on **EPLLOT** for non-midside node elements.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking**  
**Main Menu>Solution>Time Controls>Time Step Prediction**  
**Utility Menu>Plot>Elements**

**EQSLV**, *Lab*, *TOLER*, *MULT*, *--*, *KeepFile*

**Specifies the type of equation solver.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN DS DSS <> EM EH <> PP <> EME MFS

### **Lab**

Equation solver type:

#### **SPARSE**

Sparse direct equation solver. Applicable to real-value or complex-value symmetric and unsymmetric matrices. Available only for STATIC, HARMIC (full method only), TRANS (full method only), SUBSTR, and PSD spectrum analysis types [[ANTYPE](#)]. Can be used for nonlinear and linear analyses, especially nonlinear analysis where indefinite matrices are frequently encountered. Well suited for contact analysis where contact status alters the mesh topology. Other typical well-suited applications are: (a) models consisting of shell/beam or shell/beam and solid elements (b) models with a multi-branch structure, such as an automobile exhaust or a turbine fan. This is an alternative to iterative solvers since it combines both speed and robustness. Generally, it requires considerably more memory (~10x) than the PCG solver to obtain optimal performance (running totally in-core). When memory is limited, the solver works partly in-core and out-of-core, which can noticeably slow down the performance of the solver. See the [BCSOPTION](#) command for more details on the various modes of operation for this solver.

This solver can be run in shared memory parallel or distributed memory parallel (Distributed ANSYS) mode. When used in Distributed ANSYS, this solver preserves all of the merits of the classic or shared memory sparse solver. The total sum of memory (summed for all processes) is usually higher than the shared memory sparse solver. System configuration also affects the performance of the distributed memory parallel solver. If enough physical memory is available, running this solver in the in-core memory mode achieves optimal performance. The ideal configuration when using the out-of-core memory mode is to use one processor per machine on multiple machines (a cluster), spreading the I/O across the hard drives of each machine, assuming that you are using a high-speed network such as Infiniband to efficiently support all communication across the multiple machines.

This solver supports use of the [GPU accelerator capability](#).

#### **JCG**

Jacobi Conjugate Gradient iterative equation solver, in-memory option. Available only for STATIC, HARMIC (full method only), and TRANS (full method only) analysis types [[ANTYPE](#)]. Can be used for structural and multiphysics applications. Applicable for symmetric, unsymmetric, complex, definite, and indefinite matrices. Recommended for 3-D harmonic analyses in structural and multiphysics applications. Efficient for heat transfer, electromagnetics, piezoelectrics, and acoustic field problems.

This solver can be run in shared memory parallel or distributed memory parallel (Distributed ANSYS) mode. When used in Distributed ANSYS, in addition to the limitations listed above, this solver is available only for STATIC and TRANS (full method) analyses where the stiffness is symmetric, and does not support the fast thermal option ([THOPT](#)).

This solver supports use of the [GPU accelerator capability](#). When using the GPU accelerator capability, in addition to the limitations listed above, this solver is available only for STATIC and TRANS (full method) analyses where the stiffness is symmetric, and does not support the fast thermal option ([THOPT](#)).

**ICCG**

Incomplete Cholesky Conjugate Gradient iterative equation solver. Available for STATIC, HARMIC (full method only), and TRANS (full method only) analysis types [**ANTYPE**]. Can be used for structural and multiphysics applications, and for symmetric, unsymmetric, complex, definite, and indefinite matrices. The ICCG solver requires more memory than the JCG solver, but is more robust than the JCG solver for ill-conditioned matrices.

**QMR**

Quasi-Minimal Residual iterative equation solver. Available for the HARMIC (full method only) analysis type [**ANTYPE**]. Can be used for high-frequency electromagnetic applications, and for symmetric, complex, definite, and indefinite matrices. The QMR solver is more stable than the ICCG solver.

**PCG**

Preconditioned Conjugate Gradient iterative equation solver (licensed from Computational Applications and Systems Integration, Inc.). Requires less disk file space than SPARSE and is faster for large models. Useful for plates, shells, 3-D models, large 2-D models, p-method analyses, and other problems having symmetric, sparse, definite or indefinite matrices for nonlinear analysis. Requires twice as much memory as JCG. Available only for analysis types [**ANTYPE**] STATIC, TRANS (full method only), or MODAL (with PCG Lanczos option only). Also available for the use pass of substructure analyses (**MATRIX50**). The PCG solver can robustly solve equations with constraint equations (**CE**, **CEINTF**, **CPINTF**, and **CERIG**). With this solver, you can use the **MSAVE** command to obtain a considerable memory savings.

The PCG solver can handle ill-conditioned problems by using a higher level of difficulty (see **PCGOPT**). Ill-conditioning arises from elements with high aspect ratios, contact, and plasticity.

This solver can be run in shared memory parallel or distributed memory parallel (Distributed ANSYS) mode. When used in Distributed ANSYS, this solver preserves all of the merits of the classic or shared memory PCG solver. The total sum of memory (summed for all processes) is about 30% more than the shared memory PCG solver.

This solver supports use of the [GPU accelerator capability](#).

**AMG**

Algebraic Multigrid iterative equation solver. Available for STATIC analyses and TRANS (full method only) analyses. The AMG solver is applicable to symmetric matrices. This solver is not available for substructure analyses, either generation or use pass. It is very efficient for single-field structural analyses (where the solution DOFs are combinations of UX, UY, UZ, ROTX, ROTY, and ROTZ). For applications such as single-field thermal analyses (where the solution DOF is TEMP), the AMG solver is less efficient. Recommended for ill-conditioned problems in which the ICCG solver would have difficulty converging; applicable in both single- and multiprocessor environments. In terms of CPU time when used in a single-processor environment, the AMG solver performs better than the ICCG solver for ill-conditioned problems, and it delivers about the same level of performance for ordinary problems. In a multiprocessor environment, the AMG solver scales better than the ICCG solver on shared memory parallel machines. Also handles indefinite matrix problems for nonlinear analyses. The AMG solver is part of ANSYS' High Performance Computing, which is a separately-licensed product. For detailed information on multiprocessor solvers, see [Improving ANSYS Performance and High Performance Computing](#) in the *ANSYS Advanced Analysis Techniques Guide*. See [Starting an ANSYS Session from Command Level](#) in the *Operations Guide* for information on how to specify the HPCHPC add-on at ANSYS start-up.

**TOLER**

Iterative solver tolerance value. Used only with the Jacobi Conjugate Gradient, Incomplete Cholesky Conjugate Gradient, Pre-conditioned Conjugate Gradient, Quasi-Minimal Residual, and Algebraic Multigrid

equation solvers. For the PCG solver, the default is 1.0E-8. The value 1.0E-5 may be acceptable in many situations. When using the PCG Lanczos mode extraction method, the default solver tolerance value is 1.0E-4. For the JCG and ICCG solvers with symmetric matrices, and for the AMG solver, the default is 1.0E-8. For the JCG and ICCG solvers with unsymmetric matrices, and for the QMR solver, the default is 1.0E-6. Iterations continue until the SRSS norm of the residual is less than *TOLER* times the norm of the applied load vector. For the PCG solver in the linear static analysis case, 3 error norms are used. If one of the error norms is smaller than *TOLER*, and the SRSS norm of the residual is smaller than 1.0E-2, convergence is assumed to have been reached. See [Iterative Solver](#) in the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for details.

---

## Note

When used with the Pre-conditioned Conjugate Gradient equation solver, *TOLER* can be modified between load steps (this is typically useful for nonlinear analysis).

If a *Lev\_Diff* value of 5 is specified on the **PCGOPT** command (either program- or user-specified), *TOLER* has no effect on the accuracy of the obtained solution from the PCG solver; a direct solver is used when *Lev\_Diff* = 5.

## MULT

Multiplier (defaults to 2.5 for nonlinear analyses; 1.0 for linear analyses) used to control the maximum number of iterations performed during convergence calculations. Used only with the Pre-conditioned Conjugate Gradient equation solver (PCG). The maximum number of iterations is equal to the multiplier (*MULT*) times the number of degrees of freedom (DOF). If *MULT* is input as a negative value, then the maximum number of iterations is equal to  $\text{abs}(MULT)$ . Iterations continue until either the maximum number of iterations or solution convergence has been reached. In general, the default value for *MULT* is adequate for reaching convergence. However, for ill-conditioned matrices (that is, models containing elements with high aspect ratios or material type discontinuities) the multiplier may be used to increase the maximum number of iterations used to achieve convergence. The recommended range for the multiplier is  $1.0 \leq MULT \leq 3.0$ . Normally, a value greater than 3.0 adds no further benefit toward convergence, and merely increases time requirements. If the solution does not converge with  $1.0 \leq MULT \leq 3.0$ , or in less than 10,000 iterations, then convergence is highly unlikely and further examination of the model is recommended. Rather than increasing the default value of *MULT*, consider increasing the level of difficulty (*Lev\_Diff*) on the **PCGOPT** command.

--

Unused field.

## KeepFile

Determines whether files from a SPARSE solver run should be deleted or retained. Applies only to *Lab* = SPARSE for static and full transient analyses.

### DELE

Deletes all files from the SPARSE solver run, including the factorized file, *.LN22*, upon **FINISH** or **/EXIT** (default).

### KEEP

Retains all necessary files from the SPARSE solver run, including the *.LN22* file, in the working directory.

## Command Default

The sparse direct solver is the default solver for all analyses, with the exception of modal/buckling analyses.

For modal/buckling analyses, there is no default solver. You must specify a solver with the **MODOPT** or **BUCOPT** command.

## Notes

The selection of a solver can affect the speed and accuracy of a solution. For a more detailed discussion of the merits of each solver, see [Solution](#) in the *Basic Analysis Guide*.

If you use **MODOPT**,LANB or **BUCOPT**,LANB, the internal solver used is **EQSLV**,SPARSE. If you use **MOD-OPT**,LANPCG, the internal solver used is **EQSLV**, PCG.

You may only specify the solver type in the first load step. You may, however, modify the solver tolerance in subsequent load steps for the iterative solvers.

Some solvers are not supported in Distributed ANSYS. For details, see the description of each solver given above and the Distributed ANSYS restrictions listed below. Distributed ANSYS requires HPC licenses. The GPU accelerator capability also requires HPC licenses. For more information, see the *Distributed ANSYS Guide*.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** The SPARSE and PCG solvers are fully supported in Distributed ANSYS and run a fully distributed solution. The JCG and AMG solvers are partially supported in Distributed ANSYS. The ICCG and QMR solvers are not supported.

## Product Restrictions

Command Option Lab	Available Products
SPARSE	MP ME ST PR PRN DS DSS <> EM EH <> PP <> EME MFS [1]
JCG	MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS
ICCG	MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS
QMR	MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS
PCG	MP ME ST PR PRN DS DSS <> EM EH <> PP <> EME MFS [1]
AMG	MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

1. The ANSYS DesignSpace (DS) and ANSYS DesignSpace - Structural (DSS) products do not support distributed solutions (Distributed ANSYS).

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Sol'n Options**

**Main Menu>Solution>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Sol'n Options**

## ERASE

Explicitly erases the current display.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

Similar to a hardware screen erase key. Useful during an "immediate" display to erase the screen without a replot so that the display continues on a clean screen. This action is automatically included in commands such as **NPLOT** and **EPLLOT**.

If the **/NOERASE** command is active, issuing the erase command will simply clear the display area. Subsequent replots will provide the display previously generated by the **/NOERASE** command.

This command is valid in any processor.

### Menu Paths

Utility Menu>PlotCtrls>Erase Options>Erase Screen

---

## **/ERASE**

Specifies that the screen is to be erased before each display.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Command Default

Previous **/ERASE** or **/NOERASE** setting will be used. Initially defaults to **/ERASE** setting.

### Notes

Erase occurs with the next display request, but before the display is actually started. **/NOERASE** can be used to suppress the automatic screen erase.

This command is valid in any processor.

### Menu Paths

Utility Menu>PlotCtrls>Erase Options>Erase Between Plots

---

**ERead**, *Fname*, *Ext*, --

**Reads elements from a file.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `ELEM` if *Fname* is blank.

--

Unused field.

## Notes

This read operation is not necessary in a standard ANSYS run but is provided as a convenience to users wanting to read a coded element file, such as from another mesh generator or from a CAD/CAM program. Data should be formatted as produced with the **EWRITE** command. If issuing **ERead** to acquire element information generated from ANSYS **EWRITE**, you must also issue **NREAD** before the **ERead** command. The element types [**ET**] must be defined before the file is read so that the file may be read properly. Only elements that are specified with the **ERRANG** command are read from the file. Also, only elements that are fully attached to the nodes specified on the **NRRANG** command are read from the file. Elements are assigned numbers consecutively as read from the file, beginning with the current highest database element number plus one. The file is rewound before and after reading. Reading continues until the end of the file.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Elements>Read Elem File**

**ERefine**, *NE1*, *NE2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

**Refines the mesh around specified elements.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NE1*, *NE2*, *NINC***

Elements (*NE1* to *NE2* in increments of *NINC*) around which the mesh is to be refined. *NE2* defaults to *NE1*, and *NINC* defaults to 1. If *NE1* = ALL, *NE2* and *NINC* are ignored and all selected elements are used for refinement. If *NE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NE1* (*NE2* and *NINC* are ignored).

***LEVEL***

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).



**DEPTH**

Depth of mesh refinement in terms of number of elements outward from the indicated elements, *NE1* to *NE2* (defaults to 0).

**POST**

Type of postprocessing to be done after element splitting, in order to improve element quality:

**OFF**

No postprocessing will be done.

**SMOOTH**

Smoothing will be done. Node locations may change.

**CLEAN**

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

**RETAIN**

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

**ON**

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

**OFF**

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

## Notes

**EREFINE** performs local mesh refinement around the specified elements. By default, the surrounding elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

**EREFINE** refines all area elements and tetrahedral volume elements that are adjacent to the specified elements. Any volume elements that are adjacent to the specified elements, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. If you have detached the mesh from the solid model, you must disable postprocessing cleanup or smoothing (*POST* = OFF) after the refinement to preserve the element attributes.

For additional restrictions on mesh refinement, see [Revising Your Model](#) in the *Modeling and Meshing Guide*.

This command is also valid for [rezoning](#).

## Menu Paths

**Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>All**

**Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Elements**



## EREINF

**Generates reinforcing elements from selected existing (base) elements.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

### Notes

The **EREINF** command generates reinforcing elements ([REINF264](#) and [REINF265](#)) directly from selected base elements (that is, existing standard elements in your model). The command scans all selected base elements and generates (if necessary) a compatible reinforcing element type for each base element. (ANSYS allows a combination of different base element types.)

Although predefining the reinforcing element type (**ET**) is not required, you must define the reinforcing element *section* type (**SECTYPE**); otherwise, ANSYS cannot generate the reinforcing element.

The **EREINF** command does not create new nodes. The reinforcing elements and the base elements share the common nodes.

Elements generated by this command are not associated with the solid model.

After the **EREINF** command executes, you can issue **ETLIST**, **ELIST**, and **EPLLOT** commands to verify the newly created reinforcing element types and elements.

Reinforcing elements do not account for any subsequent modifications made to the base elements. ANSYS recommends issuing the **EREINF** command only *after* the base elements are finalized. If you delete or modify base elements (via **EDELE**, **EMODIF**, **ETCHG**, **EMID**, **EORIENT**, **NUMMRG**, or **NUMCMP** commands, for example), remove all affected reinforcing elements and reissue the **EREINF** command to avoid inconsistencies.

### Menu Paths

**This command cannot be accessed from a menu.**

## ERESX, *Key*

**Specifies extrapolation of integration point results.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### Key

Extrapolation key:

#### DEFA

If element is fully elastic (no active plasticity, creep, or swelling nonlinearities), extrapolate the integration point results to the nodes. If any portion of the element is plastic (or other active material nonlinearity), copy the integration point results to the nodes (default).

#### YES

Extrapolate the linear portion of the integration point results to the nodes and copy the nonlinear portion (for example, plastic strains).

**NO**

Copy the integration point results to the nodes.

**Command Default**

Extrapolate integration point results to the nodes for all elements except those with active plasticity, creep, or swelling nonlinearities (default).

**Notes**

Specifies whether the solution results at the element integration points are extrapolated or copied to the nodes for element and nodal postprocessing. The structural stresses, elastic and thermal strains, field gradients, and fluxes are affected. Nonlinear data (plastic, creep, and swelling strains) are always copied to the nodes, never extrapolated. For shell elements, **ERESX** applies only to integration point results in the in-plane directions.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Integration Pt**

**Main Menu>Solution>Load Step Opts>Output Ctrls>Integration Pt**

**ERNORM**, *Key*

**Controls error estimation calculations.**

[POST1:Controls](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

**Key**

Control key:

**ON**

Perform error estimation (default). This option is not valid for PowerGraphics.

**OFF**

Do not perform error estimation.

**Command Default**

Error estimation calculations are performed by default unless PowerGraphics is enabled [[/GRAPHICS,POWER](#)].

**Notes**

Especially for thermal analyses, program speed increases if error estimation is suppressed. Therefore, it might be desirable to use error estimation only when needed. The value of the **ERNORM** key is not saved on `file.db`. Consequently, you need to reissue the **ERNORM** key after a [RESUME](#) if you wish to deactivate error estimation again.

## Menu Paths

Main Menu>General Postproc>Options for Outp  
Utility Menu>List>Results>Options

### ERRANG, *EMIN*, *EMAX*, *EINC*

Specifies the element range to be read from a file.

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *EMIN*, *EMAX*, *EINC*

Elements with numbers from *EMIN* (defaults to 1) to *EMAX* (defaults to 99999999) in steps of *EINC* (defaults to 1) will be read.

## Notes

Defines the element number range to be read [**ERead**] from the element file. If a range is also implied from the **NRRANG** command, only those elements satisfying both ranges will be read.

## Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Read Elem File

### ESCHECK, *Sele*, *Levl*, *Defkey*

Perform element shape checking for a selected element set.

[PREP7: Database  
SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

#### *Sele*

Specifies whether to select elements for checking:

##### **(blank)**

List all warnings/errors from element shape checking.

##### **ESEL**

Select the elements based on the *Levl* criteria specified below.

#### *Levl*

##### **WARN**

Select elements producing warning and error messages.

##### **ERR**

Select only elements producing error messages (default).

#### *Defkey*

Specifies whether check should be performed on deformed element shapes. .

- 0**  
Do not update node coordinates before performing shape checks (default).
- 1**  
Update node coordinates using the current set of deformations in the database.

## Notes

Shape checking will occur according to the current **SHPP** settings. Although **ESCHECK** is valid in all processors, *Defkey* uses the current results in the database. If no results are available a warning will be issued.

This command is also valid in PREP7, SOLUTION and POST1.

## Menu Paths

**Main Menu>General Postproc>Check Elem Shape>Sel Warning/Error Elements**

---

**ESEL**, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

**Selects a subset of elements.**

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.
- ALL**  
Restore the full set.
- NONE**  
Unselect the full set.
- INVE**  
Invert the current set (selected becomes unselected and vice versa).
- STAT**  
Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

### *Item*

Label identifying data, see [Table 147: ESEL - Valid Item and Component Labels \(p. 546\)](#). Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to ELEM. If *Item* = STRA (straightened),

elements are selected whose midside nodes do not conform to the curved line or non-flat area on which they should lie. (Such elements are sometimes formed during volume meshing [**VMESH**] in an attempt to avoid excessive element distortion.) You should graphically examine any such elements to evaluate their possible effect on solution accuracy.

**Comp**

Component of the item (if required). Valid component labels are shown in *Table 147: ESEL - Valid Item and Component Labels* (p. 546) below.

**VMIN**

Minimum value of item range. Ranges are element numbers, attribute numbers, load values, or result values as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored).

**VMAX**

Maximum value of item range. *VMAX* defaults to *VMIN* for input values.

For result values, *VMAX* defaults to infinity if *VMIN* is positive, or to zero if *VMIN* is negative.

**VINC**

Value increment within range. Used only with integer ranges (such as for element and attribute numbers). Defaults to 1. *VINC* cannot be negative.

**KABS**

Absolute value key:

**0**

Check sign of value during selection.

**1**

Use absolute value during selection (sign ignored).

## Command Default

All elements are selected.

## Notes

Selects elements based on values of a labeled item and component. For example, to select a new set of elements based on element numbers 1 through 7, use **ESEL,S,ELEM,,1,7**. The subset is used when the ALL label is entered (or implied) on other commands, such as **ELIST,ALL**. Only data identified by element number are selected. Selected data are internally flagged; no actual removal of data from the database occurs. Different element subsets cannot be used for different load steps [**SOLVE**] in a **/SOLU** sequence. The subset used in the first load step will be used for all subsequent load steps regardless of subsequent **ESEL** specifications.

This command is valid in any processor.

Elements crossing the named path (see **PATH** command) will be selected. This option is only available in PREP7 and POST1. If no geometry data has been mapped to the path (i.e., via **PMAP** and **PDEF** commands), the path will assume the default mapping option (**PMAP,UNIFORM**) to map the geometry prior to selecting the elements. If an invalid path name is given, the **ESEL** command is ignored (status of selected elements is unchanged). If there are no elements crossing the path, the **ESEL** command will return zero elements selected.

For selections based on non-integer numbers (coordinates, results, etc.), items that are within the range  $VMIN - Toler$  and  $VMAX + Toler$  are selected. The default tolerance  $Toler$  is based on the relative values of  $VMIN$  and  $VMAX$  as follows:

- If  $VMIN = VMAX$ ,  $Toler = 0.005 \times VMIN$ .
- If  $VMIN = VMAX = 0.0$ ,  $Toler = 1.0E-6$ .
- If  $VMAX \neq VMIN$ ,  $Toler = 1.0E-8 \times (VMAX - VMIN)$ .

Use the **SELTOL** command to override this default and specify  $Toler$  explicitly.

**Table 147 ESEL - Valid Item and Component Labels**

Valid Item and Component Labels ESEL, Type, Item, Comp, VMIN, VMAX, VINC, KABS		
Item	Comp	Description
ELEM		Element number.
ADJ		Elements adjacent to element $VMIN$ ( $VMAX$ and $VINC$ fields are ignored). Only elements (of the same dimensionality) adjacent to lateral faces are considered. Progression continues until edge of model or until more than two elements are adjacent at a face.
CENT	X, Y, Z	X, Y, or Z location in the active coordinate system.
TYPE		Element type number.
ENAME		Element name (or identifying number).
MAT		Material number.
REAL		Real constant number.
ESYS		Element coordinate system number.
PART		LS-DYNA part number (applicable only to ANSYS LS-DYNA)
LIVE		Active elements [ <b>EALIVE</b> ]. $VMIN$ and $VMAX$ fields are ignored, etc.
LAYER		Layer number (only composite elements with a nonzero thickness for the requested layer number are included). [ <b>LAYER</b> ]
SEC		Cross section ID number [ <b>SECNUM</b> ]
STRA		Straightened. See description under Argument Descriptions' Item field above.
SFE	PRES	Element pressure.
	CONV	Element convection bulk temperature.
	HFLUX	Element heat flux.
	FSI	Element (acoustic) fluid-structure interaction flag.
	IMPD	Element (acoustic) impedance.
	SHLD	Surface shielding properties on high-frequency elements. (Select logic works only on the value of conductivity, not relative permeability.)
	MXWF	Element Maxwell force flag.
	CHRG	Electric surface charge density.
	INF	Element infinite surface flag.

**Valid Item and Component Labels** *ESEL, Type, Item, Comp, VMIN, VMAX, VINC, KABS*

Item	Comp	Description
BFE	TEMP	Element temperature.
	FLUE	Element fluence.
	HGEN	Element heat generation rate.
	JS	Element current density, magnitude only.
	MVDI	Element magnetic virtual displacements flag.
	CHRGD	Electric charge density.
PATH	<i>Lab</i>	Selects all elements being crossed by the path with name <i>Lab</i> (see <b>PATH</b> command). If <i>Lab</i> = ALL then all elements related to all defined paths will be selected.
Valid item and component labels for element result values are:		
ETAB	<i>Lab</i>	Any user-defined element table label (see <b>ETABLE</b> command).

## Menu Paths

**Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements**

**Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts**

**Utility Menu>Select>Entities**

## /ESHAPE, SCALE, KEY

**Displays elements with shapes determined from the real constants or section definition.**

GRAPHICS:Style

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### SCALE

Scaling factor:

**0**

Use simple display of line and area elements. This value is the default.

**1**

Use real constants or section definition to form a solid shape display of the applicable elements.

### FAC

Multiply certain real constants, such as thickness, by *FAC* (where *FAC* > 0.01) and use them to form a solid shape display of elements.

### KEY

Current shell thickness key:

**0**

Use current thickness in the displaced solid shape display of shell elements (valid for **SHELL181**, **SHELL208**, **SHELL209**, and **SHELL281**). This value is the default.

**1**

Use initial thickness in the displaced solid shape display of shell elements.

## Command Default

Use simple display of line and area elements ( $SCALE = 0$ ).

## Notes

The **/ESHAPE** command allows beams, shells, current sources, and certain special-purpose elements to be displayed as solids with the shape determined from the real constants or section types. Elements are displayed via the **EPlot** command. No checks for valid or complete input are made for the display.

Following are details about using this command with various element types:

- **SOLID65** elements are displayed with internal lines that represent rebar sizes and orientations (requires vector mode [**/DEVICE**] with a basic type of display [**/TYPE,,BASIC**]). The rebar with the largest volume ratio in each element plots as a red line, the next largest as green, and the smallest as blue.
- **COMBIN14**, **COMBIN39**, and **MASS21** are displayed with a graphics icon, with the offset determined by the real constants and **KEYOPT** settings.
- **BEAM188**, **BEAM189**, **PIPE288**, **PIPE289** and **ELBOW290** are displayed as solids with the shape determined via the section-definition commands (**SECTYPE** and **SECDATA**). The arbitrary section option (*Subtype* = **ASEC**) has no definite shape and appears as a thin rectangle to show orientation. The elements are displayed with internal lines representing the cross-section mesh.

**SOLID272** and **SOLID273** are displayed as solids with the shape determined via the section-definition commands (**SECTYPE** and **SECDATA**). The 2-D master plane is revolved around the prescribed axis of symmetry.

Contour plots are available for these elements in postprocessing for PowerGraphics only (**/GRAPHICS,POWER**). To view 3-D deformed shapes for the elements, issue **OUTRES,MISC** or **OUTRES,ALL** for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, expand the modes with element results calculation ON (*Elcalc* = **YES** for **MXPAND**).

- **SOURC36**, **CIRCU124**, and **TRANS126** elements always plot using **/ESHAPE** when PowerGraphics is activated (**/GRAPHICS,POWER**).

In most cases, **/ESHAPE** renders a thickness representation of your shell, plane and layered elements more readily in PowerGraphics (**/GRAPHICS,POWER**). This type of representation employs PowerGraphics to generate the enhanced representation, and will often provide no enhancement in Full Graphics (**/GRAPHICS,FULL**). This is especially true for **POST1** results displays, where **/ESHAPE** is not supported for most element types with **FULL** graphics.

When PowerGraphics is active, **/ESHAPE** may degrade the image if adjacent elements have overlapping material, such as shell elements which are not co-planar. Additionally, if adjacent elements have different thicknesses, the polygons depicting the connectivity between the “thicker” and “thinner” elements along the shared element edges may not always be displayed.

For **POST1** results displays (such as **PLNSOL**), the following limitations apply:

- Rotational displacements for beam elements are used to create a more realistic displacement display. When **/ESHAPE** is active, displacement plots (via **PLNSOL,U,X** and **PLDISP**, for example) may disagree with your **PRNSOL** listings. This discrepancy will become more noticeable when the *SCALE* value is not equal to one.



- When shell elements are not co-planar, the resulting **PLNSOL** display with **/ESHAPE** will actually be a **PLESOL** display as the non-coincident pseudo-nodes are not averaged. Additionally, **/ESHAPE** should not be used with coincident elements because the plot may incorrectly average the displacements of the coincident elements.
- If you want to view solution results (**PLNSOL**, etc.) on layered elements (such as **SHELL181**, **SOLSH190**, **SOLID185** Layered Solid, **SOLID186** Layered Solid, **SHELL208**, **SHELL209**, **SHELL281**, and **ELBOW290**), set **KEYOPT(8) = 1** for the layer elements so that the data for all layers is stored in the results file.
- You can plot the through-thickness temperatures of elements **SHELL131** and **SHELL132** regardless of the thermal DOFs in use by issuing the **PLNSOL,TEMP** command (with PowerGraphics and **/ESHAPE** active).
- The **/ESHAPE,1** and **/ESHAPE,FAC** commands are incompatible with the **/CYCEXPAND** command used in cyclic symmetry analyses.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

### ESIZE, SIZE, NDIV

Specifies the default number of line divisions.

PREP7: Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### SIZE

Default element edge length on surface boundaries (i.e., lines). Divisions are automatically calculated (rounded upward to next integer) from line lengths. If *SIZE* is zero (or blank), use *NDIV*.

#### NDIV

Default number of element divisions along region boundary lines. Not used if *SIZE* is input.

## Notes

Specifies the default number of line divisions (elements) to be generated along the region boundary lines. The number of divisions may be defined directly or automatically calculated. Divisions defined directly for any line [**LESIZE**, **KESIZE**, etc.] are retained. For adjacent regions, the divisions assigned to the common line for one region are also used for the adjacent region. See the **MOPT** command for additional meshing options.

For free meshing operations, if smart element sizing is being used [**SMRTSIZE**] and **ESIZE,SIZE** has been specified, *SIZE* will be used as a starting element size, but will be overridden (i.e., a smaller size may be used) to accommodate curvature and small features.

This command is also valid for [rezoning](#).

## Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Size  
Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Adv Opts

**ESLA**, *Type*

Selects those elements associated with the selected areas.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of element select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

**Notes**

Selects area elements belonging to meshed [\[AMESH\]](#), selected [\[ASEL\]](#) areas.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>Select>Entities**

---

**ESLL**, *Type*

Selects those elements associated with the selected lines.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of element select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

## Notes

Selects line elements belonging to meshed [LMESH], selected [LSEL] lines.

This command is valid in any processor.

## Menu Paths

Utility Menu>Select>Entities

**ESLN**, *Type*, *EKEY*, *NodeType*

Selects those elements attached to the selected nodes.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of element selected:

**S**

Select a new set (default).

**R**

Reselect a set from the current set.

**A**

Additionally select a set and extend the current set.

**U**

Unselect a set from the current set.

### *EKEY*

Node set key:

**0**

Select element if *any* of its nodes are in the selected nodal set (default).

**1**

Select element only if *all* of its nodes are in the selected nodal set.

### *NodeType*

Label identifying type of nodes to consider when selecting:

**ALL**

Select elements considering all of their nodes (default).

**ACTIVE**

Select elements considering only their active nodes. An active node is a node that contributes DOFs to the model.

**INACTIVE**

Select elements considering only their inactive nodes (such as orientation or radiation nodes).

**CORNER**

Select elements considering only their corner nodes.

**MID**

Select elements considering only their midside nodes.

**Notes**

**ESLN** selects elements which have any (or all *EKEY*) *NodeType* nodes in the currently-selected set of nodes. Only elements having nodes in the currently-selected set can be selected.

This command is valid in any processor.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements**

**Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts**

**Utility Menu>Select>Entities**

---

**ESLV**, *Type*

Selects elements associated with the selected volumes.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Type*

Label identifying the type of element selected:

**S**

Select a new set (default).

**R**

Reselect a set from the current set.

**A**

Additionally select a set and extend the current set.

**U**

Unselect a set from the current set.

**Notes**

Selects volume elements belonging to meshed [**VMESH**], selected [**VSEL**] volumes.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>Select>Entities**

---

**ESOL**, *NVAR*, *ELEM*, *NODE*, *Item*, *Comp*, *Name*

**Specifies element data to be stored from the results file.**

POST26: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NVAR***

Arbitrary reference number assigned to this variable (2 to *NV* [**NUMVAR**]). Overwrites any existing results for this variable.

***ELEM***

Element for which data are to be stored. If *ELEM* = P, graphical picking is enabled (valid only in the GUI).

***NODE***

Node number on this element for which data are to be stored. If blank, store the average element value (except for *FMAG* values, which are summed instead of averaged). If *NODE* = P, graphical picking is enabled (valid only in the GUI).

***Item***

Label identifying the item. General item labels are shown in [Table 148:ESOL - General Item and Component Labels \(p. 554\)](#) below. Some items also require a component label.

***Comp***

Component of the item (if required). General component labels are shown in [Table 148:ESOL - General Item and Component Labels \(p. 554\)](#) below. If *Comp* is a sequence number (*n*), the *NODE* field will be ignored.

***Name***

Thirty-two character name for identifying the item on the printout and displays. Defaults to a label formed by concatenating the first four characters of the *Item* and *Comp* labels.

## Notes

See [Table 148:ESOL - General Item and Component Labels \(p. 554\)](#) for a list of valid item and component labels for element (except line element) results.

The **ESOL** command defines element results data to be stored from a results file (**FILE**). Not all items are valid for all elements. To see the available items for a given element, refer to the input and output summary tables in the documentation for that element.

Two methods of data access are available via the **ESOL** command. You can access some simply by using a generic label (*component name method*), while others require a label and number (*sequence number method*).

Use the component name method to access general element data (that is, element data generally available to most element types or groups of element types).

The sequence number method is required for data that is not averaged (such as pressures at nodes and temperatures at integration points), or data that is not easily described in a generic fashion (such as all derived data for structural line elements and contact elements, all derived data for thermal line elements, and layer data for layered elements).

Element results are in the element coordinate system, except for layered elements where results are in the layer coordinate system. Element forces and moments are in the layer coordinate system. Results are obtain-

able for an element at a specified node. Further location specifications can be made for some elements via the **SHELL**, **LAYERP26**, and **FORCE** commands.

**Table 148 ESOL - General Item and Component Labels**

Component Name Method		
Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1,2,3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
CDM	DMG	Damage variable
"	LM	Maximum previous strain energy for virgin material
GKS	X	Gasket component stress (also gasket pressure).
GKD	X	Gasket component total closure.

Component Name Method		
Item	Comp	Description
GKDI	X	Gasket component total inelastic closure.
GKTH	X	Gasket component thermal closure.
SS	X, XY, XZ	Interface traction (stress)
SD	X,XY,XZ	Interface Separation.
CONT	STAT1	Contact status.
"	PENE	Contact penetration
"	PRES	Contact pressure
"	SFRIC	Contact friction stress
"	STOT	Contact total stress (pressure plus friction)
"	SLIDE	Contact sliding distance
"	GAP	Contact gap distance
"	FLUX	Total heat flux at contact surface
"	CNOS	Total number of contact status changes during substep
"	FPRS	Fluid penetration pressure
TG	X, Y, Z, SUM	Component thermal gradient or vector sum
TF	X, Y, Z, SUM	Component thermal flux or vector sum
PG	X, Y, Z, SUM	Component pressure gradient or vector sum
EF	X, Y, Z, SUM	Component electric field or vector sum
D	X, Y, Z, SUM	Component electric flux density or vector sum
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum
B	X, Y, Z, SUM	Component magnetic flux density or vector sum
FMAG	X, Y, Z, SUM	Component electromagnetic forces or vector sum
F	X, Y, Z	Component structural force
M	X, Y, Z	Component structural moment
HEAT[2]		Heat flow
FLOW		Fluid flow
AMPS		Current flow
FLUX		Magnetic flux
CSG	X, Y, Z	Component magnetic current segment
SENE		"Stiffness" energy
STEN		Elemental energy dissipation due to <a href="#">stabilization</a>
KENE		Kinetic energy
JHEAT		Element Joule heat generation
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM)
JS	X, Y, Z	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement

**Component Name Method**

Item	Comp	Description
		current densities) in low frequency electric analyses. Components (X, Y, Z)
JT	X, Y, Z, SUM	Total measurable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM)
MRE		Magnetics Reynolds number
VOLU		Volume of volume element
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only)
FICT	TEMP	Fictive temperature

**Sequence Number Method**

Item	Comp	Description
SMISC	<i>snum</i>	Summable items
NMISC	<i>snum</i>	Nonsummable items
LS	<i>snum</i>	Line element elastic stresses
LEPEL	<i>snum</i>	Line element strains
LEPTH	<i>snum</i>	Line element thermal strains
LEPPL	<i>snum</i>	Line element plastic strains
LEPCR	<i>snum</i>	Line element creep strains
LBFE	<i>snum</i>	Line element temperatures

1. For more information on the meaning of contact status and its possible values, see [Reviewing Results in POST1](#) in the *Contact Technology Guide*.
2. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels HBOT, HE2, HE3, . . . , HTOP instead of HEAT.

**Menu Paths**

**Main Menu>TimeHist Postpro>Define Variables**

**Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables**

**ESORT**, *Item, Lab, ORDER, KABS, NUMB*

**Sorts the element table.**

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Item**

Label identifying the item:



**ETAB**

(currently the only *Item* available)

**Lab**

element table label:

**Lab**

Any user-defined label from the **ETABLE** command (input in the *Lab* field of the **ETABLE** command).

**ORDER**

Order of sort operation:

**0**

Sort into descending order.

**1**

Sort into ascending order.

**KABS**

Absolute value key:

**0**

Sort according to real value.

**1**

Sort according to absolute value.

**NUMB**

Number of elements (element table rows) to be sorted in ascending or descending order (*ORDER*) before sort is stopped (remainder will be in unsorted sequence) (defaults to all elements).

**Command Default**

Use ascending element number order.

**Notes**

The element table rows are sorted based on the column containing the *Lab* values. Use **EUSORT** to restore the original order. If **ESORT** is specified with PowerGraphics on **[/GRAPHICS,POWER]**, then the nodal solution results listing **[PRNSOL]** will be the same as with the full graphics mode **[/GRAPHICS,FULL]**.

**Menu Paths**

**Main Menu>General Postproc>List Results>Sorted Listing>Sort Elems**

**ESSOLV**, *Electit*, *Strutit*, *DIMN*, *MORPHOPT*, *Mcomp*, *Xcomp*, *ELECTOL*, *STRUTOL*, *MXLOOP*, *--*, *RUSEKY*, *RESTKY*, *EISCOMP*

**Performs a coupled electrostatic-structural analysis.**

**SOLUTION: Analysis Options**

MP <> <> <> <> <> <> <> <> <> <> <> <> EME <>

***Electit***

Title of the electrostatics physics file as assigned by the **PHYSICS** command.

**Strutit**

Title of the structural physics file as assigned by the **PHYSICS** command.

**DIMN**

Model dimensionality (a default is not allowed):

- 2**  
2-D model.
- 3**  
3-D model.

**MORPHOPT**

Morphing option:

- <0**  
Do not perform any mesh morphing or remeshing.
- 0**  
Remesh the non-structural regions for each recursive loop only if mesh morphing fails (default).
- 1**  
Remesh the non-structural regions each recursive loop and bypass mesh morphing.
- 2**  
Perform mesh morphing only, do not remesh any non-structural regions.

**Mcomp**

Component name of the region to be morphed. For 2-D models, the component may be elements or areas. For 3-D models, the component may be elements or volumes. A component must be specified. You must enclose name-strings in single quotes in the **ESSOLV** command line.

**Xcomp**

Component name of entities excluded from morphing. In the 2-D case, it is the component name for the lines excluded from morphing. In the 3-D case, it is component name for the areas excluded from morphing. Defaults to exterior non-shared entities (see the **DAMORPH**, **DVMORPH**, and **DEMORPH** commands). You must enclose name-strings in single quotes in the **ESSOLV** command line.

**ELECTOL**

Electrostatic energy convergence tolerance. Defaults to .005 (.5%) of the value computed from the previous iteration. If less than zero, the convergence criteria based on electrostatics results is turned off.

**STRUTOL**

Structural maximum displacement convergence tolerance. Defaults to .005 (.5%) of the value computed from the previous iteration. If less than zero, the convergence criteria base on structural results is turned off.

**MXLOOP**

Maximum number of allowable solution recursive loops. A single pass through both an electrostatics and structural analysis constitutes one loop. Defaults to 100.

--

Unused field.

**RUSEKY**

Reuse flag option:

- ≤ 1**  
Assumes initial run of **ESSOLV** using base geometry for the first electrostatics solution.

**>1**

Assumes **ESSOLV** run is a continuation of a previous **ESSOLV** run, whereby the morphed geometry is used for the initial electrostatic simulation.

**RESTKY**

Structural restart key.

**0**

Use static solution option for structural solution.

**1**

Use static restart solution option for structural solution.

**EISCOMP**

Element component name for elements containing initial stress data residing in file `jobname.ist`. The initial stress data must be defined prior to issuing **ESSOLV** (see **INISTATE** command).

## Notes

**ESSOLV** invokes an ANSYS macro which automatically performs a coupled electrostatic-structural analysis.

The macro displays periodic updates of the convergence.

If non-structural regions are remeshed during the analysis, boundary conditions and loads applied to nodes and elements will be lost. Accordingly, it is better to assign boundary conditions and loads to the solid model.

Use `RUSEKY > 1` for solving multiple **ESSOLV** simulations for different excitation levels (i.e., for running a voltage sweep). Do not issue the **SAVE** command to save the database between **ESSOLV** calls.

For nonlinear structural solutions, the structural restart option (`RESTKY = 1`) may improve solution time by starting from the previous converged structural solution.

For solid elements, **ESSOLV** automatically detects the air-structure interface and applies a Maxwell surface flag on the electrostatic elements. This flag is used to initiate the transfer of forces from the electrostatic region to the structure. When using the **ESSOLV** command with structural shell elements (for example, **SHELL181**), you must manually apply the Maxwell surface flag on all air elements surrounding the shells before writing the final electrostatic physics file. Use the **SFA** command to apply the Maxwell surface flag to the areas representing the shell elements; doing so ensures that the air elements next to both sides of the shells receive the Maxwell surface flag.

If lower-order structural solids or shells are used, set `KEYOPT(7) = 1` for the electrostatic element types to ensure the correct transfer of forces.

Information on creating the initial stress file is documented in the "Loading" chapter in the *Basic Analysis Guide*.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Physics>Coupled Solvers>Elec/struc**

**Main Menu>Solution>Physics>Coupled Solvers>Elec/struc**

**ESTIF**, *KMULT*

**Specifies the matrix multiplier for deactivated elements.**

[SOLUTION: Birth and Death](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***KMULT***

Stiffness matrix multiplier for deactivated elements (defaults to 1.0E-6).

**Command Default**

Use 1.0E-6 as the multiplier.

**Notes**

Specifies the stiffness matrix multiplier for elements deactivated with the **EKILL** command (birth and death).

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Birth & Death>StiffnessMult**

**Main Menu>Solution>Load Step Opts>Other>Birth & Death>StiffnessMult**

**ESURF**, *XNODE*, *Tlab*, *Shape*

**Generates elements overlaid on the free faces of existing selected elements.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***XNODE***

Node number that is used only in the following two cases:

*XNODE* is a single extra node number (ID) used for generating SURF151 or SURF152 elements when KEYOPT(5)=1.

*XNODE* is a single pressure node number (ID) used for generating [HSFLD241](#) or [HSFLD242](#) elements.

There is no default. *XNODE* must be specified for the above cases. If *XNODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A parameter or parametric expression can be substituted for *XNODE*.

***Tlab***

Generates target, contact, and hydrostatic fluid elements with correct direction of normals.

This option is valid only with [TARGE169](#), [TARGE170](#), [CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#), [CONTA176](#), [CONTA177](#), [HSFLD241](#), and [HSFLD242](#) elements.

**TOP**

Generates target and contact elements over beam and shell elements, or hydrostatic fluid elements over shell elements, with the normals the same as the underlying beam and shell elements (default).

**BOTTOM**

Generates target and contact elements over beam and shell elements, or hydrostatic fluid elements over shell elements, with the normals opposite to the underlying beam and shell elements.

If target or contact elements *and* hydrostatic fluid elements are defined on the same underlying shell elements, you only need to use this option once to orient the normals opposite to the underlying shell elements.

**REVERSE**

Reverses the direction of the normals on existing selected target elements, contact elements, and hydrostatic fluid elements.

If target or contact elements *and* hydrostatic fluid elements are defined on the same underlying shell elements, you only need to use this option once to reverse the normals for all selected elements.

**Shape**

Used to specify the element shape for target element **TARGE170** (*Shape* = LINE or POINT) or **TARGE169** elements (*Shape* = POINT).

**(blank)**

The target element takes the same shape as the external surface of the underlying element (default).

**LINE**

Generates LINE or PARA (parabolic) segments on exterior of selected 3-D elements.

**POINT**

Generates POINT segments on selected nodes.

**Notes**

The **ESURF** command generates elements of the currently active element type overlaid on the free faces of existing elements. For example, surface elements (such as **SURF151**, **SURF152**, **SURF153**, **SURF154**, or **SURF159**) can be generated over solid elements (such as **PLANE55**, **SOLID70**, **PLANE182**, **SOLID185**, or **SOLID272**, respectively).

Element faces are determined from the selected node set (**NSEL**) and the load faces for that element type. The operation is similar to that used for generating element loads from selected nodes via the **SF,ALL** command, except that elements (instead of loads) are generated. All nodes on the face must be selected for the face to be used. For shell elements, only face one of the element is available. If nodes are shared by adjacent selected element faces, the faces are not free and no element is generated.

Elements created by **ESURF** are oriented such that their surface load directions are consistent with those of the underlying elements. Carefully check generated elements and their orientations.

Generated elements use the existing nodes and the active **MAT**, **TYPE**, **REAL**, and **ESYS** attributes. The exception is when *Tlab* = REVERSE. The reversed target and contact elements have the same attributes as the original elements. If the underlying elements are solid elements, *Tlab* = TOP or BOTTOM has no effect.

When the command generates a target element, the shape is by default the same as that of the underlying element. Issue **ESURF,,,LINE** or **ESURF,,,POINT** to generate LINE, PARA, and POINT segments.

The **ESURF** command can also generate the 2-D or 3-D node-to-surface element **CONTA175**, based on the selected node components of the underlying solid elements. When used to generate **CONTA175** elements, all **ESURF** arguments are ignored. (If **CONTA175** is the active element type, the path **Main Menu > Preprocessor > Modeling > Create > Elements > Node-to-Surf** uses **ESURF** to generate elements.)

To generate [SURF151](#) or [SURF152](#) elements that have two extra nodes from [FLUID116](#) elements, `KEYOPT(5)` for [SURF151](#) or [SURF152](#) is first set to 0 and `ESURF` is issued. Then `KEYOPT(5)` for [SURF151](#) or [SURF152](#) is set to 2 and `MSTOLE` is issued. For more information, see [Using the Surface Effect Elements](#) in the *Thermal Analysis Guide*.

For hydrostatic fluid elements [HSFLD241](#) and [HSFLD242](#), the `ESURF` command generates triangular (2-D) or pyramid-shaped (3-D) elements with bases that are overlaid on the faces of selected 2-D or 3-D solid or shell elements. The single vertex for all generated elements is at the pressure node specified as `XNODE`. The generated elements fill the volume enclosed by the solid or shell elements. The nodes on the overlaid faces have translational degrees of freedom, while the pressure node shared by all generated elements has a single hydrostatic pressure degree of freedom, `HDSP` (see [HSFLD241](#) and [HSFLD242](#) for more information about the pressure node).

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Inf Acoustic**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Node to Surf**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Generl Surface>Extra Node**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Generl Surface>No extra Node**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf to Surf**

---

**ESYM**, --, *NINC*, *IEL1*, *IEL2*, *IEINC*

**Generates elements from a pattern by a symmetry reflection.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

--

Unused field.

***NINC***

Increment nodes in the given pattern by *NINC*.

***IEL1*, *IEL2*, *IEINC***

Reflect elements from pattern beginning with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and pattern is all selected elements [**ESEL**].

If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *IEINC* are ignored).

## Notes

Generates additional elements from a given pattern (similar to **EGEN**) except with a "symmetry" reflection. The operation generates a new element by incrementing the nodes on the original element, and reversing and shifting the node connectivity pattern. For example, for a 4-node 2-D element, the nodes in positions I, J, K, and L of the original element are placed in positions J, I, L, and K of the reflected element.

Similar permutations occur for all other element types. For line elements, the nodes in positions I and J of the original element are placed in positions J and I of the reflected element. *In releases prior to ANSYS 5.5,*

no node pattern reversing and shifting occurred for line elements generated by **ESYM**. To achieve the same results with ANSYS 5.5 as you did in prior releases, use the **EGEN** command instead.

It is recommended that symmetry elements be displayed and graphically reviewed.

If the nodes are also reflected (as with the **NSYM** command) this pattern is such that the orientation of the symmetry element remains similar to the original element (i.e., clockwise elements are generated from clockwise elements).

For a non-reflected node pattern, the reversed orientation has the effect of reversing the outward normal direction (clockwise elements are generated from counterclockwise elements).

---

### Note

Since nodes may be defined anywhere in the model independently of this command, any orientation of the "symmetry" elements is possible. See also the **ENSYM** command for modifying existing elements.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Reflect>Elements>Auto Numbered**

---

## ESYS, KCN

**Sets the element coordinate system attribute pointer.**

[PREP7: Meshing](#)

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### KCN

Coordinate system number:

**0**

Use element coordinate system orientation as defined (either by default or by KEYOPT setting) for the element (default).

**N**

Use element coordinate system orientation based on local coordinate system N (where N must be greater than 10). For global system 0, 1, or 2, define a local system N parallel to appropriate system with the **LOCAL** or **CS** command (for example: **LOCAL,11,1**).

## Command Default

Use element coordinate system orientation as defined (either by default or by KEYOPT setting) for the element (default).

## Notes

Identifies the local coordinate system to be used to define the element coordinate system of subsequently defined elements. Used only with area and volume elements. For non-layered volume elements, the local coordinate system N is simply assigned to be the element coordinate system. For shell and layered volume

elements, the x and y axes of the local coordinate system N are projected onto the shell or layer plane to determine the element coordinate system. See [Element Coordinate Systems](#) for more details. N refers to the coordinate system reference number (*KCN*) defined using the **LOCAL** (or similar) command. Element coordinate system numbers may be displayed [**/PNUM**].

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes**

**ET**, *ITYPE*, *Ename*, *KOP1*, *KOP2*, *KOP3*, *KOP4*, *KOP5*, *KOP6*, *INOPR*

**Defines a local element type from the element library.**

PREP7: [Element Type](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

### **ITYPE**

Arbitrary local element type number. Defaults to 1 + current maximum.

### **Ename**

Element name (or number) as given in the element library in Chapter 4 of the [Element Reference](#). The name consists of a category prefix and a unique number, such as [PIPE288](#). The category prefix of the name (PIPE for the example) may be omitted but is displayed upon output for clarity. If *Ename* = 0, the element is defined as a null element.

### **KOP1, KOP2, KOP3, . . . , KOP6**

KEYOPT values (1 through 6) for this element, as described in the [Element Reference](#).

### **INOPR**

If 1, suppress all element solution printout for this element type.

## Notes

The **ET** command selects an element type from the element library and establishes it as a local element type for the current model. Information derived from the element type is used for subsequent commands, so the **ET** command(s) should be issued early. (The [Element Reference](#) describes the available elements.)

A special option, *Ename* = 0, permits the specified element type to be ignored during solution without actually removing the element from the model. *Ename* may be set to zero only after the element type has been previously defined with a nonzero *Ename*. The preferred method of ignoring elements is to use the select commands (such as **ESEL**).

*KOP<sub>n</sub>* are element option keys. These keys (referred to as KEYOPT(*n*)) are used to turn on certain element options for this element. These options are listed under "KEYOPT" in the input table for each element type in the [Element Reference](#). KEYOPT values include stiffness formulation options, printout controls, and various other element options. If KEYOPT(7) or greater is needed, input their values with the **KEYOPT** command.

The **ET** command only defines an element type local to your model (from the types in the element library). The **TYPE** or similar [**KATT**, **LATT**, **AATT**, or **VATT**] command must be used to point to the desired local element type before meshing.



To activate the ANSYS program's LS-DYNA explicit dynamic analysis capability, use the **ET** command or its GUI equivalent to choose an element that works only with LS-DYNA (such as [SHELL163](#)). Choosing LS-DYNA in the Preferences dialog box does *not* activate LS-DYNA; it simply makes items and options related to LS-DYNA accessible in the GUI.

## Menu Paths

**Main Menu>Preprocessor>Element Type>Add/Edit/Delete**

**ETABLE**, *Lab*, *Item*, *Comp*

**Fills a table of element values for further processing.**

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lab*

Any unique user defined label for use in subsequent commands and output headings (maximum of eight characters and not a General predefined *Item* label). Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels. If the same as a previous user label, this result item will be included under the same label. Up to 200 different labels may be defined. The following labels are predefined and are not available for user-defined labels: REFL, STAT, and ERAS. *Lab* = REFL refills all tables previously defined with the **ETABLE** commands (not the **CALC** module commands) according to the latest **ETABLE** specifications and is convenient for refilling tables after the load step [**SET**] has been changed. Remaining fields will be ignored if *Lab* is REFL. *Lab* = STAT displays stored table values. *Lab* = ERAS erases the entire table.

### *Item*

Label identifying the item. General item labels are shown in the table below. Some items also require a component label. Character parameters may be used. *Item* = ERAS erases a *Lab* column.

### *Comp*

Component of the item (if required). General component labels are shown in the table below. Character parameters may be used.

## Notes

Defines a table of values per element (the element table) for use in further processing (see POST1 Element Table commands). The element table is organized as a "worksheet," with the rows representing all selected elements, and the columns consisting of result items which have been moved into the table (*Item,Comp*) with **ETABLE**. Each column of data is identified by a user-defined label (*Lab*) for listings and displays.

After entering the data into the element table, you are not limited to merely listing or displaying your data [**PLESOL**, **PRESOL**, etc.]. You may also perform many types of operations on your data, such as adding or multiplying columns [**SADD**, **SMULT**], defining allowable stresses for safety calculations [**SALLOW**], or multiplying one column by another [**SMULT**]. See the [Getting Started with ANSYS](#) in the *Basic Analysis Guide* for further details.

There are different types of results data that may be stored in the element table. For example, many items for an element are inherently single-valued (i.e., there is only one value per element). The single-valued items include: SERR, SDSG, TERR, TDSG, SENE, SEDN, TENE, KENE, JHEAT, JS, VOLU, and CENT. All other items are multivalued (i.e., they vary over the element, such that there is a different value at each node). Since

only one value is stored in the element table per element, an average value (based on the number of contributing nodes) is calculated for multivalued items. Exceptions to this averaging procedure are FMAG and all element force items, which represent the sum only of the contributing nodal values.

There are two methods of data access that may be used with the **ETABLE** command. The method you choose will depend upon the type of data that you wish to store. Some results can be accessed just with the use of a generic label (Component Name method), while others require a label and number (Sequence Number method).

The Component Name method is used to access the General element data (that is, element data which is generally available to most element types or groups of element types). All of the single-valued items and some of the more general multivalued items are accessible with the Component Name method. Various element results depend on the calculation method and the selected results location (**AVPRIN**, **RSYS**, **LAYER**, **SHELL**, and **ESEL**).

Although nodal data is readily available for listings and displays [**PRNSOL**, **PLNSOL**] without using the element table, you may also use the Component Name method to enter these results into the element table for further "worksheet" manipulation (see the [Getting Started with ANSYS](#) in the *Basic Analysis Guide* for more details). A listing of the General *Item* and *Comp* labels for the Component Name method is shown below.

The Sequence Number method allows you to view results for data that is not averaged (such as pressures at nodes, temperatures at integration points, etc.), or data that is not easily described in a generic fashion (such as all derived data for structural line elements and contact elements, all derived data for thermal line elements, layer data for layered elements, etc.). A table illustrating the *Items* (such as LS, LEPEL, LEPTH, SMISC, NMISC, SURF, etc.) and corresponding sequence numbers for each element is shown in the Output Data section of each element description found in the [Element Reference](#).

Some element table data are reported in the results coordinate system. These include all component results (for example, UX, UY, etc.; SX, SY, etc.). The solution writes component results in the database and on the results file in the solution coordinate system. When you issue the **ETABLE** command, these results are then transformed into the results coordinate system [**RSYS**] before being stored in the element table. The default results coordinate system is global Cartesian [**RSYS,0**]. All other data are retrieved from the database and stored in the element table with no coordinate transformation.

Use the **PRETAB**, **PLETAB**, or **ETABLE,STAT** commands to display the stored table values. Issue **ETABLE,ERAS** to erase the entire table. Issue **ETABLE,Lab,ERAS** to erase a *Lab* column.

When the GUI is on, if a **Delete** operation in a Define Element Table Data dialog box writes this command to a log file (*Jobname.LOG* or *Jobname.LGW*), you will observe that *Lab* is blank, *Item* = ERASE, and *Comp* is an integer number. In this case, the GUI has assigned a value of *Comp* that corresponds to the location of a chosen variable name in the dialog box's list. It is *not* intended that you type in such a location value for *Comp* in an ANSYS session. However, a file that contains a GUI-generated **ETABLE** command of this form can be used for batch input or for use with the **/INPUT** command.

**Table 149 ETABLE - General Item and Component Labels**

General Item and Component Labels <b>ETABLE, Lab, Item, Comp</b>		
Item	Comp	Description
Valid Item Labels for Degree of Freedom Results		
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.

**General Item and Component Labels ETABLE, Lab, Item, Comp**

Item	Comp	Description
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity.
A	X, Y, Z	X, Y, or Z magnetic vector potential.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.
SP0 <sub>n</sub>		Mass fraction of species <i>n</i> , where <i>n</i> = 1 to 6. If a species is given a user-defined name [MSSPEC], use that name instead of SP0 <sub>n</sub> .

Valid Item Labels for FLOTRAN Nodal Results

TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.
LMD <sub>n</sub>		Laminar mass diffusion coefficient for species <i>n</i> , where <i>n</i> = 1 to 6.
EMD <sub>n</sub>		Effective mass diffusion coefficient for species <i>n</i> , where <i>n</i> = 1 to 6.

Valid Item and Component Labels for Element Results

S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.

General Item and Component Labels *ETABLE, Lab, Item, Comp*

Item	Comp	Description
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPSW		Swelling strain.
EPTO	X, Y, Z, XY, YZ, XZ	Component total mechanical strain ( <i>excluding</i> thermal) (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total mechanical strain.
"	INT	Total mechanical strain intensity.
"	EQV	Total equivalent mechanical strain.
EPTT	X, Y, Z, XY, YZ, XZ	Component total strain <i>including</i> thermal (EPEL + EPTH + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT	Total strain intensity.
"	EQV	Total equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
CDM	DMG	Damage variable
"	LM	Maximum previous strain energy for virgin material
FAIL	MAX	Maximum of all active failure criteria defined at the current location (See the <b>FCTYP</b> command for details.) [1][3]
"	EMAX	Maximum Strain Failure Criterion. [1][3]
"	SMAX	Maximum Stress Failure Criterion. [1][3]

**General Item and Component Labels ETABLE, Lab, Item, Comp**

<b>Item</b>	<b>Comp</b>	<b>Description</b>
"	TWSI	Tsai-Wu Strength Index Failure Criterion. [1][3]
"	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion. [1][3]
"	HFIB	Hashin Fiber Failure Criterion. [1][3][5]
"	HMAT	Hashin Matrix Failure Criterion. [1][3][5]
"	PFIB	Puck Fiber Failure Criterion. [1][3][5]
"	PMAT	Puck Matrix Failure Criterion. [1][3][5]
"	USR1, USR2, ..., USR9	User-defined failure criteria. [1][3][4][5]
FCMX	LAY	Layer number where the maximum of all active failure criteria over the entire element occurs. [1][3]
"	FC	Number of the maximum-failure criterion over the entire element [1 (p. 571)][3 (p. 571)]:  1 - EMAX 2 - SMAX 3 - TWSI 4 - TWSR 5 - HFIB 6 - HMAT 7 - PFIB 8 - PMAT 9~17 - USR1~USR9
"	VAL	Value of the maximum failure criterion over the entire element. [1][3]
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG[4]	X, Y, Z, SUM	Component electromagnetic forces or vector sum.
SERR[5]		Structural error energy.
SDSG5		Absolute value of maximum variation of any nodal stress component.
TERR5		Thermal error energy.
TDSG5		Absolute value of the maximum variation of any nodal thermal gradient component.
F	X, Y, Z	Component structural force. Sum of element nodal values.
M	X, Y, Z	Component structural moment. Sum of element nodal values.
HEAT		Heat flow. Sum of element nodal values.
FLOW		Fluid flow. Sum of element nodal values.

**General Item and Component Labels ETABLE, Lab, Item, Comp**

<b>Item</b>	<b>Comp</b>	<b>Description</b>
AMPS		Current flow. Sum of element nodal values.
FLUX		Magnetic flux. Sum of element nodal values.
CSG	X, Y, Z	Component magnetic current segment.
SENE		"Stiffness" energy or thermal heat dissipation (applies to all elements where meaningful). Same as TENE.
SEDN		Strain energy density.
AENE		Artificial energy of the element. This includes the sum of hourglass control energy and energy generated by in-plane drilling stiffness from shell elements (applies to all elements where meaningful). The energy is used for comparisons to SENE energy to predict the solution error due to artificial stiffness. See the <a href="#">Theory Reference for the Mechanical APDL and Mechanical Applications</a> .
TENE		Thermal heat dissipation or "stiffness" energy (applies to all elements where meaningful). Same as SENE.
KENE		Kinetic energy (applies to all elements where meaningful).
STEN		Elemental energy dissipation due to <a href="#">stabilization</a> .
JHEAT		Element Joule heat generation.
JS	X, Y, Z, SUM	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM).
JT	X, Y, Z, SUM	Total measureable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM).
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).
MRE		Magnetics Reynolds number
VOLU		Element volume. Based on unit thickness for 2-D plane elements (unless the thickness option is used) and on the full 360 degrees for 2-D axisymmetric elements.
CENT	X, Y, Z	Undeformed X, Y, or Z location (based on shape function) of the element centroid in the active coordinate system.
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
SMISC	<i>snum</i>	Element summable miscellaneous data value at sequence number <i>snum</i> (shown in the Output Data section of each applicable element description in the <a href="#">Element Reference</a> ).
NMISC	<i>snum</i>	Element non-summable miscellaneous data value at sequence number <i>snum</i> (shown in the Output Data section of each applicable element description found in the <a href="#">Element Reference</a> ).
SURF	<i>snum</i>	Element surface data value at sequence number <i>snum</i> (shown in <a href="#">Surface Solution</a> of the <a href="#">Element Reference</a> ).

**General Item and Component Labels ETABLE, Lab, Item, Comp**

Item	Comp	Description
CONT	STAT	Contact status[3]:  3-closed and sticking 2-closed and sliding 1-open but near contact 0-open and not near contact
"	PENE	Contact penetration (zero or positive).
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
"	GAP	Contact gap distance (0 or negative).
"	FLUX	Total heat flux at contact surface.
"	CNOS	Total number of contact status changes during substep.
"	FPRS	Fluid penetration pressure
TOPO		Densities used for topological optimization.
CAP	C0,X0,K0,ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; l1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2: shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.
EDPC	CSIG,CSTR	Material EDP creep model only (not including the cap model): Equivalent creep stress; equivalent creep strain.
ESIG	X,Y,Z,XY,YZ,ZX	Components of Biot's effective stress.
"	1, 2, 3	Principal stresses of Biot's effective stress.
"	INT	Stress intensity of Biot's effective stress.
"	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
"	GPOR	Porosity due to void growth.
"	NPOR	Porosity due to void nucleation.
FFLX	X,Y,Z	Fluid flow flux in poromechanics.
FICT	TEMP	Fictive temperature.

1. For [SHELL131](#) and [SHELL132](#) elements with KEYOPT(3) = 0 or 1, use labels TBOT, TE2, TE3, . . . , TTOP instead of TEMP.
2. For the CONT items for elements [CONTA171](#) through [CONTA177](#), the reported data is averaged across the element.
3. For MPC-based contact definitions, the value of STAT can be negative. This indicates that one or more contact constraints were intentionally removed to prevent overconstraint. STAT = -3 is used for MPC bonded contact; STAT = -2 is used for MPC no-separation contact.

4. When using the **EMFT** procedure to calculate electromagnetic force ([SOLID117](#), [PLANE121](#), [SOLID122](#), [SOLID123](#), [PLANE233](#), [SOLID236](#) or [SOLID237](#) elements only), the FMAG sum will be zero or near zero.
5. Some element- and material-type limitations apply. For more information, see the documentation for the **PRERR** command.

## Menu Paths

**Main Menu>General Postproc>Element Table>Define Table**  
**Main Menu>General Postproc>Element Table>Erase Table**

## ETCHG, *Cnv*

Changes element types to their corresponding types.

[PREP7: Element Type](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

### *Cnv*

Converts the element types to the corresponding type. Valid labels are:

#### **ETI**

Explicit to Implicit

#### **ITE**

Implicit to Explicit

#### **TTE**

Thermal to Explicit

#### **TTS**

Thermal to Structural

#### **STT**

Structural to Thermal

#### **MTT**

Magnetic to Thermal

#### **FTS**

Fluid to Structural

#### **ETS**

Electrical to Structural

## Notes

Changes the currently defined element types to their corresponding types. Elements without a companion element (listed above) are not switched and should be switched with the **ET** command to an appropriate element type or to a null element. The KEYOPT values for the switched element types are reset to zero or to their default values. You must check these values to see if they are still meaningful. Additionally, if *Cnv* = ETI, ITE, or TTE, all real constants are set to zero.

If *Cnv* = ITE, you will need to choose a material model that corresponds to your previously-defined material properties. If working interactively, you will be prompted to do so.



## Element Pairs

### ETI -- Explicit to Implicit

163 &gt; 181

165 &gt; 14

164 &gt; 185

166 &gt; 21

### ITE -- Implicit to Explicit

14 &gt; 165

21 &gt; 166

185 &gt; 164

181 &gt; 163

### TTE -- Thermal to Explicit

55 &gt; 162

70 &gt; 164

### TTS -- Thermal to Structural

33 &gt; 180

67 &gt; 42

75 &gt; 25

131 &gt; 181

55 &gt; 182

69 &gt; 45

77 &gt; 183

151 &gt; 153

70 &gt; 185

78 &gt; 83

152 &gt; 154

71 &gt; 21

87 &gt; 187

278 &gt; 185

90 &gt; 186

279 &gt; 186

### STT -- Structural to Thermal

21 &gt; 71

62 &gt; 70

88 &gt; 77

154 &gt; 152

25 &gt; 75

64 &gt; 70

89 &gt; 90

158 &gt; 87

41 &gt; 131

65 &gt; 70

91 &gt; 132

180 &gt; 33

42 &gt; 55

73 &gt; 70

92 &gt; 87

181 &gt; 131

45 &gt; 70

74 &gt; 77

93 &gt; 132

182 &gt; 55

56 &gt; 55

82 &gt; 77

95 &gt; 90

183 &gt; 77

58 &gt; 70

83 &gt; 78

106 &gt; 55

185 &gt; 70

84 &gt; 55/77

107 &gt; 70

186 &gt; 90

86 &gt; 70

108 &gt; 77

187 &gt; 87

153 &gt; 151

### MTT -- Magnetic to Thermal

53 &gt; 77

96 &gt; 70

117 &gt; 90

120 &gt; 90

62 &gt; 70

97 &gt; 70

119 &gt; 87

### FTS -- Fluid to Structural

141 &gt; 42

142 &gt; 45

### ETS -- Electrical to Structural

67 &gt; 42

121 &gt; 42/82

123 &gt; 92

128 &gt; 45/95

69 &gt; 45

122 &gt; 45/95

127 &gt; 92

## Menu Paths

**Main Menu>Preprocessor>Element Type>Switch Elem Type**

---

**ETCONTROL**, *Eltech*, *Eldegene***Control the element technologies used in element formulation (for applicable elements).**

PREP7: Element Type

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***Eltech***

Element technology control:

**SUGGESTION**

ANSYS makes a suggestion for the best element technology before solving. If necessary, mixed u-P (KEYOPT(6)) will also be included and reset. This behavior is the default.

**SET**

ANSYS informs you of the best settings and automatically resets any applicable KEYOPT settings; this will override any manual settings you provided previously.

**OFF**

Turns automatic selection of element technology off. No suggestions will be issued, and no automatic resetting is done

***Eldegene***

Element degenerated shape control:

**ON**

If element shapes are degenerated, the degenerated shape function is employed and enhanced strain, simplified enhanced strain, and B-bar formulations are turned off (default).

**OFF**

If element shapes are degenerated, regular shape functions are still used, and the specified element technologies (e.g., enhanced strain, B-bar, uniform reduced integration) are still used.

**Notes**

The command default is **ETCONTROL,SUGGESTION,ON**.

This command is valid for elements SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, SHELL209, REINF264, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290.

For more information, see [Automatic Selection of Element Technologies](#) in the *Element Reference*.

**Menu Paths****Main Menu>Preprocessor>Element Type>Elem Tech Control**

---

**ETDELE**, *ITYP1*, *ITYP2*, *INC*

Deletes element types.

[PREP7: Element Type](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*ITYP1*, *ITYP2*, *INC*

Deletes element types from *ITYP1* to *ITYP2* (defaults to *ITYP1*) in steps of *INC* (defaults to 1). If *ITYP1* = ALL, *ITYP2* and *INC* are ignored and all element types are deleted. Element types are defined with the **ET** command.

**Menu Paths**

**Main Menu>Preprocessor>Element Type>Add/Edit/Delete**

**ETLIST**, *ITYP1*, *ITYP2*, *INC*

Lists currently defined element types.

[PREP7: Element Type](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*ITYP1*, *ITYP2*, *INC*

Lists element types from *ITYP1* to *ITYP2* (defaults to *ITYP1*) in steps of *INC* (defaults to 1). If *ITYP1* = ALL (default), *ITYP2* and *INC* are ignored and all element types are listed.

**Notes**

This command is valid in any processor.

**Menu Paths**

**Utility Menu>List>Properties>Element Types**

**ETYPE**

Specifies "Element types" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Preprocessor>Element Types

---

### EUSORT

Restores original order of the element table.

[POST1: Element Table](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

Changing the selected element set [[ESEL](#)] also restores the original element order.

## Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Unsort Elems

---

**EWRITE**, *Fname*, *Ext*, *--*, *KAPPND*, *Format*

Writes elements to a file.

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

#### *Ext*

Filename extension (8 character maximum).

The extension defaults to `ELEM` if *Fname* is blank.

--

Unused field.

#### *KAPPND*

Append key:

**0**

Rewind file before the write operation.

**1**

Append data to the end of the existing file.

#### *Format*

Format key:

**SHORT**

l6 format (the default).

**LONG**

l8 format.

**Notes**

Writes the selected elements to a file. The write operation is not necessary in a standard ANSYS run but is provided as convenience to users wanting a coded element file. If issuing **EWRITE** from ANSYS to be used in ANSYS, you must also issue **NWRITE** to store nodal information for later use. Only elements having all of their nodes defined (and selected) are written. Data are written in a coded format. The data description of each record is: I, J, K, L, M, N, O, P, MAT, TYPE, REAL, SECNUM, ESYS, IEL, where MAT, TYPE, REAL, and ESYS are attribute numbers, SECNUM is the beam section number, and IEL is the element number.

The format is (14l6) if *Format* is set to SHORT and (14l8) if the *Format* is set to LONG, with one element description per record for elements having eight nodes or less. For elements having more than eight nodes, nodes nine and above are written on a second record with the same format.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Elements>Write Elem File**

**\*EXIT**

**Exits a do-loop.**

APDL: Process Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

The command following the **\*ENDDO** is executed next. The exit option may also be conditional [Use the **\*IF**]. The **\*EXIT** command must appear on the same file as the **\*DO** command.

This command is valid in any processor.

**Menu Paths**

**This command cannot be accessed from a menu.**

**/EXIT**, *Slab*, *Fname*, *Ext*, --

**Stops the run and returns control to the system.**

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Slab***

Mode for saving the database:

**MODEL**

Save the model data (solid model, finite element model, loadings, etc.) only (default).

**SOLU**

Save the model data and the solution data (nodal and element results).

**ALL**

Save the model data, solution data and post data (element tables, path results, etc.)

**NOSAVE**

Do not save any data on `File.DB` (an existing DB file will not be overwritten).

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name, defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `DB` if *Fname* is blank.

--

Unused field.

**Notes**

The current database information may be written on `File.DB` or a named file. If `File.DB` already exists, a backup file (`File.DBB`) will also be written whenever a new `File.DB` is written.

This command is valid in any processor. Issuing this command at any point will exit the program.

**Menu Paths**

**Utility Menu>File>Exit**

**EXP, IR, IA, --, --, Name, --, --, FACTA, FACTB**

**Forms the exponential of a variable.**

[POST26: Operations](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to NV [\[NUMVAR\]](#)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

***IA***

Reference number of the variable to be operated on.

--, --

Unused fields.

**Name**

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --

Unused fields.

**FACTA**

Scaling factor applied to variable *IA* (defaults to 1.0).

**FACTB**

Scaling factor (positive or negative) applied to the operation (defaults to 1.0).

**Notes**

Forms the exponential of a variable according to the operation:

$$IR = FACTB * EXP(FACTA \times IA)$$

**Menu Paths**

**Main Menu>TimeHist Postpro>Math Operations>Exponentiate**

**EXPAND**, *Nrepeat*, MODAL, *HIndex*, *Icsys*, *SctAng*, --, *Phase*

**Displays the results of a modal cyclic symmetry analysis.**

POST1: Special Purpose

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***Nrepeat***

Number of sector repetitions for expansion. The default is 0 (no expansion).

**MODAL**

Specifies that the expansion is for a modal cyclic symmetry analysis.

***HIndex***

The harmonic index ID for the results to expand.

***Icsys***

The coordinate system number used in the modal cyclic symmetry solution. The default is the global cylindrical coordinate system (specified via the **CSYS** command where *KCN* = 1).

***SctAng***

The sector angle in degrees, equal to 360 divided by the number of cyclic sectors.

--

This field is reserved for future use.

***Phase***

The phase angle in degrees to use for the expansion. The default is 0. Typically, the value is the peak displacement (or stress/strain) phase angle obtained via the **CYCPHASE** command.

**Notes**

Issue this command to display the results of a modal cyclic symmetry analysis.

When you issue the **EXPAND**,*Nrepeat* command, subsequent **SET** commands read data from the results file and expand them to *Nrepeat* sectors. As long as no entities have been modified, this expansion can be negated (that is, reverted to single sector) by issuing **EXPAND** with no arguments. If you modify entities and wish to return to the partial model, use the Session Editor (see [Restoring Database Contents](#) in the *Operations Guide*).

**EXPAND** displays the results and allows you to print them, as if for a full model. The harmonic index (automatically retrieved from the results file) appears in the legend column.

When plotting or printing element strain energy (SENE), the **EXPAND** command works with brick or tet models only. Element kinetic energy (KENE) plotting or printing is not supported.

**EXPAND** is a specification command valid only in POST1. It is significantly different from the **/EXPAND** command in several respects, (although you can use either command to display the results of a modal cyclic symmetry analysis):

- **EXPAND** has none of the limitations of the **/EXPAND** command.
- **EXPAND** changes the database by modifying the geometry, the nodal displacements, and element stresses as they are read from the results file, whereas the **/EXPAND** command does not change the database.
- You can print results displayed via **EXPAND**.

---

### Caution

The **EXPAND** command creates new nodes and elements; therefore, saving (or issuing the **/EXIT**, **ALL** command) after issuing the **EXPAND** command can result in large databases.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Cyc Expansion

---

**/EXPAND**, *Nrepeat1*, *Type1*, *Method1*, *DX1*, *DY1*, *DZ1*, *Nrepeat2*, *Type2*, *Method2*, *DX2*, *DY2*, *DZ2*, *Nrepeat3*, *Type3*, *Method3*, *DX3*, *DY3*, *DZ3*

**Allows the creation of a larger graphic display than represented by the actual finite element analysis model.**

POST1: Special Purpose

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

*Nrepeat1*, *Nrepeat2*, *Nrepeat3*

The number of repetitions required for the element pattern. The default is 0 (no expansion).

*Type1*, *Type2*, *Type3*

The type of expansion requested.

**RECT**

Causes a Cartesian transformation of DX, DY, and DZ for each pattern (default).



**POLAR**

Causes a polar transformation of DR, D-Theta and DZ for each pattern.

**AXIS**

Causes 2-D axisymmetric expansion (that is, rotates a 2-D model created in the X-Y plane about the Y axis to create a 3-D model).

**LRECT**

Causes a Cartesian transformation of DX, DY, and DZ for each pattern about the current local coordinate system (specified via the **CSYS** command).

**LPOLAR**

Causes a polar transformation of DR, D-Theta, and DZ for each pattern about the local coordinate system (specified via the **CSYS** command).

**Method1, Method2, Method3**

The method by which the pattern is repeated.

**FULL**

Causes a normal repeat of the pattern (default).

**HALF**

Uses a symmetry transformation for alternate repeats (to produce an image of a complete circular gear from the image of half a tooth, for example).

**DX1, DY1, DZ1, DX2, DY2, DZ2, DX3, DY3, DZ3**

The Cartesian or polar increments between the repeated patterns. Also determines the reflection plane. Reflection is about the plane defined by the normal vector (DX, DY, DZ). If you want no translation, specify a small nonzero value. For a half-image expansion, the increment DX, DY, or DZ is doubled so that POLAR,HALF,45 produces full images on 90° centers, and RECT,HALF,1 produces full images on 2-meter centers.

**Notes**

You can use the **/EXPAND** command to perform up to three symmetry expansions at once (that is, X, Y, and Z which is equal to going from a 1/8 model to a full model). Polar expansions allow you to expand a wheel section into a half wheel, then into the half section, and then into the whole.

The command displays elements/results when you issue the **EPLLOT** command or postprocessing commands.

The command works on all element and result displays, except as noted below. As the graphic display is created, the elements (and results) are repeated as many times as necessary, expanding the geometry and, if necessary, the displacements and stresses.

Derived results are not supported.

The **/EXPAND** command has the following limitations:

- It does not support solid model entities.
- POLAR, FULL or HALF operations are meaningful only in global cylindrical systems and are unaffected by the **RSYS** or **DSYS** commands. Cartesian symmetry or unsymmetric operations also occur about the global Cartesian system.
- It does not average nodal results across sector boundaries, even for averaged plots (such as those obtained via the **PLNSOL** command).
- Axisymmetric harmonic element results are not supported for *Type* = AXIS.

The **/EXPAND** command differs significantly from the **EXPAND** command in several respects:

- The uses of **/EXPAND** are of a more general nature, whereas the **EXPAND** command is intended *primarily* to expand modal cyclic symmetry results.
- **/EXPAND** does not change the database as does the **EXPAND** command.
- You cannot print results displayed via **/EXPAND**.

## Menu Paths

Utility Menu>PlotCtrls>Style>Symmetry Expansion>

Utility Menu>PlotCtrls>Style>Symmetry Expansion>Expansion by values

Utility Menu>PlotCtrls>Style>Symmetry Expansion>Modal Cyclic Symmetry

Utility Menu>PlotCtrls>Style>Symmetry Expansion>Periodic/Cyclic Symmetry Expansion

---

## EXPASS, Key

Specifies an expansion pass of an analysis.

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

### Key

Expansion pass key:

#### OFF

No expansion pass will be performed (default).

#### ON

An expansion pass will be performed.

## Notes

Specifies that an expansion pass of a modal, substructure, buckling, transient, or harmonic analysis is to be performed.

---

### Note

This separate solution pass requires an explicit **FINISH** to preceding analysis and reentry into SOLUTION.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>ExpansionPass

Main Menu>Solution>Analysis Type>ExpansionPass

---

**\*EXPORT**, *Matrix, Format, Fname, Val1, Val2, Val3*

**Exports a matrix to a file in the specified format.**

APDL: Matrix Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Matrix*

Name of the matrix to export (must be a matrix previously created with **\*DMAT** or **\*SMAT**, or a vector previously created with **\*VEC**).

### *Format*

Format of the output file:

#### **MMF --**

Export the matrix in the Matrix Market Format.

#### **SUB --**

Export the matrix in the SUB file format.

#### **HBMAT --**

Export the matrix in the Harwell-Boeing file format.

#### **MAT --**

Export the matrix in a native format, to be re-imported using the **\*DMAT** or **\*SMAT** command.

#### **EMAT --**

Export the matrix to an existing EMAT file.

#### **APDL --**

Export the matrix to an APDL array parameter.

#### **PS --**

Export the matrix profile to a Postscript file.

### *Fname*

Name of the file, or name of the array parameter if *Format* = APDL.

### *Val1, Val2, Val3*

Additional input. The meaning of *Val1* through *Val3* will vary depending on the specified *Format*. See table below for details.

Additional input for <i>Format</i> = SUB, HBMAT, EMAT, APDL, and PS				
<i>Format</i>	<i>Val1</i>	<i>Val2</i>	<i>Val3</i>	Description
SUB	Matrix type: STIFF - Stiffness (or conductivity) matrix MASS - Mass (or specific heat) matrix DAMP - Damping matrix RHS - Load vector	Matrix containing the row information (see <a href="#">Notes</a> ). Not needed if the matrix was imported from an existing SUB or DMIG file.	WAIT DONE	Identify matrix type to write and specify when to perform the export (see <a href="#">Notes</a> ).

HB-MAT	File format: ASCII BINARY	(not used)	(not used)	Identify the output format.
EMAT	Matrix type: STIFF - Stiffness (or conductivity) matrix MASS - Mass (or specific heat) matrix DAMP - Damping matrix STRESS - Stress-stiffness matrix	Element number	(not used)	Identify the matrix type and the element number.
AP-DL	First column	Last Column	(not used)	Block of columns to export.
PS	Color key: BW - black and white (default) COLOR - color	(not used)	(not used)	Identify the output format.

## Notes

Only sparse matrices can be exported to Postscript files. This option plots the matrix profile as a series of dots.

If you want to create a .SUB file from several matrices, you need to set *Val3* = WAIT for all matrices but the last, and *Val3* = DONE for the last one. The export will be effective at the last \*EXPORT command.

To create a .SUB file from scratch, you must supply the row information array in the *Val2* field. It is an  $m \times 2$  array, where  $m$  is the size of the matrix. The first column is the node number and the second column is the DOF number corresponding to each row of the matrix.

## Menu Paths

**This command cannot be accessed from a menu.**

**EXPROFILE**, *Ldtype*, *Load*, *VALUE*, *Pname*, *Fname*, *Fext*, *Fdir*

Exports ANSYS interface loads to a CFX Profile file.

POST1: Special Purpose

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *Ldtype*

Load type:

#### **SURF**

Surface load

**VOLU**

Volumetric load

**Load**

Surface loads:

**DISP**

Displacement

**TEMP**

Temperature

**HFLU**

Heat flux

Volumetric loads:

**DISP**

Displacement

**FORC**

Force

**HGEN**

Heat generation

**VALUE**

Surface or volume interface number.

**Pname**

Field name in CFX Profile file (32 characters maximum). Defaults to `bcploadnumber` for a surface load and `subdloadnumber` for volumetric load.

**Fname**

The CFX Profile filename (248 character maximum). Defaults to `jobname_bcploadnumber` for a surface load and `jobname_subdloadnumber` for a volumetric load.

**Fext**

The Profile file extension (8 character maximum). Defaults to `csv`.

**Fdir**

The Profile file directory (8 characters maximum). Defaults to current directory.

**Notes**

To transfer multiple loads across an interface, specify a unique file name and extension for each load.

Force (FORC) and heat generation (HGEN) are per unit volume.

The **EXPROFILE** command must be accompanied by the **EXUNIT** command in order to write unit information into a CFX Profile export file.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**This command cannot be accessed from a menu.**

**EXPSOL**, *LSTEP*, *SBSTEP*, *TIMFRQ*, *Elcalc*

Specifies the solution to be expanded for reduced analyses.

[SOLUTION: Load Step Options](#)

MP ME ST PR PRN <> <> <> <> <> <> <> <> EME MFS

***LSTEP*, *SBSTEP***

Expand the solution identified as load step *LSTEP* and substep *SBSTEP*.

***TIMFRQ***

As an alternative to *LSTEP* and *SBSTEP*, expand the solution at, or nearest to, the time value *TIMFRQ* (for **ANTYPE**,TRANS or **ANTYPE**,SUBSTR) or frequency value *TIMFRQ* (for **ANTYPE**,HARMIC). *LSTEP* and *SBSTEP* should be blank.

***Elcalc***

Element calculation key:

**YES**

Calculate element results, nodal loads, and reaction loads.

**NO**

Do not calculate these items.

**Notes**

Specifies the solution to be expanded from analyses that use reduced or mode superposition methods (**ANTYPE**,HARMIC, TRANS, or SUBSTR). Use the **NUMEXP** command to expand a group of solutions.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Load Step**

**Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq**

**Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Load Step**

**Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq**

**EXTOPT**, *Lab*, *Val1*, *Val2*, *Val3*, *Val4*

Controls options relating to the generation of volume elements from area elements.

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Lab***

Label identifying the control option. The meanings of *Val1*, *Val2*, and *Val3* will vary depending on *Lab*.

**ON**

Sets carryover of the material attributes, real constant attributes, and element coordinate system attributes of the pattern area elements to the generated volume elements. Sets the pattern area mesh to clear when volume generations are done. *Val1*, *Val2*, and *Val3* are ignored.

**OFF**

Removes all settings associated with this command. *Val1*, *Val2*, and *Val3* are ignored.

**STAT**

Shows all settings associated with this command. *Val1*, *Val2*, *Val3*, and *Val4* are ignored.

**ATTR**

Sets carryover of particular pattern area attributes (materials, real constants, and element coordinate systems) of the pattern area elements to the generated volume elements. (See 2 (p. 588).) *Val1* can be:

**0**

Sets volume elements to use current **MAT** command settings.

**1**

Sets volume elements to use material attributes of the pattern area elements.

*Val2* can be:

**0**

Sets volume elements to use current **REAL** command settings.

**1**

Sets volume elements to use real constant attributes of the pattern area elements.

*Val3* can be:

**0**

Sets volume elements to use current **ESYS** command settings.

**1**

Sets volume elements to use element coordinate system attributes of the pattern area elements.

*Val4* can be:

**0**

Sets volume elements to use current **SECNUM** command settings.

**1**

Sets volume elements to use section attributes of the pattern area elements.

**ESIZE**

*Val1* sets the number of element divisions in the direction of volume generation or volume sweep. For **VDRAG** and **VSWEEP**, *Val1* is overridden by the **LESIZE** command *NDIV* setting. *Val2* sets the spacing ratio (bias) in the direction of volume generation or volume sweep. If positive, *Val2* is the nominal ratio of last division size to first division size (if > 1.0, sizes increase, if < 1.0, sizes decrease). If negative, *Val2* is the nominal ratio of center division(s) size to end divisions size. Ratio defaults to 1.0 (uniform spacing). *Val3* and *Val4* are ignored.

**ACLEAR**

Sets clearing of pattern area mesh. (See 3 (p. 588).) *Val1* can be:

**0**

Sets pattern area to remain meshed when volume generation is done.

**1**

Sets pattern area mesh to clear when volume generation is done. *Val2*, *Val3*, and *Val4* are ignored.

**VSWE**

Indicates that volume sweeping options will be set using *Val1* and *Val2*. Settings specified with **EXTOPT**, **VSWE** will be used the next time the **VSWEEP** command is invoked. If *Lab* = **VSWE**, *Val1* becomes a label. *Val1* can be:

**AUTO**

Indicates whether you will be prompted for the source and target used by **VSWEEP** or if VSWE should automatically determine the source and target. If  $Val1 = \text{AUTO}$ ,  $Val2$  is ON by default. VSWE will automatically determine the source and target for **VSWEEP**. You will be allowed to pick more than one volume for sweeping. When  $Val2 = \text{OFF}$ , the user will be prompted for the source and target for **VSWEEP**. You will only be allowed to pick one volume for sweeping.

**TETS**

Indicates whether **VSWEEP** will tet mesh non-sweepable volumes or leave them unmeshed. If  $Val1 = \text{TETS}$ ,  $Val2$  is OFF by default. Non-sweepable volumes will be left unmeshed. When  $Val2 = \text{ON}$ , the non-sweepable volumes will be tet meshed if the assigned element type supports tet shaped elements.

$Val3$  is ignored for  $Lab = \text{VSWE}$ .

 **$Val1, Val2, Val3, Val4$** 

Additional input values as described under each option for  $Lab$ .

**Notes**

1. **EXTOPT** controls options relating to the generation of volume elements from pattern area elements using the **VEXT**, **VROTAT**, **VOFFST**, **VDRAG**, and **VSWEEP** commands. (When using **VSWEEP**, the pattern area is referred to as the source area.)
2. Enables carryover of the attributes of the pattern area elements to the generated volume elements when you are using **VEXT**, **VROTAT**, **VOFFST**, or **VDRAG**. (When using **VSWEEP**, since the volume already exists, use the **VATT** command to assign attributes before sweeping.)
3. When you are using **VEXT**, **VROTAT**, **VOFFST**, or **VDRAG**, enables clearing of the pattern area mesh when volume generations are done. (When you are using **VSWEEP**, if selected, the area meshes on the pattern (source), target, and/or side areas clear when volume sweeping is done.)
4. Neither **EXTOPT,VSWE,AUTO** nor **EXTOPT,VSWE,TETS** will be affected by **EXTOPT,ON** or **EXTOPT,OFF**.

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Mesh>Volume Sweep>Sweep Opts**

**Main Menu>Preprocessor>Modeling>Operate>Extrude>Elem Ext Opts**

**EXTREM**,  $NVAR1$ ,  $NVAR2$ ,  $NINC$ 

**Lists the extreme values for variables.**

[POST26: Listing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

 **$NVAR1$ ,  $NVAR2$ ,  $NINC$** 

List extremes for variables  $NVAR1$  through  $NVAR2$  in steps of  $NINC$ . Variable range defaults to its maximum.  $NINC$  defaults to 1.



## Notes

Lists the extreme values (and the corresponding times) for stored and calculated variables. Extremes for stored variables are automatically listed as they are stored. Only the real part of a complex number is used. Extreme values may also be assigned to parameters [**\*GET**].

## Menu Paths

**Main Menu>TimeHist Postpro>List Extremes**

**EXUNIT**, *Ldtype*, *Load*, *Untype*, *Name*

**Indicates units assumed for an interface load for ANSYS to CFX transfer.**

POST1: Special Purpose

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *Ldtype*

Load type:

#### **SURF**

Surface load

#### **VOLU**

Volumetric load

### *Load*

Surface loads:

#### **DISP**

Displacement

#### **TEMP**

Temperature

#### **HFLU**

Heat flux

Volumetric loads:

#### **DISP**

Displacement

#### **FORC**

Force

#### **HGEN**

Heat generation

### *Untype*

Unit type:

#### **COMM**

Pre-defined unit

#### **USER**

User-specified unit

**Name**

Commonly used pre-defined unit name or user-specified unit name.

**SI**

International System of Units (meter-kilogram-second)

**FT**

English System of Units (feet-pound-second)

In SI units, surface loads are: DISP m, TEMP K, and HFLU W/m<sup>2</sup> and volumetric loads are: DISP m, FORC N/m<sup>3</sup>, and HGEN W/m<sup>3</sup>.

In English units, surface loads are: DISP ft, TEMP F, and HFLU BTU/sec-ft<sup>2</sup> and volumetric loads are: DISP ft, FORC pdl/ft<sup>3</sup>, and HGEN BTU/sec-ft<sup>3</sup>.

**Notes**

This command only indicates the assumed units. It does not perform unit conversions.

A pdl is a poundal, and 1 lbf = 32.174.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**This command cannot be accessed from a menu.**

---

---

## F Commands

**F**, *NODE*, *Lab*, *VALUE*, *VALUE2*, *NEND*, *NINC*

**Specifies force loads at nodes.**

SOLUTION: FE Forces

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### **NODE**

Node at which force is to be specified. If ALL, *NEND* and *NINC* are ignored and forces are applied to all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

### **Lab**

Valid force label. Structural labels: FX, FY, or FZ (forces); MX, MY, or MZ (moments). Thermal labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow). Fluid labels: FLOW (fluid flow). Electric labels: AMPS (current flow), CHRG (electric charge). Magnetic labels: FLUX (magnetic flux); CSGX, CSGY, or CSGZ (magnetic current segments). FLOTRAN labels: FX, FY, or FZ (forces).

For structural analyses, DVOL (fluid mass flow rate) is also a valid label. See [Notes](#) for more information.

### **VALUE**

Force value or table name reference for specifying tabular boundary conditions. To specify a table, enclose the table name in percent signs (%), e.g., **F**, *NODE*, HEAT, %*t abname*%. Use the **\*DIM** command to define a table.

### **VALUE2**

Second force value (if any). If the analysis type and the force allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

### **NEND, NINC**

Specifies the same values of force at the nodes ranging from *NODE* to *NEND* (defaults to *NODE*), in steps of *NINC* (defaults to 1).

## Notes

The available force loads per node correspond to the degrees of freedom listed under "Degrees of Freedom" in the input table for each element type in the [Element Reference](#). If both a force and a constrained degree of freedom [**D**] are specified at the same node, the constraint takes precedence. Forces are defined in the nodal coordinate system. The positive directions of structural forces and moments are along and about the positive nodal axis directions. The node and the degree of freedom label corresponding to the force must be selected [**NSEL**, **DOFSEL**].

For hydrostatic fluid elements ([HSFLD241](#) and [HSFLD242](#)), DVOL is used to specify fluid mass flow rate (with units of mass/time) at the pressure node. This allows fluid to be added or taken out of the fluid elements sharing the pressure node. A fluid density must also be specified (via the **MP** command or **TB** command) to apply a volume change corresponding to the prescribed fluid mass flow rate.

Tabular boundary conditions ( $VALUE = \%tabname\%$ ) are available only for the following labels: Fluid (FLOW), Electric (AMPS), Structural force (FX, FY, FZ, MX, MY, MZ), FLOTRAN (FX, FY, FZ), and Thermal (HEAT, HBOT, HE2, HE3, . . . , HTOP). Tabular boundary conditions are valid only in static (**ANTYPE**,STATIC), full transient (**ANTYPE**,TRANS), and modal superposition transient analyses.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharge>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCurrent>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>ImprCurr>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Node Components**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>NodePSD>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flow>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharge>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCurrent>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>ImprCurr>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Node Components**  
**Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>NodePSD>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flow>On Nodes**

## /FACET,Lab

Specifies the facet representation used to form solid model displays.

[GRAPHICS:Style](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Lab

Valid labels:

#### FINE

Use finer tessellation to increase the number of facets for the display. Provides the best representation (but decreases speed of operation).

#### NORML

Use the basic number of facets for the display (default).

#### COAR

Use a limited number of facets for the display. This option will increase the speed of the operations, but may produce poor representations for some imported models.

**WIRE**

Display model with a wireframe representation (fast, but surfaces will not be shown).

**Command Default**

Basic number of facets.

**Notes**

Specifies the facet (or polygon) representation used to form solid model displays. Used only with the **APLOT**, **ASUM**, **VPLOT**, and **VSUM** commands.

This command is valid in any processor.

**Menu Paths**

Utility Menu>PlotCtrls>Style>Solid Model Facets

**FATIGUE**

Specifies "Fatigue data status" as the subsequent status topic.

PREP7: Status

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**Notes**

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (JOBNAME.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

**Menu Paths**

Utility Menu>List>Status>General Postproc>Fatigue Calcs

**FC**, MAT, Lab1, Lab2, DATA1, DATA2, DATA3, DATA4, DATA5, DATA6

**Provides failure criteria information and activates a data table to input temperature-dependent stress and strain limits.**

PREP7: Materials

POST1: Failure Criteria

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

**MAT**

Material reference number. You can define failure criteria for up to 250 different materials.

**Lab1**

Type of data.

**TEMP**

Temperatures. Each of the materials you define can have a different set of temperatures to define the failure criteria.

**EPEL**

Strains.

**S**

Stresses.

**Lab2**

Specific criteria. Not used if *Lab1* = TEMP.

**XTEN**

Allowable tensile stress or strain in the x-direction. (Must be positive.)

**XCMP**

Allowable compressive stress or strain in the x-direction. (Defaults to negative of XTEN.)

**YTEN**

Allowable tensile stress or strain in the y-direction. (Must be positive.)

**YCMP**

Allowable compressive stress or strain in the y-direction. (Defaults to negative of YTEN.)

**ZTEN**

Allowable tensile stress or strain in the z-direction. (Must be positive.)

**ZCMP**

Allowable compressive stress or strain in the z-direction. (Defaults to negative of ZTEN.)

**XY**

Allowable XY stress or shear strain. (Must be positive.)

**YZ**

Allowable YZ stress or shear strain. (Must be positive.)

**XZ**

Allowable XZ stress or shear strain. (Must be positive.)

**XYCP**

XY coupling coefficient (Used only if *Lab1* = S). Defaults to -1.0.

**YZCP**

YZ coupling coefficient (Used only if *Lab1* = S). Defaults to -1.0.

**XZCP**

XZ coupling coefficient (Used only if *Lab1* = S). Defaults to -1.0.

**DATA1, DATA2, DATA3, . . . , DATA6**

Description of *DATA1* through *DATA6*.

**T1, T2, T3, T4, T5, T6**

Temperature at which limit data is input. Used only when *Lab1* = TEMP.

**V1, V2, V3, V4, V5, V6**

Value of limit stress or strain at temperature T1 through T6. Used only when *Lab1* = S or EPEL.

## Notes

The data table can be input in either PREP7 or POST1. This table is used only in POST1. When you postprocess failure criteria results defined via the **FC** command (**PLESOL**, **PRESOL**, **PLNSOL**, **PRNSOL**, **PRRSOL**, etc.), the active coordinate system must be the coordinate system of the material being analyzed. You do this using **RSYS**, **SOLU**. For layered applications, you also use the **LAYER** command. See the specific element documentation in the *Element Reference* for information on defining your coordinate system for layers.

Some plotting and printing functions will not support Failure Criteria for your PowerGraphics displays. This could result in minor changes to other data when Failure Criteria are applied. See the appropriate plot or print command documentation for more information .

## Menu Paths

**Main Menu>General Postproc>Failure Criteria>Add/Edit**  
**Main Menu>General Postproc>Failure Criteria>Temp Variation**  
**Main Menu>Preprocessor>Material Props>Failure Criteria>Add/Edit**  
**Main Menu>Preprocessor>Material Props>Failure Criteria>Temp Variation**

---

## FCCHECK

Checks both the strain and stress input criteria for all materials.

PREP7: Materials  
 POST1: Failure Criteria

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

## Notes

Issue the **FCCHECK** command to check the completeness of the input during the input phase.

## Menu Paths

**Main Menu>General Postproc>Failure Criteria>Criteria Check**  
**Main Menu>Preprocessor>Material Props>Failure Criteria>Criteria Check**

---

## FCDELE, MAT

Deletes previously defined failure criterion data for the given material.

PREP7: Materials  
 POST1: Failure Criteria

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### **MAT**

Material number. Deletes all **FC** command input for *this* material.

A value of ALL deletes all **FC** command input for *all* materials.

## Notes

This command is also valid in POST1.

## Menu Paths

**Main Menu>General Postproc>Failure Criteria>Delete**  
**Main Menu>Preprocessor>Material Props>Failure Criteria>Delete**

## FCLIST, MAT, --, TEMP

To list what the failure criteria is that you have input.

PREP7: Materials  
 POST1: Failure Criteria

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### MAT

Material number (defaults to ALL for all materials).

--

Unused field.

### TEMP

Temperature to be evaluated at (defaults to TUNIF).

## Notes

This command allows you to see what you have already input for failure criteria using the FC commands.

## Menu Paths

**Main Menu>General Postproc>Failure Criteria>List**  
**Main Menu>Preprocessor>Material Props>Failure Criteria>List**

## FCUM, Oper, RFACT, IFACT

Specifies that force loads are to be accumulated.

SOLUTION: FE Forces

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### Oper

Accumulation key:

#### REPL

Subsequent values replace the previous values (default).

#### ADD

Subsequent values are added to the previous values.

#### IGNO

Subsequent values are ignored.



**RFACT**

Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

**IFACT**

Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

## Command Default

Replace previous values.

## Notes

Allows repeated force load (force, heat flow, etc.) values to be replaced, added, or ignored. Operations apply to the selected nodes [NSEL], and the force labels corresponding to the selected force labels [DOFSEL]. The operations occur when the next force specifications are defined. For example, issuing the command **F,1,FX,250** after a previous **F,1,FX,200** causes the current value of the force on node 1 in the x-direction to be 450 with the add operation, 250 with the replace operation, or 200 with the ignore operation. Scale factors are also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a force of 700. Scale factors are applied even if no previous values exist. Issue **FCUM,STAT** to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

---

### Note

FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

**FCUM** does not work for tabular boundary conditions.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Forces**

**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Forces**

---

## FCTYP, Oper, Lab

**Activates or removes failure-criteria types for postprocessing.**

[POST1: Failure Criteria](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### Oper

Operation key:

#### ADD

Activate failure-criteria types. This option is the default behavior.

**DELE**

Remove failure-criteria types.

**Lab**

Valid failure-criteria labels. If ALL, select all available (including user-defined) failure criteria.

**EMAX**

Maximum strain criterion (default)

**SMAX**

Maximum stress criterion (default)

**TWSI**

Tsai-Wu strength index (default)

**TWSR**

Inverse of Tsai-Wu strength ratio index (default)

**HFIB**

Hashin fiber failure criterion

**HMAT**

Hashin matrix failure criterion

**PFIB**

Puck fiber failure criterion

**PMAT**

Puck inter-fiber (matrix) failure criterion

**USR1 through USR9**

User-defined failure criteria

## Command Default

**FCTYP**, ADD, EMAX, SMAX, TWSI, TWSR

## Notes

The **FCTYP** command modifies the list of active failure criteria.

By default, active failure criteria include EMAX, SMAX, TWSI, and TWSR.

The command affects any subsequent postprocessing listing and plotting commands (such as **PRESOL**, **PRNSOL**, **PLESOL**, **PLNSOL**, and **ETABLE**).

A single **FCTYP** command allows up to six failure-criteria labels. If needed, reissue the command to activate or remove additional failure-criteria types.

## Menu Paths

**Menupath to be provided at release.**

---

**FDELE**, *NODE*, *Lab*, *NEND*, *NINC*

Deletes force loads on nodes.

[SOLUTION: FE Forces](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***NODE***

Node for which force is to be deleted. If ALL, *NEND* and *NINC* are ignored and forces are deleted on all selected nodes [[NSEL](#)]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

***Lab***

Valid force label. If ALL, use all appropriate labels. Structural labels: FX, FY, or FZ (forces); MX, MY, or MZ (moments). Thermal labels: HEAT, HBOT, HE2, HE3, . . . , HTOP (heat flow). Fluid labels: FLOW (fluid flow). Electric labels: AMPS (current flow), CHRGE (electric charge). Magnetic labels: FLUX (magnetic flux); CSGX, CSGY, or CSGZ (magnetic current segments). FLOTRAN labels: FX, FY, or FZ (forces).

***NEND*, *NINC***

Delete forces from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1).

**Notes**

The node and the degree of freedom label corresponding to the force must be selected [[NSEL](#), [DOFSEL](#)].

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Forces>On All Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharge>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCurrent>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>ImprCurr>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Flow>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Node Components**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>NodePSD>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flow>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Forces>On All Nodes**  
**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharge>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCurrent>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>ImprCurr>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Flow>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Nodes**  
**Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Node Components**  
**Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Nodes**

Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Nodes  
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>NodePSD>On Nodes  
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Nodes  
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flow>On Nodes

---

**/FDELE**, *Ident*, *Stat*

**Deletes a binary file after it is used.**

SESSION: Files

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *Ident*

ANSYS file name identifier. Valid identifiers are: EMAT, ESAV, FULL, SUB, MODE, TRI, DSUB, USUB, OSAV, and SELD. See the [Basic Analysis Guide](#) for file descriptions.

### *Stat*

Keep or delete key:

#### **KEEP**

Keep this file.

#### **DELE**

Delete (or do not write, if not necessary) this file.

## **Command Default**

Keep all files.

## **Notes**

Deletes as soon as possible (or prevents writing) a binary file created by the ANSYS program to save space.

---

### **Caution**

Deleting files that are necessary for the next substep, load step, or analysis will prevent continuation of the run.

This command is valid only at the Begin Level.

## **Menu Paths**

**Utility Menu>File>ANSYS File Options**

---

**FE**, *NEV*, *CYCLE*, *FACT*, *Title*

**Defines a set of fatigue event parameters.**

[POST1: Fatigue](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**NEV**

Reference number for this event (within *MXEV*).

**CYCLE**

Number of required cycles (defaults to 1). If -1, erase all parameters and fatigue stresses for this event.

**FACT**

Scale factor to be applied to all loadings in this event (defaults to 1.0).

**Title**

User defined identification title for this event (up to 20 characters).

## Command Default

Event assigned one cycle, unity scale factor, and no title.

## Notes

Repeat **FE** command to define additional sets of event parameters (*MXEV* limit), to redefine event parameters, or to delete event stress conditions.

The set of fatigue event parameters is associated with all loadings and all locations. See the **FTSIZE** command for the maximum set of events (*MXEV*) allowed.

## Menu Paths

**Main Menu>General Postproc>Fatigue>Assign Events**

**Main Menu>General Postproc>Fatigue>Erase Event Data**

## FEBODY

**Specifies "Body loads on elements" as the subsequent status topic.**

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (*Jobname.LOG*) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Solution>Body Loads

---

## FECONS

Specifies "Constraints on nodes" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [\[STAT\]](#) topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the [STAT](#) command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Solution>DOF Constraints

---

## FEFOR

Specifies "Forces on nodes" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [\[STAT\]](#) topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the [STAT](#) command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Solution>Forces

---

**FELIST**, *NEV1*, *NEV2*, *NINC*

**Lists the fatigue event parameters.**

[POST1: Fatigue](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

***NEV1***, ***NEV2***, ***NINC***

List event parameters from *NEV1* (defaults to 1) to *NEV2* (defaults to *NEV1*) in steps of *NINC* (defaults to 1). If *NEV1* = ALL, *NEV2* and *NINC* are ignored and all events are listed. Fatigue event parameters are defined with the **FE** command.

**Menu Paths**

**Main Menu>General Postproc>Fatigue>List Event Data**

---

**FESURF**

**Specifies "Surface loads on elements" as the subsequent status topic.**

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

**Menu Paths**

**Utility Menu>List>Status>Solution>Surface Loads**

---

**FILE**, *Fname*, *Ext*, --

**Specifies the data file where results are to be found.**

[POST1: Set Up](#)

[POST26: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

**Ext**

Filename extension (8 character maximum).

If *Fname* is blank, the extension defaults to RST (for structural, fluid, or coupled-field analyses), to RTH (for thermal or electrical analyses), to RMG (for magnetic analyses), or to RFL (for FLOTRAN analyses). For postprocessing reduced structural analyses in POST26, use the RDSP extension for displacements from transient dynamic analyses or the RFRQ extension from harmonic response analyses. For postprocessing contact results corresponding to the initial contact state in POST1, use the RCN extension.

--

Unused field.

## Command Default

Use the result file with the `Jobname` as *Fname* and with the extension corresponding to the analysis type.

## Notes

Specifies the ANSYS data file where the results are to be found for postprocessing.

## Menu Paths

**Main Menu>General Postproc>Data & File Opts**

**Main Menu>TimeHist Postpro>Settings>File**

**Utility Menu>File>List>Binary Files**

**Utility Menu>List>Files>Binary Files**

---

## FILEAUX2, *Fname*, *Ident*, --

**Specifies the binary file to be dumped.**

[AUX2: Binary Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to the current `Jobname` if *Ident* is specified.

***Ident***

ANSYS filename identifier. See the [Basic Analysis Guide](#) for file descriptions and identifiers. If not an ANSYS identifier, *Ident* will be used as the filename extension.

--

Unused field.

## Notes

Specifies the binary file to be dumped with the **DUMP** command.



## Menu Paths

Utility Menu>File>List>Binary Files  
Utility Menu>List>Files>Binary Files

**FILEAUX3**, *Fname*, *Ext*, --

Specifies the results file to be edited.

[AUX3: Results Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to the current `Jobname` if *Ext* is specified.

### *Ext*

Filename extension (8 character maximum).

--

Unused field.

## Notes

Specifies the results file to be edited.

## Menu Paths

**This command cannot be accessed from a menu.**

**FILEDISP**, *Fname*, *Ext*

Specifies the file containing the graphics data.

[DISPLAY: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

### *Ext*

Filename extension (8 character maximum).

## Notes

Specifies the input file containing the graphics data (defaults to `File.GRPH`).

## Menu Paths

It is part of the DISPLAY program.

**FILL**, *NODE1*, *NODE2*, *NFILL*, *NSTRT*, *NINC*, *ITIME*, *INC*, *SPACE*

**Generates a line of nodes between two existing nodes.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NODE1*, *NODE2***

Beginning and ending nodes for fill-in. *NODE1* defaults to next to last node specified, *NODE2* defaults to last node specified. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### ***NFILL***

Fill *NFILL* nodes between *NODE1* and *NODE2* (defaults to  $|NODE2-NODE1|-1$ ). *NFILL* must be positive.

### ***NSTRT***

Node number assigned to first filled-in node (defaults to  $NODE1 + NINC$ ).

### ***NINC***

Add this increment to each of the remaining filled-in node numbers (may be positive or negative). Defaults to the integer result of  $(NODE2-NODE1)/(NFILL + 1)$ , i.e., linear interpolation. If the default evaluates to zero, or if zero is input, *NINC* is set to 1.

### ***ITIME*, *INC***

Do fill-in operation a total of *ITIMES*, incrementing *NODE1*, *NODE2* and *NSTRT* by *INC* each time after the first. *ITIME* and *INC* both default to 1.

### ***SPACE***

Spacing ratio. Ratio of last division size to first division size. If  $> 1.0$ , divisions increase. If  $< 1.0$ , divisions decrease. Ratio defaults to 1.0 (uniform spacing).

## Notes

Generates a line of nodes (in the active coordinate system) between two existing nodes. The two nodes may have been defined in any coordinate system. Nodal locations and rotation angles are determined by interpolation. Any number of nodes may be filled-in and any node number sequence may be assigned. See the **CSCIR** command when filling across the 180° singularity line in a non-Cartesian system.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Nodes>Fill between Nds**

**FILLDATA**, *IR*, *LSTRT*, *LSTOP*, *LINC*, *VALUE*, *DVAL*

**Fills a variable by a ramp function.**

POST26: Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***IR***

Define data table as variable *IR* (2 to *NV* [**NUMVAR**]).

***LSTRT***

Start at location *LSTRT* (defaults to 1).

***LSTOP***

Stop at location *LSTOP* (defaults to maximum location as determined from data previously stored).

***LINC***

Fill every *LINC* location between *LSTRT* and *LSTOP* (defaults to 1).

***VALUE***

Value assigned to location *LSTRT*.

***DVAL***

Increment value of previous filled location by *DVAL* and assign sum to next location to be filled (may be positive or negative.)

**Notes**

Locations may be filled continuously or at regular intervals (*LINC*). Previously defined data at a location will be overwritten.

**Menu Paths**

**Main Menu>TimeHist Postpro>Table Operations>Fill Data**

**/FILENAME**, *Fname*, *Key*

**Changes the Jobname for the analysis.**

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Fname***

Name (32 characters maximum) to be used as the *Jobname*. Defaults to the initial *Jobname* as specified on the ANSYS execution command, or to *file* if none specified.

***Key***

Specify whether to use the existing log, error, lock, and page files or start new files.

**0, OFF**

Continue using current log, error, lock, and page files.

**1, ON**

Start new log, error, lock, and page files (old log and error files are closed and saved, but old lock and page files are deleted). Existing log and error files are appended.

## Notes

All subsequently created files will be named with this `Jobname` if `Key = 0`. Use `Key = 1` to start new log, error, lock, and page files. The previous `Jobname` is typically defined on the ANSYS program execution line (see the [Operations Guide](#)). This command is useful when different groups of files created throughout the run are to have different names. For example, the command may be used before each substructure pass to avoid overwriting files or having to rename each file individually.

This command is valid only at the Begin level.

## Menu Paths

Utility Menu>File>Change Jobname

---

## FINISH

Exits normally from a processor.

SESSION: Processor Entry

DISPLAY: Action

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Exits any of the ANSYS processors or the DISPLAY program. For the ANSYS processors, data will remain intact in the database but the database is not automatically written to a file (use the **SAVE** command to write the database to a file). See also the **/QUIT** command for an alternate processor exit command. If exiting POST1, POST26, or OPT, see additional notes below.

POST1: Data in the database will remain intact, including the POST1 element table data, the path table data, the fatigue table data, and the load case pointers.

POST26: Data in the database will remain intact, except that POST26 variables are erased and specification commands (such as **FILE**, **PRTIME**, **NPRINT**, etc.) are reset. Use the **/QUIT** command to exit the processor and bypass these exceptions.

OPT: Current optimization data are written to `File.OPT` for possible resume later [**OPRESU**]. See also the **OPSAVE** command to write optimization data.

This command is valid in any processor. This command is not valid at the Begin level.

## Menu Paths

Main Menu>DesignXplorer>Solution>Solve

Main Menu>Finish

---

**FILOT**, *Option, Filename, Ext, Dir, REQLAYER, FSPRINT***Plots a FiberSIM model.**

PREP7: Elements

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

**Option**

One of the following options:

**FIBER**

Displays a FiberSIM model with graphic settings.

**ORIENT**

Displays orientation lines on a FiberSIM model, as follows:

**/DEVICE**,VECTOR -- Activates vector mode.**/PSYMB**,LAYR,*N* -- Activates layer plotting, where *N* indicates the requested layer number.**BOTH**Displays both FiberSIM and ANSYS models. To activate translucency, issue **/TRLCY**,*TLEVEL*, where *TLEVEL* represents the translucency level (0.0 = opaque to 1.0 = transparent). The recommended value is 0.9.**NOTHIC**

Displays elements that include no FiberSIM layers. This option changes the currently selected set of elements.

**RESTORE**After issuing this command with *Option* = NOTHIC, this option restores the previously selected element set.**Filename**

Name of the file containing the FiberSIM information.

**Ext**Extension of the name of the file containing the FiberSIM information. The default is `.xml`.**Dir**

Directory where the file is located.

**REQLAYER**

The layer number being requested.

**FSPRIN**

Debug switch:

**0**

No debug. This value is the default.

**1**

Debug output. Use the debugging option for very small files only.

**Notes**The **FILOT** command plots **FiberSIM** models, and the interaction of those models with ANSYS models.The FIBER, ORIENT, and BOTH options plot the FiberSIM facets as **SHELL181** elements.

If *Option* = NOTHC, all other arguments are ignored.

For more information, see [The FiberSIM-ANSYS Interface](#) in the *Structural Analysis Guide*.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**FITEM**, *NFIELD*, *ITEM*, *ITEMY*, *ITEMZ*

**Identifies items chosen by a picking operation (GUI).**

DATABASE: Picking

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NFIELD*

Field number on the command which uses the picking data. (Count the command name as a field, so that a 2 indicates the first command argument, 3 the second command argument, etc.) The corresponding field on the command will have a P51X label.

### *ITEM*

Entity number of the entity picked. Negative entity numbers are used to indicate a range of entities. If the item picked is a coordinate location, then this field represents the X-coordinate. See also the **FLST** command.

### *ITEMY*, *ITEMZ*

Y and Z coordinates of a picked coordinate location. *ITEM* represents the X coordinate. See also the **FLST** command.

## Notes

This is a command generated by the GUI and will appear in the log file (*Jobname.LOG*) if graphical picking is used. This command is *not* intended to be typed directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

On the log file, a set of **FITEM** commands is preceded by one **FLST** command which defines the picking specifications for that pick operation. The data listed in the **FITEM** commands are used by the first subsequent command containing a P51X label in one of its fields.

---

### Caution

For a given entity type, a list containing an *ITEM* that is larger than the maximum defined entity, could deplete the system memory and produce unpredictable results.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**FJ**, *ELEM*, *LABEL*, *VALUE*

**Specify forces or moments on the components of the relative motion of a joint element.**

[SOLUTION: FE Forces](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

***ELEM***

Element number or ALL to specify all joint elements.

***LABEL***

Valid labels:

**FX**

Force in local x direction.

**FY**

Force in local y direction.

**FZ**

Force in local z direction.

**MX**

Moment about local x axis.

**MY**

Moment about local y axis.

**MZ**

Moment about local z axis.

***VALUE***

Value of the label.

## Notes

Valid for [MPC184](#) (joint options in KEYOPT(1)).

See [FJDELE](#) for information on deleting forces and moments.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Joint Elems**

**Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Joint Elems**

**FJDELE**, *ELEM*, *LAB*

**Deletes forces (or moments) on the components of the relative motion of a joint element.**

[SOLUTION: FE Forces](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

***ELEM***

Element number, or ALL. (leaving this blank defaults to ALL)

***LAB***

Valid labels are:

**FX**

Force in local x direction.

**FY**

Force in local y direction.

**FZ**

Force in local z direction.

**MX**

Moment about local x axis.

**MY**

Moment about local y axis.

**MZ**

Moment about local z axis.

**ALL, or (blank)**

Delete all valid forces or moments.

## Notes

Valid for [MPC184](#) (joint options in KEYOPT(1)).

See [FJ](#) for information on specifying forces (or moments).

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Forces>On Joint Elems**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Joint Elems**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Forces>On Joint Elems**  
**Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Joint Elems**

---

## FJLIST, *Elem*

**Lists forces and moments applied on joint elements.**

[SOLUTION:FE Forces](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Elem*

Element number or ALL (or blank). Lists joint element forces and moments on the specified element(s).

## Notes

Valid for [MPC184](#) joint elements. See [FJ](#) for information on specifying forces and moments.

## Menu Paths

**Utility Menu>List>Loads>Joint Element Forces>On Picked Element**

---



**FK**, *KPOI*, *Lab*, *VALUE*, *VALUE2*

**Defines force loads at keypoints.**

**SOLUTION:** Solid Forces

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### ***KPOI***

Keypoint at which force is to be specified. If ALL, apply to all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

#### ***Lab***

Valid force label. Structural labels: FX, FY, or FZ (forces); MX, MY, or MZ (moments). Thermal labels: HEAT, HBOT, HE2, HE3, . . . , HTOP (heat flow). Fluid labels: FLOW (fluid flow). Electric labels: AMPS (current flow), CHRG (electric charge). Magnetic labels: FLUX (magnetic flux); CSGX, CSGY, or CSGZ (magnetic current segments).

#### ***VALUE***

Force value or table name reference for specifying tabular boundary conditions. To specify a table, enclose the table name in percent signs (%), e.g., **FK**, *KPOI*, HEAT,%*tablename*%). Use the **\*DIM** command to define a table.

#### ***VALUE2***

Second force value (if any). If the analysis type and the force allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

## **Notes**

Forces may be transferred from keypoints to nodes with the **FTRAN** or **SBCTRAN** commands. See the **F** command for a description of force loads.

Tabular boundary conditions (*VALUE* = %*tablename*%) are available only for the following labels: Fluid (FLOW), Electric (AMPS), Structural force (FX, FY, FZ, MX, MY, MZ), and Thermal (HEAT, HBOT, HE2, HE3, . . . , HTOP).

This command is also valid in PREP7.

## **Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharge>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCurrent>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>ImprCurr>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>NodePSD>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flow>On Keypoints**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharge>On Keypoints**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCurrent>On Keypoints**

**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>ImprCurr>On Keypoints**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Keypoints**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Keypoints**  
**Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Keypoints**  
**Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Keypoints**  
**Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>NodePSD>On Keypoints**  
**Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Keypoints**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flow>On Keypoints**

## FKDELE, *KPOI*, *Lab*

Deletes force loads at a keypoint.

SOLUTION: Solid Forces

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *KPOI*

Keypoint at which force is to be deleted. If ALL, delete forces at all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

### *Lab*

Valid force label. If ALL, use all appropriate labels. See the **FDELE** command for labels.

## Notes

Deletes force loads (and all corresponding finite element loads) at a keypoint. See the **FDELE** command for details.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Forces>On All KPs**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharge>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCurrent>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>ImprCurr>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Flow>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>NodePSD>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Keypoints**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flow>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Forces>On All KPs**  
**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharge>On Keypoints**

**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCurrent>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>ImprCurr>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Flow>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>NodePSD>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Keypoints**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flow>On Keypoints**

## **FKLIST**, *KPOI*, *Lab*

**Lists the forces at keypoints.**

**SOLUTION:** Solid Forces

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *KPOI*

List forces at this keypoint. If ALL (default), list for all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

### *Lab*

Force label to be listed (defaults to ALL). See the **DOFSEL** command for labels.

## **Notes**

Listing applies to the selected keypoints [**KSEL**] and the selected force labels [**DOFSEL**].

This command is valid in any processor.

## **Menu Paths**

**Utility Menu>List>Loads>Forces>On All Keypoints**

**Utility Menu>List>Loads>Forces>On Picked KPs**

## **FL**, *NLOC*, *NODE*, *SCFX*, *SCFY*, *SCFZ*, *Title*

**Defines a set of fatigue location parameters.**

**POST1:** Fatigue

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *NLOC*

Reference number for this location (within *MXLOC*). When defining a new location, defaults to lowest unused location. If the specified *NODE* is already associated with a location, *NLOC* defaults to that existing location.

**NODE**

Node number corresponding to this location (must be unique). Used only to associate a node with a new location or to find an existing location (if *NLOC* is not input). If *NODE* = -1 (or redefined), erase all parameters and fatigue stresses for this location.

**SCFX, SCFY, SCFZ**

Stress concentration factors applied to the total stresses. Factors are applied in the global X, Y, and Z directions unless the axisymmetric option of the **FSSECT** is used (i.e., *RHO* is nonzero), in which case the factors are applied in the section x, y, and z (radial, axial, and hoop) directions.

**Title**

User-defined title for this location (up to 20 characters).

## Notes

Repeat **FL** command to define additional sets of location parameters (*MXLOC* limit), to redefine location parameters, or to delete location stress conditions.

One location must be defined for each node of interest and only one node can be associated with each location. See the **FTSIZE** command for the maximum locations (*MXLOC*) allowed. A location will be automatically defined for a node not having a location when the **FSSECT**, **FSNODE**, or **FS** command is issued. Automatically defined locations are assigned the lowest available location number, unity stress concentration factors, and no title.

## Menu Paths

**Main Menu>General Postproc>Fatigue>Stress Locations**

---

**FLDATA**, *Name*, *Label*, *Value*

Sets up a FLOTRAN analysis.

[PREP7: FLOTRAN Options](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**Name**

The name identifying the group of FLOTRAN parameters being defined or controlled on this command.

**Label**

The label of the specific FLOTRAN parameter being input or controlled. *Label* determines the meaning of the *Value* argument.

**Value**

The numeric value of an input item, the logical value of a switch (T or F, for example), or an alphanumeric label, depending on the *Label* argument.

## Notes

The **FLDATA** command is used to define FLOTRAN-specific input data, solution controls, and output controls. It is valid only with the FLOTRAN CFD option.

The **FLDATA** command controls groups of FLOTRAN parameters, and the group name is input as the first argument, *Name*.

To give you the ability to link directly to the documentation on any of these groups, we have documented each group under its own name--from **FLDATA1** through **FLDATA36**. For example, documentation on **FLDATA** with the first argument set to SOLU is documented as the **FLDATA1** command. You may enter the command with either name--**FLDATA** or **FLDATA1**, and you must remember to input the appropriate first argument, as shown in the list below.

### Option and Output Control Commands

<b>FLDATA1</b> ,SOLU	Controls which features of the solution algorithm are activated.
<b>FLDATA2</b> ,ITER	Sets iteration and output controls for steady state analyses.
<b>FLDATA3</b> ,TERM	Sets the convergence monitors for the pressure and temperature equations.
<b>FLDATA4</b> ,TIME	Sets controls for transient analyses based on transient time and convergence monitors or sets time integration method.
<b>FLDATA4A</b> ,STEP	Sets output controls for transient analyses based on number of time steps.
<b>FLDATA5</b> ,OUTP	Sets output and storage controls.
<b>FLDATA6</b> ,CONV	Controls the output of the convergence monitor.

### Property Definition Commands

<b>FLDATA7</b> ,PROT	Specifies the type of fluid property.
<b>FLDATA8</b> ,NOMI	Specifies the NOMI coefficient of the fluid property equation.
<b>FLDATA9</b> ,COF1	Specifies the COF1 coefficient of the fluid property equation.
<b>FLDATA10</b> ,COF2	Specifies the COF2 coefficient of the fluid property equation.
<b>FLDATA11</b> ,COF3	Specifies the COF3 coefficient of the fluid property equation.
<b>FLDATA12</b> ,PROP	Sets the property update frequency flag.
<b>FLDATA13</b> ,VARY	Sets the property variation flag.

### Operating Condition Commands

<b>FLDATA14</b> ,TEMP	Specifies the reference temperature.
<b>FLDATA15</b> ,PRES	Specifies the reference pressure.
<b>FLDATA16</b> ,BULK	Specifies the bulk modulus parameter.
<b>FLDATA17</b> ,GAMM	Specifies the specific heat ratio.

### Algebraic Solver Commands

<b>FLDATA18</b> ,METH	Selects the algebraic solver.
<b>FLDATA19</b> ,TDMA	Specifies the number of TDMA sweeps.
<b>FLDATA20</b> ,SRCH	Specifies the number of conjugate direction search vectors.
<b>FLDATA20A</b> ,PGMR	Specifies the amount of fill-in when preconditioning the coefficient matrix.
<b>FLDATA20B</b> ,PBCGM	Specifies the number of fill-ins for the ILU preconditioner.
<b>FLDATA21</b> ,CONV	Specifies the solver convergence criterion.
<b>FLDATA22</b> ,MAXI	Maximum number of semi-direct iterations.
<b>FLDATA23</b> ,DELT	Specifies the solver minimum normalized rate of change.

### Turbulence Commands

<b>FLDATA24</b> ,TURB	Sets the turbulence model and the constants used in the Standard $k-\epsilon$ Model and the Zero Equation Turbulence Model.
<b>FLDATA24A</b> ,RNGT	Sets constants for the Re-Normalized Group Turbulence Model (RNG).
<b>FLDATA24B</b> ,NKET	Sets constants for the $k-\epsilon$ Turbulence Model due to Shih (NKE).
<b>FLDATA24C</b> ,GIRT	Sets constants for the Nonlinear Turbulence Model of Girimaji (GIR).
<b>FLDATA24D</b> ,SZLT	Sets constants for the Shih, Zhu, Lumley Turbulence Model (SZL).
<b>FLDATA24E</b> ,SKWT	Sets constants for the $k-\omega$ turbulence model.
<b>FLDATA24F</b> ,SST1	Sets the turbulent production clip factor for the Shear Stress Transport (SST) turbulence model.
<b>FLDATA24G</b> ,SST1	Sets constants in the $k-\omega$ regime for the Shear Stress Transport (SST) turbulence model.
<b>FLDATA24H</b> ,SST2	Sets constants in the $k-\epsilon$ regime for the Shear Stress Transport (SST) turbulence model.

### Stability Control Commands

<b>FLDATA25</b> ,RELX	Sets solution and property relaxation factors.
<b>FLDATA26</b> ,STAB	Sets stability controls.

### Miscellaneous Commands

<b>FLDATA27</b> ,PRIN	Controls printing flags.
<b>FLDATA28</b> ,MODR	Specifies that variable results are to be replaced.
<b>FLDATA29</b> ,MODV	Re-initializes a results variable.
<b>FLDATA30</b> ,QUAD	Controls the quadrature orders.
<b>FLDATA31</b> ,CAPP	Specifies dependent variable caps.
<b>FLDATA32</b> ,REST	Controls restart options.
<b>FLDATA33</b> ,ADVM	Specifies the approach to discretize the advection term.
<b>FLDATA34</b> ,MIR	Sets modified inertial relaxation factors.
<b>FLDATA35</b> ,VFTOL	Specifies tolerances for the lower and upper bound of the volume fraction.
<b>FLDATA36</b> ,AMBV	Specifies ambient reference values outside of the fluid for the volume of fluid (VOF) method.
<b>FLDATA37</b> ,ALGR	Specifies segregated solution or film coefficient algorithms.
<b>FLDATA38</b> ,MASS	Specifies the mass type for a fluid transient analysis.
<b>FLDATA39</b> ,REMESH	Specifies remeshing parameters for transient fluid flow and fluid-solid interaction analyses.
<b>FLDATA40</b> ,WADV	Controls activation of thermal stabilization near walls.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Advection**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Algorithm Ctrl**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl**  
**Main Menu>Solution>FLOTRAN Set Up>Advection**

Main Menu>Solution>FLOTRAN Set Up>Algorithm Ctrl

Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl

**FLDATA1**, SOLU, *Label*, *Value*

Controls which features of the solution algorithm are activated.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

#### SOLU

Enter the word SOLU in this field.

**FLDATA1**, SOLU is the **FLDATA** command with its first argument set to SOLU. It can be entered into the program as either **FLDATA1**, SOLU, *Label*, *Value* or **FLDATA**, SOLU, *Label*, *Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

#### *Label*

Solution algorithm choices:

##### **FLOW**

Solves the momentum and pressure equations. Defaults to TRUE (on).

##### **TURB**

Activates the turbulence model. Defaults to FALSE (off).

##### **TEMP**

Solves the temperature equation. Defaults to FALSE (off).

##### **COMP**

Uses the compressible algorithm. Defaults to FALSE (off).

##### **SWRL**

Activates the swirl option. Defaults to FALSE (off).

##### **TRAN**

Activates the transient solution algorithm. Defaults to FALSE (off).

##### **SPEC**

Activates multiple species transport. Defaults to FALSE (off).

##### **IVSH**

Activates viscous heating for incompressible flow. Defaults to FALSE (off).

##### **VOF**

Activates volume of fluid method. Defaults to FALSE (off).

##### **SFTS**

Activates surface tension. Defaults to FALSE (off).

##### **ALE**

Activates Arbitrary Lagrangian-Eulerian formulation. Defaults to FALSE (off).

##### **RDSF**

Activates the surface-to-surface radiation solution. Defaults to TRUE (on).

#### *Value*

Value controlling *Label*:

**TRUE or T**

Turn this feature on.

**FALSE or F**

Turn this feature off.

## Notes

Repeat command to set each *Label* as required.

The analyst must choose the appropriate features, as FLOTRAN will not determine, for example, whether or not the case is turbulent or whether or not the compressible algorithm is appropriate.

SWRL should only be activated for axisymmetric cases when there is a velocity component normal to the axisymmetric plane.

Generally, the use of the compressible algorithm (COMP) is not warranted for Mach numbers less than 0.3. Density may be assumed to vary via the ideal gas law without activating the compressible option.

If fluid properties are not a function of temperature in non-adiabatic flow problems, it is not necessary to activate the flow (FLOW) and temperature (TEMP) solutions together. First solve the flow problem and then restart to solve the temperature equation.

For compressible flow, FLOTRAN automatically includes viscous heating.

When the Arbitrary Lagrangian-Eulerian (ALE) formulation is on, FLOTRAN allows the fluid nodes to move in a manner that satisfies the displacement boundary conditions.

See also the [FLDATA2,ITER](#), [FLDATA3,TERM](#), [FLDATA4,TIME](#), [FLDATA4A,STEP](#), [FLDATA5,OUTP](#), and [FLDATA6,CONV](#) commands for other Solution and Output Controls.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Solution Options**

**Main Menu>Solution>FLOTRAN Set Up>Solution Options**

---

## FLDATA2, ITER, *Label*, *Value*

**Sets iteration and output controls for steady state analyses.**

[PREP7:FLOTRAN Options](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### ITER

Enter the word ITER in this field.

**FLDATA2,ITER** is the **FLDATA** command with its first argument set to ITER. It can be entered into the program as either **FLDATA2,ITER,*Label*,*Value*** or **FLDATA,ITER,*Label*,*Value*** where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.



**Label**

Type of iteration control (dictates the meaning of *Value*):

**EXEC**

*Value* is the number of global iterations, defined as the sequential solution of the governing equations of all the features activated, to be performed during this execution of FLOTRAN if the case is steady state. Defaults to *Value* = 10.

**APPE**

*Value* is the number of steady state global iterations between appends to the results file (*Jobname.RFL*). This feature is used to ensure that restarts can be made from earlier points if unforeseen difficulties occur before the end of the analysis. The default value implies that results will be saved only for the initial (0<sup>th</sup>) iteration and the final global iteration. Saving many intermediate results can produce a large results file. Defaults to *Value* = 0 (implies **FLDATA2**,ITER,EXEC).

If you are creating domain files (*jobname.pv\_0000n*) for use by ICEM CFD's PV3 postprocessing visualization tool, this sets the frequency with which domain files are written. See **FLDATA5** for more details.

**OVER**

*Value* is the number of steady state global iterations between overwrites of a temporary set of results in the results file (*Jobname.RFL*). (This temporary set of results will itself be overwritten by the set stored for *Label* = APPE or at the end of the run.) OVER enables the user to retain the most up-to-date set of results without having the results file continue to grow in size. Defaults to *Value* = 0 (no overwrites will be made).

**Value**

Number of iterations for *Label* above.

**Notes**

Repeat command to set each *Label* as required.

Sets the number of global iterations to control the length of execution for steady state analyses, the frequency with which results are added to the results file (*Jobname.RFL*), and the frequency with which results are overwritten in the results file. Also sets the frequency with which the domain files, used by the ICEM PV3 postprocessing visualization tool, are written. For more information about domain files, see the **FLDATA5**,OUTP,DOMA command.

See also the **FLDATA1**,SOLU, **FLDATA3**,TERM, **FLDATA4**,TIME, **FLDATA4A**,STEP, **FLDATA5**,OUTP, and **FLDATA6**,CONV commands for other Solution and Output Controls.

This command is accessible in the menu if **FLDATA1**,SOLU,TRAN,FALSE has been issued (default).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl**

**Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl**

**FLDATA3**, TERM, Label, Value

Sets the convergence monitors for the degree of freedom set.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**TERM**

Enter the word TERM in this field.

**FLDATA3**,TERM is the **FLDATA** command with its first argument set to TERM. It can be entered into the program as either **FLDATA3**,TERM,Label,Value or **FLDATA**,TERM,Label,Value where Label and Value are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Type of convergence monitor:

**PRES**

Steady-state run will not terminate until the convergence monitor for pressure falls below Value, unless the specified number of global iterations has been executed. Defaults to Value =  $1.0 \times 10^{-8}$ .

**TEMP**

Steady-state run (with the temperature equation solution activated) will not terminate until the convergence monitor for temperature falls below Value, unless the specified number of global iterations has been executed. Defaults to Value =  $1.0 \times 10^{-8}$ .

**VX**

Steady-state run will not terminate until the convergence monitor for X velocity component falls below Value, unless the specified number of global iterations has been executed. Defaults to Value =  $1.0 \times 10^{-2}$ .

**VY**

Steady-state run will not terminate until the convergence monitor for Y velocity component falls below Value, unless the specified number of global iterations has been executed. Defaults to Value =  $1.0 \times 10^{-2}$ .

**VZ**

Steady-state run will not terminate until the convergence monitor for Z velocity component falls below Value, unless the specified number of global iterations has been executed. Defaults to Value =  $1.0 \times 10^{-2}$ .

**ENKE**

Steady-state run will not terminate until the convergence monitor for turbulent kinetic energy falls below Value, unless the specified number of global iterations has been executed. Defaults to Value =  $1.0 \times 10^{-2}$ .

**ENDS**

Steady-state run will not terminate until the convergence monitor for turbulence dissipation falls below Value, unless the specified number of global iterations has been executed. Defaults to Value =  $1.0 \times 10^{-2}$ .

**Value**

Value of convergence monitor criterion, above.

## Notes

Repeat command to set each *Label* as required.

The convergence monitors appear as output for each degree of freedom for each global iteration. The value is calculated for each degree of freedom individually by summing the absolute value of the change in solution between global iterations for all the nodes and dividing it by the sum of the absolute values of the solution for all the nodes. It is an approximation of the normalized rate of change of the solution between global iterations.

All specified criteria must be met before the case is terminated.

If a termination criterion for a specific label is set negative, the termination check is ignored for that particular DOF.

See also the **FLDATA1**,**SOLU**, **FLDATA2**,**ITER**,**FLDATA4**,**TIME**, **FLDATA4A**,**STEP**, **FLDATA5**,**OUTP**, and **FLDATA6**,**CONV** commands for other Solution and Output Controls.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl**  
**Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl**

---

## FLDATA4, TIME, Label, Value

**Sets controls for transient analyses based on transient time and convergence monitors or sets time integration method.**

[PREP7: FLOTRAN Options](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### TIME

Enter the word TIME in this field.

**FLDATA4**,**TIME** is the **FLDATA** command with its first argument set to TIME. It can be entered into the program as either **FLDATA4**,**TIME**,*Label*,*Value* or **FLDATA**,**TIME**,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### Label

Type of transient analysis control or time integration method (dictates the meaning of *Value*):

### STEP

Value controls the time step size. If *Value* is greater than zero, *Value* is the time step size. If *Value* is less than zero, the program chooses the time step, according to the following (*Value* defaults to -1):

#### 1

The resulting time step will be small enough to prevent the passage of an arbitrarily small fluid parcel through more than one element length in a single time step.

**2**

This choice is only applicable for compressible analyses, and is equivalent to the Courant limit. The time step is small enough to prevent a pressure signal from propagating through more than one element during a single time step.

**3**

This choice is only applicable for compressible analyses. The smaller of the two time steps computed for options -1 and -2 is used.

**4**

This choice is applicable to "conduction only" cases (**FLDATA1**,SOLU,FLOW,F). The resulting time step prevents an arbitrarily small "parcel of heat" from diffusing or conducting through more than one element within a time step.

**ISTEP**

*Value* is the time step size for the first time step in an analysis using a FLOTRAN-calculated time step size (STEP = -1 through -4).

**BC**

*Value* is a flag indicating whether a transient boundary condition should be applied as a step change (when *Value* = 0) or as a linear ramp (*Value* = 1). This label is analogous to the ANSYS command **KBC**, except that the default for FLOTRAN is a step change. Tabular boundary conditions do not support ramping and instead apply their full value regardless of the **KBC** setting.

**NUMB**

*Value* is the number of time steps which will be executed unless the transient end time has been encountered first.

**GLOB**

*Value* is the number of global iterations per time step. The user may elect to set this to a high value (typically between 30 and 50), and expect the time step to converge to the convergence criterion set by the **FLDATA3**,TERM command before completing *Value* iterations. Default to *Value* = 10.

**TEND**

*Value* is the transient end time. This time is used for the calculation of ramped transient boundary conditions. The case will execute until this end time if the number of time steps specified is large enough. Defaults to *Value* =  $1.0 \times 10^{+6}$ .

**APPE**

*Value* is the results output frequency based on transient time. Every *Value* seconds in the transient, results are written to the `Jobname.RFL` file. If you are creating domain files (`jobname.pv_0000n`) for use by ICEM CFD's PV3 postprocessing visualization tool, this sets the frequency with which domain files are written. See **FLDATA5** for more details. Defaults to *Value* =  $1.0 \times 10^{+6}$ .

**SUMF**

*Value* is the output summary frequency based on transient time. Every *Value* seconds in the transient, a results summary is written to the `Jobname.PFL` file.

---

**Note**

A summary will always be written when a set of results is written to the `Jobname.RFL` file. Defaults to *Value* =  $1.0 \times 10^{+6}$ .

**OVER**

*Value* is the time interval between overwrites of the temporary set of results in the results file (*Jobname.RFL*). This feature enables the user to retain the most up-to-date set of results without having the results file continue to grow in size. Defaults to *Value* = 0 (no overwrites will be made).

**PRES**

If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for pressure falls below *Value*. Defaults to *Value* =  $1.0 \times 10^{-6}$ .

**TEMP**

If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for temperature falls below *Value*. Defaults to *Value* =  $1.0 \times 10^{-6}$ .

**VX**

If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for X velocity component falls below *Value*. Defaults to *Value* =  $1.0 \times 10^{-2}$ .

**VY**

If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for Y velocity component falls below *Value*. Defaults to *Value* =  $1.0 \times 10^{-2}$ .

**VZ**

If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for Z velocity component falls below *Value*. Defaults to *Value* =  $1.0 \times 10^{-2}$ .

**ENKE**

If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for turbulent kinetic energy falls below *Value*. Defaults to *Value* =  $1.0 \times 10^{-2}$ .

**ENDS**

If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for turbulence dissipation falls below *Value*. Defaults to *Value* =  $1.0 \times 10^{-2}$ .

**NTVF**

Value controls the time stepping strategy for Volume of Fluid (VOF) Advection. The value gives the number of time steps used in the VOF Advection for each time step in the FLOTRAN solution. Defaults to 1.

**METH**

Value specifies the time integration method. Allowable choices for *Value* are:

**BACK**

Backward method (default).

**NEWM**

Newmark time integration method.

**DELT**

Newmark parameter to control integration accuracy and stability. It must be  $\geq 0.5$ . Defaults to 0.5 (2nd order accuracy). If it is larger than 0.5, numerical damping is introduced and the calculation is more stable. The recommended maximum value is 0.6.

**Value**

Value as described for *Label* above.

**Notes**

Repeat command to set each *Label* as required.

All specified criteria must be met before the time step is terminated.

If a termination criterion for a specific label is set negative, the termination check is ignored for that particular DOF.

See the **FLDATA4A,STEP** command to specify output controls based on the number of time steps.

See also the **FLDATA1,SOLU**, **FLDATA2,ITER**, **FLDATA3,TERM**, **FLDATA5,OUTP**, and **FLDATA6,CONV** commands for other Solution and Output Controls.

This command is accessible in the menu if **FLDATA1,SOLU,TRAN,TRUE** has been issued.

For ramped loading (**FLDATA4,TIME,BC,1**), when a load is applied for the first time, it is interpolated from zero to the value of the current load step, and not from the initial condition or value of the DOF from the previous load step.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Transient Ctrl>Time Integration Meth**  
**Main Menu>Preprocessor>FLOTRAN Set Up>VOF Environment>Time Stepping**  
**Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl**  
**Main Menu>Solution>FLOTRAN Set Up>Transient Ctrl>Time Integration Meth**  
**Main Menu>Solution>FLOTRAN Set Up>VOF Environment>Time Stepping**

**FLDATA4A, STEP, Label, Value**

Sets controls for transient analyses based on the number of time steps.

[PREP7: FLOTRAN Options](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <>

**STEP**

Enter the word STEP in this field.

**FLDATA4A,STEP** is the **FLDATA** command with its first argument set to STEP. It can be entered into the program as either **FLDATA4A,STEP,Label,Value** or **FLDATA,STEP,Label,Value** where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Type of transient analysis control (dictates the meaning of *Value*):

**APPE**

*Value* is the results output frequency based on the number of time steps. Every *Value* time steps (substeps), results are written to the `Jobname.RFL` file. If you are creating domain files (`job-name.pv_0000n`) for use by ICEM CFD's PV3 postprocessing visualization tool, this sets the frequency with which domain files are written. See **FLDATA5** for more details. Defaults to *Value* = 10.

**SUMF**

*Value* is the output summary frequency based on the number of time steps. Every *Value* time steps (substeps), a results summary is written to the `Jobname.PFL` file.

**Note**

A results summary will be produced automatically when the results of a time step are written to the `Jobname.RFL` file. Defaults to *Value* = 10.

**OVER**

*Value* is the number of time steps between overwrites of the temporary set of results in the results file (`Jobname.RFL`). This feature enables the user to retain the most up-to-date set of results without having the results file continue to grow in size. Defaults to *Value* = 0 (no overwrites will be made).

**Notes**

Repeat command to set each *Label* as required.

See the **FLDATA4**,`TIME` command to specify output controls based on transient time.

See also the **FLDATA1**,`SOLU`, **FLDATA2**,`ITER`, **FLDATA3**,`TERM`, **FLDATA5**,`OUTP`, and **FLDATA6**,`CONV` commands for other solution and output controls.

This command is accessible in the menu if **FLDATA1**,`SOLU`,`TRAN`,`TRUE` has been issued.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl

Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl

**FLDATA5**, `OUTP`, *Label*, *Value*

Sets output and storage controls.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**OUTP**

Enter the word `OUTP` in this field.

**FLDATA5,OUTP** is the **FLDATA** command with its first argument set to **OUTP**. It can be entered into the program as either **FLDATA5,OUTP,Label,Value** or **FLDATA,OUTP,Label,Value** where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Output and storage controls (dictates the meaning of *Value*):

**SUMF**

*Value* is the number of global iterations between output of results summaries. The results summary, output to the file `Jobname.PFL`, contains a tabulation of the maximum, minimum and average values of each degree of freedom. It also contains the flow rates, pressures, and temperatures at all the inlets and outlets. Defaults to *Value* = 10.

**DEBG**

*Value* sets the debug file printout level. The debug file `Jobname.DBG` contains information on the behavior of the semi-direct solvers used in the solution of the equations. Allowable choices for *Value* are (defaults to 1):

**0**

No information.

**1**

Initial and final information for each global iteration.

**2**

Complete information for each global iteration.

**3**

Produces printout of global coefficient matrices (not recommended, since the files will become very large).

**4**

Produces printout of matrices and additional values at every global iteration (not recommended, because files are usually very large).

**RESI**

*Value* is a flag controlling whether or not the nodal residual file `Jobname.RDF` is written. The nodal residual file contains the nodal residuals corresponding to the current solution (n-1st global iteration), and the new coefficient matrices and forcing function (nth global iteration). It indicates whether or not the solution is oscillating between global iterations on a nodal basis. The file produced is a text file (`Jobname.RDF`) that is read with the **FLREAD** command and that contains the residuals for the degrees of freedom (VX, VY, VZ, PRES, ENKE, ENDS and TEMP) for each node. Allowable values for *Value* are T (writes the file) or F (does not write the file). Defaults to *Value* = F.

**DENS**

*Value* controls the storage of the laminar density. This and the following labels are provided to control the size of the `Jobname.RFL` file (see notes below). The choices are *Value* = T (stores this component) or *Value* = F (does not store this component). Defaults to *Value* = T.

**VISC**

*Value* controls the storage of the laminar viscosity. Defaults to *Value* = T.

**COND**

*Value* controls the storage of the laminar conductivity. Defaults to *Value* = T.

**EVIS**

*Value* controls the storage of the effective viscosity. Defaults to *Value* = T.



**ECON**

*Value* controls the storage of the effective conductivity. Defaults to *Value* = T.

**TTOT**

*Value* controls the storage of the total temperature. Defaults to *Value* = T.

**HFLU**

*Value* controls the storage of the heat flux. Defaults to *Value* = T.

**HFLM**

*Value* controls the storage of the heat transfer (film) coefficient. Defaults to *Value* = T.

**SPHT**

*Value* controls the storage of the specific heat. Defaults to *Value* = F.

**STRM**

*Value* controls the storage of the stream function (2-D). Defaults to *Value* = T.

**MACH**

*Value* controls the storage of the Mach number. Defaults to *Value* = T.

**PTOT**

*Value* controls the storage of the total (stagnation) pressure. Defaults to *Value* = T.

**PCOE**

*Value* controls the storage of the pressure coefficient. Defaults to *Value* = T.

**YPLU**

*Value* controls the storage of the Y+ turbulence quantity. Defaults to *Value* = F.

**TAUW**

*Value* controls the storage of the shear stress at the wall. Defaults to *Value* = F.

**SFTS**

*Value* controls the storage of the surface tension coefficient. Defaults to *Value* = F.

**RDFL**

*Value* controls the storage of the radiation heat flux. Defaults to *Value* = F.

**LMD<sub>*n*</sub>**

Laminar mass diffusion coefficient for species *n*, where *n* = 1 to 6.

**EMD<sub>*n*</sub>**

Effective mass diffusion coefficient for species *n*, where *n* = 1 to 6.

**DOMA**

*Value* controls whether domain files are written. For transient analysis, if *Value* = T, domain files are written at the time frequency specified by the **FLDATA4** command or the step frequency set by the **FLDATA4A** command. For steady state analysis, the files are written as specified by the **FLDATA2** command. Files are named `jobname.pv_00001`, `jobname.pv_00002`, etc., corresponding to the first time increment, second time increment, and so forth. Result set numbers (First, Last, Next, Max, etc.) are preserved to ensure that any domain files created during a previous analysis are either deleted or preserved appropriately.

Setting *Value* = F turns domain file writing off (the default state).

**DRAD**

*VALUE* controls debug level of convergence monitor for surface-to-surface radiation method. Allowable choices for value are (defaults to 0):

- 0**  
No information (default).
- 1**  
Final convergence information.
- 2**  
Complete information for each global iteration.

**Value**

Value as described above.

## Notes

Repeat command to set each *Label* as required.

SUMF, DEBUG, and RESI control output to files other than the nodal results file (*Jobname.RFL*). DOMA controls output to domain files (*Jobname.pv0000n*), which are for use with the ICFM CFD PV3 postprocessing visualization tool.

The remaining *Label* labels are provided in the event the user needs to control the size of the *Jobname.RFL* file. This may be the case when the results of many load steps or time steps are being stored for large models. The choices are T (stores this component) or F (does not store this component). Since the controls should only be set to F if the file size is a problem, most default values are T, and values of listed parameters are stored for every node.

FLOTRAN only allows non-storage of variables which it can calculate from some other means. If the value is not stored, it is replaced by calculations identical to those of the normal solution algorithm. The difference is that the new properties calculated in the absence of storage have not been relaxed as those in storage would have been. See [FLDATA25,RELX](#) for a description of relaxation.

Laminar properties such as density, viscosity, and thermal conductivity are either constant or a function of temperature. In the former case, the constant value can be obtained from the input. In the case of temperature dependent properties, the correct values are obtained from the temperature field (always stored if the temperature equation solution was activated [[FLDATA1,SOLU,TEMP,TRUE](#)]). The newly calculated value is stored as calculated at the initialization of the load step, whereas the value obtained from storage would have undergone relaxation.

If the effective viscosity and effective thermal conductivity are not stored, upon restart they will be calculated from the existing field of turbulent kinetic energy, turbulent kinetic energy dissipation rate, and the density. Again, this is similar to the usual calculation.

See also the [FLDATA1,SOLU](#), [FLDATA2,ITER](#), [FLDATA3,TERM](#), [FLDATA4,TIME](#), [FLDATA4A,STEP](#), and [FLDATA6,CONV](#) commands for other Solution and Output Controls.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>Print Controls**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>Residual File**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>RFL Out Derived**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>RFL Prop Based**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl**

**Main Menu>Solution>FLOTRAN Set Up>Additional Out>Print Controls**  
**Main Menu>Solution>FLOTRAN Set Up>Additional Out>Residual File**  
**Main Menu>Solution>FLOTRAN Set Up>Additional Out>RFL Out Derived**  
**Main Menu>Solution>FLOTRAN Set Up>Additional Out>RFL Prop Based**  
**Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl**

**FLDATA6**, CONV, *Label*, *Value*

**Controls the output of the convergence monitor.**

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

#### CONV

Enter the word CONV in this field.

**FLDATA6**,CONV is the **FLDATA** command with its first argument set to CONV. It can be entered into the program as either **FLDATA6**,CONV,*Label*,*Value* or **FLDATA**,CONV,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

#### *Label*

Format and frequency controls (dictates the meaning of *Value*):

#### OUTP

*Value* controls which display format is used. *Value* can be LAND, BLOC, or BNOW, as described below (defaults to BNOW).

#### ITER

If OUTP *Value* is LAND or BLOC, *Value* is the number of iterations for which the convergence monitor information is stored in memory before being printed. Defaults to *Value* = 1.

#### *Value*

Value as described for *Label* above:

#### LAND

All DOF are output in landscape mode (valid only if *Label* = OUTP).

#### BLOC

All DOF are output in block mode (valid only if *Label* = OUTP).

#### BNOW

All DOF are presented in block format immediately as iterations are completed (valid only if *Label* = OUTP).

#### *n*

Number of iterations (valid only if *Label* = ITER).

## Notes

The convergence monitor output will appear on the screen during an interactive FLOTRAN analysis as well as in the printed output. OUTP controls what display format is used, and ITER controls how many iterations are stored and displayed.

See also the **FLDATA1**,SOLU, **FLDATA2**,ITER, **FLDATA3**,TERM, **FLDATA4**,TIME, **FLDATA4A**,STEP, and **FLDATA5**,OUTP commands for other Solution and Output Controls.

The choices for OUTP are LANDscape mode, BLOCk mode, or the "block now" mode, BNOW. If OUTP is set to BNOW, convergence monitor information is not stored in the memory, but is presented immediately as it is calculated. If OUTP is set to either LAND or BLOC, information is first stored for  $n$  (ITER) iterations before being printed.

In the following illustrations, "x" denotes convergence monitor numbers greater than zero:

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Landscape Mode

Iter<	1	2	3	4	5	6
VX	x	x	x	x	x	x
VY	x	x	x	x	x	x
VZ	x	x	x	x	x	x
PRES	x	x	x	x	x	x
ENKE	x	x	x	x	x	x
ENDS	x	x	x	x	x	x
TEMP	x	x	x	x	x	x

### Block Mode

Iter ^	VX	VY	VZ	PRES	ENKE	ENDS	TEMP
1	x	x	x	x	x	x	X
2	x	x	x	x	x	x	X
3	x	x	x	x	x	x	X
4	x	x	x	x	x	x	X
5	x	x	x	x	x	x	X
6	x	x	x	x	x	x	X

### Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>Print Controls**  
**Main Menu>Solution>FLOTRAN Set Up>Additional Out>Print Controls**

**FLDATA7**, PROT, *Label*, *Value*

Specifies the type of fluid property.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

#### PROT

Enter the word PROT in this field.

**FLDATA7**,PROT is the **FLDATA** command with its first argument set to PROT. It can be entered into the program as either **FLDATA7**,PROT,*Label*,*Value* or **FLDATA**,PROT,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

#### *Label*

Fluid property being typed:

#### DENS

Density (*Value* defaults to CONSTANT).

**VISC**

Viscosity (*Value* defaults to CONSTANT).

**COND**

Thermal conductivity (*Value* defaults to CONSTANT).

**SPHT**

Specific heat (*Value* defaults to CONSTANT).

**SFTS**

Surface tension coefficient (*Value* defaults to CONSTANT).

**WSCA**

Wall static contact angle. (*Value* defaults to CONSTANT).

**Value**

Fluid property type.

You can enter one of the values shown below, or if your fluid properties can be expressed as a table, you can enter a table name. To enter a table name, you must first define a TABLE type array parameter using the **\*DIM** command. Note that you must enclose the table name in % symbols in the **FLDATA7** command line (e.g., **FLDATA7,PROT,DENS,%tablename%**). For more information on defining tables, see [TABLE Type Array Parameters](#) in the *ANSYS Parametric Design Language Guide*.

**CONSTANT**

Constant properties. The **FLDATA8,NOMI** command must be used to specify nominal properties.

**GAS**

Gas properties. The **FLDATA8,NOMI** command must be used to specify nominal properties. **FLDATA9,COF1**, **FLDATA10,COF2**, and possibly **FLDATA11,COF3**, must also be used.

**LIQUID**

Liquid properties. The **FLDATA8,NOMI** command must be used to specify nominal properties. **FLDATA9,COF1**, **FLDATA10,COF2**, and possibly **FLDATA11,COF3** must be used.

**TABLE**

Indicates that a table of property values and corresponding temperature values are input using the **MPDATA** and **MPTEMP** commands. You cannot use this option with the %table% method described above.

**POWL**

This choice for viscosity type activates the Power Law viscosity model, which is non-Newtonian. (For a description of the model see [Viscosity](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*. The *Fluids Analysis Guide* explains how to use the model.) The Power Law model requires you to specify four coefficients via the **FLDATA8,NOMI,VISC** command, the **FLDATA9,COF1,VISC** command, the **FLDATA10,COF2,VISC** command, and the **FLDATA11,COF3,VISC** command.

**CARR**

This choice for viscosity type activates the Carreau viscosity model, which is non-Newtonian. (For a description of this model, see [Viscosity](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*. The *Fluids Analysis Guide* explains how to use the non-Newtonian viscosity models.) The Carreau model requires you to specify four coefficients via the **FLDATA8,NOMI,VISC** command, the **FLDATA9,COF1,VISC** command, the **FLDATA10,COF2,VISC** command and the **FLDATA11,COF3,VISC** command.

**BING**

This choice for viscosity type activates the Bingham viscosity model, which is non-Newtonian. (For a description of this model, see *Viscosity* in the *Theory Reference for the Mechanical APDL and Mechanical Applications*. The *Fluids Analysis Guide* explains how to use the non-Newtonian viscosity models.) The Bingham model requires you to specify three coefficients, using the **FLDATA8**,NOMI,VISC command, the **FLDATA9**,COF1,VISC command, and the **FLDATA10**,COF2,VISC command.

**USRV**

This choice for viscosity activates the user-programmable subroutine, UserVisLaw. In this routine, you can define your own constitutive relationship between viscosity and other variables such as position, time, temperature, pressure, velocity, velocity gradients, etc. For details, see *Viscosity* in the *Theory Reference for the Mechanical APDL and Mechanical Applications*, the *Fluids Analysis Guide*, and the *Guide to ANSYS User Programmable Features*.

The UserVisLaw subroutine uses the four coefficients you specify via the **FLDATA8**,NOMI,VISC command, the **FLDATA9**,COF1,VISC command, the **FLDATA10**,COF2,VISC command, and the **FLDATA11**,COF3,VISC command.

**AIR**

Air properties in units of meter-kg-sec.

**AIR\_B**

Air properties in units of meter-kg-sec, and the pressure is set to the reference pressure for the evaluation of density.

**AIR-SI**

Air properties in units of meter-kg-sec.

**AIR-SI\_B**

Air properties in units of meter-kg-sec, and the pressure is set to the reference pressure for the evaluation of density.

**AIR-CM**

Air properties in units of cm-g-sec.

**AIR-CM\_B**

Air properties in units of cm-g-sec, and the pressure is set to the reference pressure for the evaluation of density.

**AIR-MM**

Air properties in units of mm-g-sec.

**AIR-MM\_B**

Air properties in units of mm-g-sec, and the pressure is set to the reference pressure for the evaluation of density.

**AIR-FT**

Air properties in units of ft-slug-sec.

**AIR-FT\_B**

Air properties in units of ft-slug-sec, and the pressure is set to the reference pressure for the evaluation of density.

**AIR-IN**

Air properties in units of in-(lbf-s\*\*2/in)-sec (results in units of psi for pressure).

**AIR-IN\_B**

Air properties in units of in-(lbf-s\*\*2/in)-sec (results in units of psi for pressure), and the pressure is set to the reference pressure for the evaluation of density.

**CMIX**

The property is the mass fraction average of the individual species property. You can use this option only if the species are defined.

**USER**

Use one of the following user-programmable subroutines to define the property: UserDens to define density, UserVisLaw to define viscosity, UserSpht to define specific heat, UserCond to define conductivity, or UserSfts to define surface tension.

**CGAS**

Available only for density. The density is calculated using the ideal gas law, with the molecular weight calculated from the mass fraction average of the individual species molecular weights. You can only use this option if species are defined.

**Notes**

This command is used to specify the fluid property type (constant, gas, liquid, air) for the density, viscosity, and (if required) thermal conductivity and specific heat properties. Repeat the **FLDATA7** command for each property as required.

The choice of fluid property type implies the use of one of the following equations. In all cases the program assumes that consistent units are being used. The value of unity for the gravitational constant  $g_c$  implies a consistent set of units.

If the property type is CONSTANT, then the equations used are as follows:

DENS = NOMI  
 VISC = NOMI  
 COND = NOMI  
 SPHT = NOMI

If the property type is LIQUID, then Sutherland's liquid law is used for the viscosity and conductivity and a second order polynomial as a function of temperature is used for density:

$$\text{DENS} = \text{NOMI} + \text{COF2} * (\text{T} - \text{COF1}) + \text{COF3} * (\text{T} - \text{COF1})^2$$

$$\text{VISC} = \text{NOMI} * \text{EXP}[(\text{COF2} * (1/\text{T} - 1/\text{COF1}) + \text{COF3} * (1/\text{T} - 1/\text{COF1})^2)]$$

$$\text{COND} = \text{NOMI} * \text{EXP}[(\text{COF2} * (1/\text{T} - 1/\text{COF1}) + \text{COF3} * (1/\text{T} - 1/\text{COF1})^2)]$$

If the type is GAS, Sutherland's law for gases is used for conductivity and viscosity and the ideal gas law is used for the density:

$$\text{DENS} = \text{NOMI} * (\text{P}/\text{COF2}) / (\text{T}/\text{COF1})$$

$$\text{VISC} = \text{NOMI} * (\text{T}/\text{COF1})^{1.5} * (\text{COF1} + \text{COF2}) / (\text{T} + \text{COF2})$$

$$\text{COND} = \text{NOMI} * (\text{T}/\text{COF1})^{1.5} * (\text{COF1} + \text{COF2}) / (\text{T} + \text{COF2})$$

For *Label* = SFTS, CONSTANT, LIQUID, and USER are the only valid property types. If the property type is CONSTANT, the equation is:

$$\text{SFTS} = \text{NOMI}$$

If the property type is LIQUID, the equation is:

$$\text{SFTS} = \text{NOMI} + \text{COF2} * (\text{T} - \text{COF1}) + \text{COF3} * (\text{T} - \text{COF1})^2$$

For *Label* = WSCA, CONSTANT is the only valid property type and the equation is:

$$\text{WSCA} = \text{NOMI}$$

In each case, the value of NOMI is input with the **FLDATA8**,NOMI command, COF1 with **FLDATA9**,COF1, COF2 with **FLDATA10**,COF2, and COF3 with **FLDATA11**,COF3. No defaults are assumed.

For types other than GAS, LIQUID, CONSTANT, AIR or AIR\_B, the values of constants in the evaluation of properties will be obtained from the file FLOPRP.ANS, where the data for AIR resides. See the *Fluids Analysis Guide* for information on how to put data into the FLOPRP.ANS file.

See also the **FLDATA8**,NOMI, **FLDATA9**,COF1, **FLDATA10**,COF2, **FLDATA11**,COF3, **FLDATA12**,PROP, and **FLDATA13**,VARY commands for other Fluid Property Definition commands.

This command is accessed in the menu as **FLDATA12**,PROP.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties**

**Main Menu>Preprocessor>FLOTRAN Set Up>Table Props**

**Main Menu>Solution>FLOTRAN Set Up>Fluid Properties**

**Main Menu>Solution>FLOTRAN Set Up>Table Props**

---

## FLDATA8, NOMI, *Label*, *Value*

**Specifies the NOMI coefficient of the fluid property equation.**

[PREP7: FLOTRAN Property](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### NOMI

Enter the word NOMI in this field.

**FLDATA8**,NOMI is the **FLDATA** command with its first argument set to NOMI. It can be entered into the program as either **FLDATA8**,NOMI,*Label*,*Value* or **FLDATA**,NOMI,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### *Label*

Fluid property NOMI is being defined for:

#### **DENS**

Density.

#### **VISC**

Viscosity.

#### **COND**

Thermal conductivity.



**SPHT**

Specific heat.

**SFTS**

Surface tension coefficient.

**WSCA**

Wall static contact angle (*Value* defaults to 90).

**Value**

Enter the value for NOMI.

**Notes**

Specifies the value NOMI as described on the **FLDATA7,PROT** command. NOMI is the constant fluid property value, or is the value of the property at the specified temperature COF1 (or simply the value of a coefficient). NOMI is only valid for property types GAS, LIQUID, and CONSTANT [**FLDATA7,PROT**]. If the property is a gas or liquid, **FLDATA9,COF1**, **FLDATA10,COF2**, and possibly **FLDATA11,COF3**, must also be used.

If the label is VISC, the **FLDATA7,PROT** command you enter determines what NOMI is. For example, if you issue the **FLDATA7,PROT,VISC,POWL** command, NOMI is the nominal viscosity for the Power Law model. If you issue the **FLDATA7,PROT,VISC,BING** command, the NOMI is the plastic viscosity for the Bingham model. If you issue the **FLDATA7,PROT,VISC,CARR** command, the NOMI is the zero shear rate viscosity for the Carreau model. If you issue the **FLDATA7,PROT,USRV** command, the NOMI is a coefficient available in UserVisLaw, the user-programmable subroutine for viscosity models.

See also the **FLDATA12,PROP** and **FLDATA13,VARY** commands for other Fluid Property Definition commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties

Main Menu>Solution>FLOTRAN Set Up>Fluid Properties

**FLDATA9,COF1,Label,Value**

Specifies the COF1 coefficient of the fluid property equation.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**COF1**

Enter the word COF1 in this field.

**FLDATA9,COF1** is the **FLDATA** command with its first argument set to COF1. It can be entered into the program as either **FLDATA9,COF1,Label,Value** or **FLDATA,COF1,Label,Value** where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Fluid property being described:

**DENS**

Density.

**VISC**

Viscosity.

**COND**

Thermal conductivity.

**SPHT**

Specific heat.

**SFTS**

Surface tension coefficient.

**Value**

Absolute temperature at which the *Label* property has the value NOMI (defaults to 0.0) or simply a coefficient.

## Notes

Specifies the value of absolute temperature associated with the property value set be **FLDATA8,NOMI**, as described on the **FLDATA7,PROT** command, or simply a coefficient. Setting the value of COF1 to zero results in a constant property for liquids or gas. If the property is a gas or liquid, **FLDATA8,NOMI**, **FLDATA10,COF2**, and possibly **FLDATA11,COF3**, must also be used.

If the label is VISC and you issue the **FLDATA7,PROT,VISC,POWL** command, the COF1 value is the cutoff shear rate for the Power Law Model. If you issue the **FLDATA7,PROT,VISC,BING** command, COF1 is the plastic stress value for the Bingham model. If you issue the **FLDATA7,PROT,VISC,CARR** command, the COF1 value is the infinite shear rate viscosity for the Carreau model. If you issue the **FLDATA7,PROT,VISC,USRV** command, COF1 is a coefficient available in the UserVisLaw subroutine.

If the label is SPHT, the COF1 value is available in the user-programmable subroutine UserSpht.

See also the **FLDATA12,PROP**, and **FLDATA13,VARY** commands for other Fluid Property Definition commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties

Main Menu>Solution>FLOTRAN Set Up>Fluid Properties

---

### **FLDATA10**, COF2, Label, Value

Specifies the COF2 coefficient of the fluid property equation.

[PREP7: FLOTRAN Property](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**COF2**

Enter the word COF2 in this field.

**FLDATA10,COF2** is the **FLDATA** command with its first argument set to COF2. It can be entered into the program as either **FLDATA10,COF2,Label,Value** or **FLDATA,COF2,Label,Value** where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Fluid property being described:

**DENS**

Density.

**VISC**

Viscosity.

**COND**

Thermal conductivity.

**SPHT**

Specific heat.

**SFTS**

Surface tension coefficient.

**Value**

Coefficient COF2 (defaults to 0.0).

**Notes**

Specifies the value COF2 as described on the **FLDATA7,PROT** command.

If the label is VISC and you issue the **FLDATA7,PROT,VISC,POWL** command, the COF2 value is the consistency coefficient for the Power Law Model. If you issue the **FLDATA7,PROT,VISC,BING** command, COF2 is the Newtonian viscosity for the Bingham model. If you issue the **FLDATA7,PROT,VISC,CARR** command, the COF2 value is the time constant for the Carreau model. If you issue the **FLDATA7,PROT,VISC,USRV** command, COF2 is a coefficient available in the UserVisLaw subroutine.

For viscosity, liquid, gas, or thermal conductivity, COF2 is a Sutherland's law constant. For liquid density, it is a polynomial coefficient. For gas density, it is the pressure in the nominal ideal gas law constant evaluation. If the property is a gas or liquid, **FLDATA8,NOMI**, **FLDATA9,COF1**, and possibly **FLDATA11,COF3**, must also be used.

See also the **FLDATA12,PROP**, and **FLDATA13,VARY** commands for other Fluid Property Definition commands.

If the label is SPHT, the COF2 value is available in the user-programmable subroutine UserSpht.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties**

**Main Menu>Solution>FLOTRAN Set Up>Fluid Properties**

**FLDATA11**, COF3, *Label*, *Value*

Specifies the COF3 coefficient of the fluid property equation.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**COF3**

Enter the word COF3 in this field.

**FLDATA11**,COF3 is the **FLDATA** command with its first argument set to COF3. It can be entered into the program as either **FLDATA11**,COF3,*Label*,*Value* or **FLDATA**,COF3,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Fluid property being described:

**DENS**

Density.

**VISC**

Viscosity.

**COND**

Thermal conductivity.

**SPHT**

Specific heat.

**SFTS**

Surface tension coefficient.

**Value**

Coefficient COF3 (defaults to 0.0).

**Notes**

Specifies the value COF3 as described on the **FLDATA7**,PROT command.

If the label is VISC and you issue the **FLDATA7**,PROT,VISC,POWL command, the COF3 value is the power for the Power Law Model. COF3 is not used in the Bingham model. If you issue the **FLDATA7**,PROT,VISC,CARR command, the COF3 value is the power for the Carreau model. If you issue the **FLDATA7**,PROT,VISC,USRV command, COF3 is a coefficient available in the UserVisLaw subroutine.

For the viscosity or thermal conductivity of a liquid, COF3 is a Sutherland's law constant. For the density of a liquid, it is a polynomial coefficient. It is not used for gases. **FLDATA8**,NOMI, **FLDATA9**,COF1, and **FLDATA10**,COF2, must also be used.

If the label is SPHT, the COF3 value is available in the user-programmable subroutine UserSpht.

See also the **FLDATA12**,PROP, and **FLDATA13**,VARY commands for other Fluid Property Definition commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties**

---

**Main Menu>Solution>FLOTRAN Set Up>Fluid Properties**


---

**FLDATA12**, PROP, *Label*, *Value*

Sets the property update frequency flag.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**PROP**

Enter the word PROP in this field.

**FLDATA12**,PROP is the **FLDATA** command with its first argument set to PROP. It can be entered into the program as either **FLDATA12**,PROP,*Label*,*Value* or **FLDATA**,PROP,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Updating label (dictates the meaning of *Value*):

**IVIS**

*Value* is the initial guess for viscosity. If you do not specify any value, the nominal viscosity issued via the **FLDATA8**,NOMI,VISC command is the initial viscosity. For information on using this label, see the *Fluids Analysis Guide*.

**UFRQ**

*Value* is the number of global iterations (frequency of updating) between the update of the properties.

**Value**

Number of iterations as described above (defaults to 1).

**Notes**

Properties will not be updated if the all the property types are set to CONSTANT with the **FLDATA7** command.

See also the **FLDATA7**,PROT, **FLDATA8**,NOMI, **FLDATA9**,COF1, **FLDATA10**,COF2, **FLDATA11**,COF3, and **FLDATA13**,VARY commands for other Fluid Property Definition commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties**

**Main Menu>Solution>FLOTRAN Set Up>Fluid Properties**

---

**FLDATA13**, VARY, *Label*, *Value*

Sets the property variation flag.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**VARY**

Enter the word VARY in this field.

**FLDATA13**,VARY is the **FLDATA** command with its first argument set to VARY. It can be entered into the program as either **FLDATA13**,VARY,*Label*,*Value* or **FLDATA**,VARY,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Fluid property being described:

**DENS**

Density.

**VISC**

Viscosity.

**COND**

Thermal conductivity.

**SPHT**

Specific heat.

**SFTS**

Surface tension coefficient.

**Value**

Flag value for property variation:

**T**

To turn property variation on.

**F**

To turn property variation off (default).

## Notes

For nonconstant fluid properties [**FLDATA7**,PROT], the appropriate flags must be set to T to allow property variation between global iterations.

See also the **FLDATA7**,PROT, **FLDATA8**,NOMI, **FLDATA9**,COF1, **FLDATA10**,COF2, **FLDATA11**,COF3, and **FLDATA12**,PROP commands for other Fluid Property Definition commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties**

**Main Menu>Solution>FLOTRAN Set Up>Fluid Properties**

---

**FLDATA14**, TEMP, Label, Value

Specifies the reference temperature.

PREP7: FLOTRAN Operating

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**TEMP**

Enter the word TEMP in this field.

**FLDATA14**,TEMP is the **FLDATA** command with its first argument set to TEMP. It can be entered into the program as either **FLDATA14**,TEMP,Label,Value or **FLDATA**,TEMP,Label,Value where Label and Value are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Type of temperature specification (dictates the meaning of Value):

**NOMI**

Value is the initial temperature if a boundary condition or transient initial condition has not been set. Value defaults to 293.0.

**BULK**

Value is the temperature used to evaluate heat transfer coefficients given a heat flux or temperature at a boundary. Value defaults to 293.0.

**TTOT**

Value is the total (stagnation) temperature used in compressible adiabatic flow. The static temperature is calculated from the kinetic energy evaluated in terms of the velocity magnitude, specific heat, and gravitational constant:

$$T_{\text{static}} = T_{\text{total}} - \frac{V^2}{2g_c C_p}$$

**Value**

Temperature as described above.

**Notes**

See also the **FLDATA15**,PRES, **FLDATA16**,BULK, and **FLDATA17**,GAMM commands for other Operating Condition commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Ref Conditions

Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Ref Conditions

**FLDATA15**, PRES, *Label*, *Value*

Specifies the reference pressure.

PREP7: FLOTRAN Operating

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**PRES**

Enter the word PRES in this field.

**FLDATA15**,PRES is the **FLDATA** command with its first argument set to PRES. It can be entered into the program as either **FLDATA15**,PRES,*Label*,*Value* or **FLDATA**,PRES,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Reference label (dictates the meaning of *Value*):

**REFE**

*Value* is the reference pressure. The absolute pressure results from adding the pressure components from rotating terms, the static pressure head, the FLOTRAN pressure, and the reference pressure.

*Value* defaults to  $1.0135 \times 10^{+5}$ .

**Value**

Reference pressure as described above.

**Notes**

See also the **FLDATA14**,TEMP, **FLDATA16**,BULK, and **FLDATA17**,GAMM commands for other Operating Condition commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Ref Conditions  
Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Ref Conditions

---

**FLDATA16**, BULK, *Label*, *Value*

Specifies the bulk modulus parameter.

PREP7: FLOTRAN Operating

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**BULK**

Enter the word BULK in this field.

**FLDATA16**,BULK is the **FLDATA** command with its first argument set to BULK. It can be entered into the program as either **FLDATA16**,BULK,*Label*,*Value* or **FLDATA**,BULK,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Label (dictates the meaning of *Value*):



**BETA**

*Value* is the bulk modulus parameter.

**Value**

Value of bulk modulus parameter (defaults to  $10^{15}$ ).

**Notes**

The bulk modulus parameter,  $\beta_p$ , is used in the transient algorithm for incompressible flows:

$$\beta_p = \frac{\partial p}{\partial \rho}$$

The default value corresponds to that of a constant density fluid.

See also the [FLDATA14](#),TEMP, [FLDATA15](#),PRES, and [FLDATA17](#),GAMM commands for other Operating Condition commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Ref Conditions**  
**Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Ref Conditions**

**FLDATA17**, GAMM, *Label*, *Value*

**Specifies the specific heat ratio.**

PREP7: FLOTRAN Operating

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**GAMM**

Enter the word GAMM in this field.

**FLDATA17**,GAMM is the **FLDATA** command with its first argument set to GAMM. It can be entered into the program as either **FLDATA17**,GAMM,*Label*,*Value* or **FLDATA**,GAMM,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Label (dictates the meaning of *Value*):

**COMP**

*Value* is the ratio of specific heats.

**Value**

Value of the ratio (defaults to 1.4).

**Notes**

Specifies the ratio of specific heat at constant pressure to the specific heat at constant volume, Cp/Cv. It is used in compressible analyses.

See also the [FLDATA14](#),TEMP, [FLDATA15](#),PRES, and [FLDATA16](#),BULK commands for other Operating Condition commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Ref Conditions**  
**Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Ref Conditions**

**FLDATA18**, METH, *Label*, *Value*

Selects the algebraic solver.

[PREP7: FLOTRAN Solver](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### METH

Enter the word METH in this field.

**FLDATA18**,METH is the **FLDATA** command with its first argument set to METH. It can be entered into the program as either **FLDATA18**,METH,*Label*,*Value* or **FLDATA**,METH,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### *Label*

Degree of freedom set for which solver is being specified:

#### **PRES**

Pressure equation.

#### **TEMP**

Energy equation.

#### **VX**

U velocity.

#### **VY**

V velocity.

#### **VZ**

W velocity.

#### **ENKE**

Turbulent kinetic energy.

#### **ENDS**

Turbulent kinetic energy dissipation rate.

### *Value*

Solver to be used for the degree of freedom set:

**0**

Do not solve the equation set.

**1**

Use the Tri-Diagonal Matrix Algorithm (TDMA) (default for all DOF sets except PRES and TEMP). Also see the [FLDATA19](#),TDMA command.

- 2 Use the Conjugate Residual method. See also the [FLDATA20,SRCH](#), [FLDATA21,CONV](#), [FLDATA22,MAXI](#), and [FLDATA23,DELT](#) commands.
- 3 Use the Preconditioned Conjugate Residual method (default for PRES DOF set). See also the [FLDATA20,SRCH](#), [FLDATA21,CONV](#), [FLDATA22,MAXI](#), and [FLDATA23,DELT](#) commands.
- 4 Use the Preconditioned Generalized Minimum Residual (PGMR) solution method (default for TEMP DOF set). See [FLOTRAN Thermal Analyses](#) in the *Fluids Analysis Guide* for more information on conjugate heat transfer.
- 5 Use the Sparse Direct method.
- 6 Use the Preconditioned BiCGStab method (PBCGM).

## Notes

A solver can be specified for each degree of freedom set. Repeat the **FLDATA18** command as needed.

The Tri-Diagonal Matrix Algorithm (TDMA) is a special case of the standard Gauss-Seidel iterative method for the solution of sets of algebraic equations. It is the preferred method for providing approximate solutions for the momentum and turbulence equations since exact solutions are not required. A convergence criterion is not specified for the TDMA method, merely the number of iterations (sweeps) to be performed [[FLDATA19,TDMA](#)].

Three methods are semi-direct solution methods based on search directions. The Conjugate Residual method requires the least memory, but stalls when solving ill-conditioned problems. (In an ill-conditioned problem, the thermal properties of fluid and non-fluid materials are different by several orders of magnitude). The Preconditioned Conjugate Residual method requires much more memory but performs better for ill-conditioned matrix problems which can arise when you are solving conjugate heat transfer problems. The Preconditioned Generalized Minimum Residual (PGMR) method is memory-intensive; by necessity, it incorporates a tight convergence criterion. The PGMR method is recommended for solving the energy equation for ill-conditioned conjugate transfer problems. In general, the Preconditioned BiCGStab method (PBCGM) requires less memory than the PGMR method. It is also recommended for extremely ill-conditioned conjugate heat transfer problems.

The Sparse Direct method is based on Gaussian elimination to factorize the matrix. This method is memory intensive and creates temporary files on the hard disk. It is robust and can be used for symmetric as well as non-symmetric equation systems. The Sparse solver is not recommended for use with *Label* = PRES. If you do use the Sparse solver with *Label* = PRES, you will need to save and then resume your database prior to solving.

For incompressible flow problems that involve the solution for the pressure degree of freedom, if you set *Value* equal to 2 or 3, the Preconditioned Conjugate Gradient method solver is used instead of the Conjugate Residual or Preconditioned Conjugate Residual method solvers.

The [Theory Reference for the Mechanical APDL and Mechanical Applications](#) contains more detail on these methods.

See also the [FLDATA19](#),TDMA, [FLDATA20](#),SRCH, [FLDATA21](#),CONV, [FLDATA22](#),MAXI, and [FLDATA23](#),DELT commands for other Algebraic Solver controls.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**

**FLDATA19**, TDMA, *Label*, *Value*

Specifies the number of TDMA sweeps.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### TDMA

Enter the word TDMA in this field.

**FLDATA19**,TDMA is the **FLDATA** command with its first argument set to TDMA. It can be entered into the program as either **FLDATA19**,TDMA,*Label*,*Value* or **FLDATA**,TDMA,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### *Label*

Degree of freedom set for which the TDMA solver is being used:

#### **PRES**

Pressure equation (*Value* defaults to 100).

#### **TEMP**

Energy equation (*Value* defaults to 100).

#### **VX**

U velocity (*Value* defaults to 1).

#### **VY**

V velocity (*Value* defaults to 1).

#### **VZ**

W velocity (*Value* defaults to 1).

**ENKE**

Turbulent kinetic energy (*Value* defaults to 10).

**ENDS**

Turbulent kinetic energy dissipation rate (*Value* defaults to 10).

**Value**

Number of iterations (sweeps) for this degree of freedom set.

**Notes**

Specifies the number of iterations (sweeps) the Tri-Diagonal Matrix Algorithm [FLDATA18,METH] will perform during the solution. A different number of sweeps may be specified for each degree of freedom set the TDMA solver is used for. Increasing the number for the velocity DOF may cause instability.

See also the FLDATA18,METH command for other Algebraic Solver controls.

This command is accessible in the menu if FLDATA18,METH,Label,1 has been issued for the PRES, TEMP, ENKE, or ENDS labels.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**

**FLDATA20**, SRCH, *Label*, *Value*

Specifies the number of conjugate direction search vectors.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**SRCH**

Enter the word SRCH in this field.

FLDATA20,SRCH is the FLDATA command with its first argument set to SRCH. It can be entered into the program as either FLDATA20,SRCH,Label,Value or FLDATA,SRCH,Label,Value where Label and Value are as described below. See the FLDATA command for other FLOTRAN CFD input choices.

**Label**

Degree of freedom set for which the solver is being used:

**PRES**

Pressure equation.

**TEMP**

Energy equation.

**VX**

U velocity.

**VY**

V velocity.

**VZ**

W velocity.

**ENKE**

Turbulent kinetic energy.

**ENDS**

Turbulent kinetic energy dissipation rate.

**Value**

Number of search directions (defaults to 2). If you are using the PGMR solver, the default is to use 12 search vectors. You cannot use fewer than 12 for the PGMR solver. If you are using the PBCGM solver, the number of search directions is 1 to 8, with 2 as the default.

## Notes

The conjugate direction iterative techniques (methods 2 and 3 on the **FLDATA18**,METH command) develop a solution as a linear combination of search directions. In the solution with methods 2 and 3, new search vectors are made orthogonal to *Value* previous vectors in the solution of the non-symmetric matrix systems. See the *Fluids Analysis Guide* for details on when to change these values.

---

### Note

The parameter is not applicable to the incompressible pressure equation since the new search vector is automatically orthogonal to all the previous ones.

See also the **FLDATA18**,METH,TDMA, **FLDATA21**,CONV, **FLDATA22**,MAXI, and **FLDATA23**,DELT commands for other Algebraic Solver controls.

This command is accessible in the menu if **FLDATA18**,METH,*Label*,(2 or 3) has been issued for the relevant labels.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**

**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**

**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**

**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**

**FLDATA20A**, *PGMR*, *Label*, *Value*

Specifies the amount of fill-in when preconditioning the coefficient matrix.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

#### PGMR

Enter the word PGMR in this field.

**FLDATA20A**, *PGMR* is the **FLDATA** command with its first argument set to PGMR. It can be entered into the program as either **FLDATA20A**, *PGMR*, *Label*, *Value* or **FLDATA**, *PGMR*, *Label*, *Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

#### Label

Label (dictates the meaning of Value):

#### FILL

Value represents the number of extra elements allowed in each row of the L and U decomposition matrices. An extra element is defined as being in addition to the number of nonzero elements in the row of the original matrix. The allowable range for the fill parameter is 1 to 10 (defaults to 6).

#### MODP

*Value* represents the number of global iterations performed using the TDMA method between global iterations performed using the PGMR method for the temperature DOF. The first global iteration always uses the PGMR method. The global iteration count is based on the total number, not the number for a restart. Value defaults to 0 (PGMR always used).

#### Value

Value as described above.

## Notes

The fill parameter specifies the amount of fill-in when constructing the L and U decomposition of the coefficient matrix.

Use of FILL requires selection of the PGMR solver for the PRES or TEMP degree of freedom. Use of MODP requires selection of the PGMR solver for the temperature DOF.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**

## FLDATA20B, PBCGM, Label, Value

Specifies the number of fill-ins for the ILU preconditioner.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### PBCGM

Enter the word PBCGM in this field.

**FLDATA20B,PBCGM** is the **FLDATA** command with its first argument set to PBCGM. It can be entered into the program as either **FLDATA20B,PBCGM,Label,Value** or **FLDATA,PBCGM,Label,Value** where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### Label

Label (dictates the meaning of Value):

#### FILL

Value represents the number of extra elements allowed in each row of the L and U decomposition matrices. An extra element is defined as being in addition to the number of nonzero elements in the row of the original matrix. The allowable range for the fill parameter is 0 to 10 (defaults to 6).

### Value

Value as described above.

## Notes

The fill parameter specifies the amount of fill-in when constructing the L and U decomposition of the coefficient matrix.

A single constant for FILL is used for the PBCGM solver, and it will be applied to all degrees of freedom that use this solver.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**



Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD  
 Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD  
 Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD  
 Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD  
 Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD  
 Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD  
 Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD  
 Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD  
 Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD  
 Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD

**FLDATA21**, CONV, *Label*, *Value*

Specifies the convergence criterion for FLOTRAN algebraic solvers.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**CONV**

Enter the word CONV in this field.

**FLDATA21**, CONV is the **FLDATA** command with its first argument set to CONV. It can be entered into the program as either **FLDATA21**, CONV, *Label*, *Value* or **FLDATA**, CONV, *Label*, *Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Degree of freedom set for which the solver is being used:

**PRES**

Pressure equation (*Value* defaults to  $1.0 \times 10^{-12}$ ).

**TEMP**

Energy equation (*Value* defaults to  $1.0 \times 10^{-12}$ ).

**VX**

U velocity (*Value* defaults to  $1.0 \times 10^{-5}$ ).

**VY**

V velocity (*Value* defaults to  $1.0 \times 10^{-5}$ ).

**VZ**

W velocity (*Value* defaults to  $1.0 \times 10^{-5}$ ).

**ENKE**

Turbulent kinetic energy (*Value* defaults to  $1.0 \times 10^{-5}$ ).

**ENDS**

Turbulent kinetic energy dissipation rate (*Value* defaults to  $1.0 \times 10^{-5}$ ).

**Value**

Convergence criterion factor.

## Notes

The convergence monitor (for methods 2 or 3 on the **FLDATA18**,METH command) represents the factor by which the inner product of the residual vector is to be reduced during the solution of the equations at any global iteration.

If you are using the Preconditioned Generalized Minimum Residual (PGMR) solver, the least restrictive convergence criterion allowed is 1.E-10. If you try to use a less restrictive convergence criterion, FLOTRAN will change it to 1.E-10. A convergence criterion as low as 1.E-20 may be necessary for some problems.

Less restrictive values are specified for the velocities and turbulence parameters because the iterative nature of the segregated solution algorithm in FLOTRAN does not require exact solutions to these equations at any global iteration. The default solution method for these degrees of freedom is the Tri-Diagonal Matrix Algorithm (TDMA) method.

See also the **FLDATA18**,METH, **FLDATA20**,SRCH, **FLDATA22**,MAXI, and **FLDATA23**,DELT commands for other Algebraic Solver controls.

This command is accessible in the menu if **FLDATA18**,METH,*Label*,(2 or 3) has been issued for the relevant labels.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**

**FLDATA22**, MAXI, *Label*, *Value*

**Specifies the maximum number of semi-direct iterations.**

[PREP7: FLOTRAN Solver](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**MAXI**

Enter the word MAXI in this field.

**FLDATA22,MAXI** is the **FLDATA** command with its first argument set to **MAXI**. It can be entered into the program as either **FLDATA22,MAXI,Label,Value** or **FLDATA,MAXI,Label,Value** where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### *Label*

Degree of freedom set for which the solver is being used:

#### **PRES**

Pressure equation (*Value* defaults to 1000).

#### **TEMP**

Energy equation (*Value* defaults to 1000).

#### **VX**

U velocity (*Value* defaults to 100).

#### **VY**

V velocity (*Value* defaults to 100).

#### **VZ**

W velocity (*Value* defaults to 100).

#### **ENKE**

Turbulent kinetic energy (*Value* defaults to 100).

#### **ENDS**

Turbulent kinetic energy dissipation (*Value* defaults to 100).

### *Value*

Limit on the number of iterations.

## Notes

These limits apply to the semi-direct solution methods (method 2 or 3 on the **FLDATA18,METH** command). If this number of iterations is reached before the convergence criterion is met, the solution is accepted, a warning message is printed, and the program continues normally. However if the pressure equation is not solved to the precision desired three times during a load step, execution will terminate.

See also the **FLDATA18,METH**, **FLDATA20,SRCH**, **FLDATA21,CONV**, and **FLDATA23,DELT** commands for other Algebraic Solver controls.

This command is accessible in the menu if **FLDATA18,METH,Label,(2 or 3)** has been issued for the relevant labels.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**

**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**

**FLDATA23**, DELT, *Label*, *Value*

Specifies the solver minimum normalized rate of change.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

## DELT

Enter the word DELT in this field.

**FLDATA23**, DELT is the **FLDATA** command with its first argument set to DELT. It can be entered into the program as either **FLDATA23**, DELT, *Label*, *Value* or **FLDATA**, DELT, *Label*, *Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

## *Label*

Degree of freedom set for which the solver is being used:

### **PRES**

Pressure equation (*Value* defaults to  $1.0 \times 10^{-10}$ ).

### **TEMP**

Energy equation (*Value* defaults to  $1.0 \times 10^{-10}$ ).

### **VX**

U velocity (*Value* defaults to  $1.0 \times 10^{-10}$ ).

### **VY**

V velocity (*Value* defaults to  $1.0 \times 10^{-10}$ ).

### **VZ**

W velocity (*Value* defaults to  $1.0 \times 10^{-10}$ ).

### **ENKE**

Turbulent kinetic energy (*Value* defaults to  $1.0 \times 10^{-10}$ ).

### **ENDS**

Turbulent kinetic energy dissipation (*Value* defaults to  $1.0 \times 10^{-10}$ ).

## *Value*

Minimum normalized rate of change (delta).

## Notes

Delta is the minimum normalized rate of change which will permit the semi-direct solution methods (method 2 or 3 on the **FLDATA18**, METH command) to continue.

Delta is used to terminate the semi-direct solvers in the event that stall occurs. If the methods stall, the solver increments the solution only a very small amount despite the fact that the correct solution has not

been achieved. The maximum nodal difference between the solutions, normalized to the value of the variable, is compared to delta.

Termination of the algebraic solver due to the small rate of change is considered a normal function and no warning message is printed. Execution of FLOTRAN continues normally.

See also the **FLDATA18**,*METH*, **FLDATA20**,*SRCH*, **FLDATA21**,*CONV*, and **FLDATA22**,*MAXI* commands for other Algebraic Solver controls.

This command is accessible in the menu if **FLDATA18**,*METH,Label*,(2 or 3) has been issued for the relevant labels.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD**  
**Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD**

## FLDATA24, TURB, Label, Value

Sets the turbulence model and the constants used in the Standard k- $\epsilon$  Model and the Zero Equation Turbulence Model.

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### TURB

Enter the word TURB in this field.

**FLDATA24**,TURB is the **FLDATA** command with its first argument set to TURB. It can be entered into the program as either **FLDATA24**,TURB,*Label,Value* or **FLDATA**,TURB,*Label,Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### Label

Turbulence model parameter label (dictates the meaning of *Value*):

### MODL

*Value* sets the turbulence model used. Valid values are the integers 0 through 8 as follows:

**0,1**Standard k- $\epsilon$  Model (default).**2**

Zero Equation Turbulence Model.

**3**

Re-Normalized Group Turbulence Model (RNG).

**4**New k- $\epsilon$  Model due to Shih (NKE).**5**

Nonlinear Model of Girimaji (GIR).

**6**

Shih, Zhu, Lumley Model (SZL).

**7**k- $\omega$  Turbulence Model**8**

Shear-Stress Transport Turbulence Model (SST)

**RATI**

*Value* is the turbulence ratio. The initial turbulent viscosity is equal to the laminar value times the turbulence ratio. *Value* defaults to 1000.

**ININ**

*Value* is the inlet intensity. The fluctuating velocity component at the inlet is assumed to be the magnitude of the inlet velocity times the inlet intensity. *Value* defaults to 0.01.

**INSF**

*Value* is the inlet scale factor. This factor is used to provide a length scale inlet region of the problem. The hydraulic diameter of the inlet is multiplied by the inlet scale factor. If a negative value is input, the absolute value is used as direct input of the hydraulic diameter. *Value* defaults to 0.01.

**SCTK**

*Value* is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.0.

**SCTD**

*Value* is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 1.3.

**SCTM**

*Value* is the Schmidt number for the momentum equation. The turbulent contribution to the diffusion term of the momentum equation is divided by this factor. *Value* defaults to 1.

**SCTT**

*Value* is the Schmidt number for the energy (temperature) equation. The turbulent contribution to the diffusion term of the energy equation is divided by this factor. The Prandtl number is equal to the Schmidt number. *Value* defaults to 0.85.

**CMU**

*Value* is the k- $\epsilon$  turbulence model constant

$$\mu_t = C_\mu \frac{\rho k^2}{\epsilon}$$

which is used in the update of the turbulent viscosity. *Value* defaults to 0.09.

**C1**

*Value* is the  $k$ - $\epsilon$  turbulence model C1 constant. It is the multiplier of the shear rate generation term of the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.44.

**C2**

*Value* is the  $k$ - $\epsilon$  turbulence model C2 constant. It is the multiplier of the dissipation source term in the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.92.

**BUC3**

*Value* is the  $k$ - $\epsilon$  buoyancy model constant. A value of zero means that there is no contribution to the turbulent kinetic energy dissipation rate equation. The default value of one is appropriate for stable thermally stratified flows. A value of zero is appropriate for unstable thermally stratified flows

**BUC4**

*Value* is the  $k$ - $\epsilon$  multiplier applied to the buoyancy term of the turbulent kinetic energy equation. A value of 1.0 is appropriate for the calculation of stable thermally stratified flows. *Value* defaults to 0.0.

**BETA**

*Value* is  $\beta$ , the coefficient of thermal expansion

$$\beta = \frac{1}{\rho} \frac{\partial \rho}{\partial T}$$

This term is used in the buoyancy terms of the  $k$ - $\epsilon$  model.

**KAPP**

*Value* is the slope parameter of the law of the wall constant. It is the slope of the plot of normalized shear velocity ( $u^+$ ) versus the nondimensionalized distance from the wall ( $y^+$ ). See [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for details. *Value* defaults to 0.4.

**EWLL**

*Value* is the law of the wall constant. It is related to the  $y$  intercept value for a plot of normalized shear velocity ( $u^+$ ) versus the nondimensionalized distance from the wall ( $y^+$ ). See the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for more details. *Value* defaults to 9.0.

**WALL**

*Value* is the choice of wall conductivity model. The default model is the Van Driest model (*Value* = VAND), used most often for high Prandtl number fluids. The second choice is the Spalding model (*Value* = SPAL), applicable to low Prandtl number fluids. The third choice is the Equilibrium model (*Value* = EQLB). The equilibrium model is also automatically invoked for the wall viscosity by this command.

**VAND**

*Value* is the constant in the Van Driest wall conductivity model. See the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for details. *Value* defaults to 26.0.

**TRAN**

*Value* is the magnitude of  $y^+$  marking the outer boundary of the laminar sublayer. Used only for the Equilibrium Wall model. *Value* defaults to 11.5.

**ZELS**

*Value* is the Zero Equation Model length scale (defaults to -1). A negative value means that FLOTRAN will calculate the value internally.

**KS**

*Value* is the local uniform wall roughness in length units. The default value of 0.0 implies a smooth wall.

**CKS**

*Value* is an empirical dimensionless factor between 0.5 and 1.0 that specifies the degree of nonuniformity of the surface. The default value of 0.5 means that the roughness signified by KS is uniformly distributed. Higher values increase the roughness losses without changing the flow regime implied by the value of KS.

**Value**

Turbulence model parameters values as explained above.

**Notes**

Sets the turbulence model and constants used in the Standard k- $\epsilon$  Model and the Zero Equation Turbulence Model. The other turbulence models are an extension of the Standard k- $\epsilon$  Model and different values are applied to some of the Standard k- $\epsilon$  Model constants and some additional constants are added.

Applicable only if the turbulence model is activated [**FLDATA1**,SOLU,TURB,TRUE]. The default values are used most often. See the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the turbulence models.

See also the **FLDATA24A**,RNGT, **FLDATA24B**,NKET, **FLDATA24C**,GIRT, and **FLDATA24D**,SZLT commands for other turbulence model constants.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Buoyancy Terms**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Parameters**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Wall Parameters**  
**Main Menu>Solution>FLOTRAN Set Up>Turbulence>Buoyancy Terms**  
**Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model**  
**Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Parameters**  
**Main Menu>Solution>FLOTRAN Set Up>Turbulence>Wall Parameters**

**FLDATA24A**, RNGT, *Label*, *Value*

Sets constants for the Re-Normalized Group Turbulence Model (RNG).

[PREP7: FLOTRAN Turbulence](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**RNGT**

Enter the word RNGT in this field.

**FLDATA24A**,RNGT is the **FLDATA** command with its first argument set to RNGT. It can be entered into the program as either **FLDATA24A**, RNGT, *Label*,*Value* or **FLDATA**,RNGT,*Label*,*Value* where *Label* and *Value* are as described below.



**Label**

Turbulence model parameter label (dictates the meaning of *Value*):

**SCTK**

*Value* is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 0.72.

**SCTD**

*Value* is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 0.72.

**CMU**

*Value* is the turbulence model constant

$$\mu_t = C_\mu \frac{\rho k^2}{\varepsilon}$$

which is used in the update of the turbulent viscosity. *Value* defaults to 0.085.

**C1**

*Value* is the multiplier of the shear rate generation term of the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.42.

**C2**

*Value* is the multiplier of the dissipation source term in the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.68.

**BETA**

*Value* is the RNG model constant,

$$\beta_\infty$$

*Value* defaults to 0.012, which corresponds to a value of 0.4 for the von Karman constant.

**ETAI**

*Value* is the asymptotic value of the strain rate parameter eta. *Value* defaults to 4.38.

**Value**

Turbulence model parameters values as explained above.

**Notes**

The Re-Normalized Group Turbulence Model (RNG) is an extension of the Standard k- $\varepsilon$  Model. Different values are applied to five of the Standard k- $\varepsilon$  Model constants and two additional constants are added. The values for the RNG Model are applied with the **FLDATA24A**,RNGT command and are separate from the Standard k- $\varepsilon$  Model constants.

Applicable only if the Re-Normalized Group Turbulence Model (RNG) is activated [**FLDATA24**,TURB,MODL,3]. The default values are used most often. See [RNG Turbulence Model](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model  
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

**FLDATA24B**, NKET, *Label*, *Value*

Sets constants for the k- $\epsilon$  Turbulence Model due to Shih (NKE).

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### NKET

Enter the word NKET in this field.

**FLDATA24B**,NKET is the **FLDATA** command with its first argument set to NKET. It can be entered into the program as either **FLDATA24B**,NKET, *Label*,*Value* or **FLDATA**,NKET,*Label*,*Value* where *Label* and *Value* are as described below.

### *Label*

Turbulence model parameter label (dictates the meaning of *Value*):

#### SCTK

*Value* is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. It is the  $\sigma_k$  constant. *Value* defaults to 1.0.

#### SCTD

*Value* is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. It is the  $\sigma_\epsilon$  constant. *Value* defaults to 1.2.

#### C2

*Value* is the multiplier of the dissipation source term in the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.90.

#### C1MX

*Value* is the maximum allowed value of the C1 constant in the turbulent kinetic energy dissipation rate equation. *Value* defaults to 0.43.

### *Value*

Turbulence model parameters values as explained above.

## Notes

The k- $\epsilon$  Turbulence Model due to Shih (NKE) is an extension of the Standard k- $\epsilon$  Model. Different values are applied to three of the Standard k- $\epsilon$  Model constants and an additional constant is added. The values for the NKE Model are applied with the **FLDATA24B**,NKET command and are separate from the Standard k- $\epsilon$  Model constants.

Applicable only if the k- $\epsilon$  Model due to Shih (NKE) is activated [**FLDATA24**,TURB,MODL,4]. The default values are used most often. See [Standard k-epsilon Model](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model  
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

**FLDATA24C**, *GIRT*, *Label*, *Value*

**Sets constants for the Nonlinear Turbulence Model of Girimaji (GIR).**

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### GIRT

Enter the word GIRT in this field.

**FLDATA24C**,*GIRT* is the **FLDATA** command with its first argument set to GIRT. It can be entered into the program as either **FLDATA24C**,*GIRT*,*Label*,*Value* or **FLDATA**,*GIRT*,*Label*,*Value* where *Label* and *Value* are as described below.

### *Label*

Turbulence model parameter label (dictates the meaning of *Value*):

#### SCTK

*Value* is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.0.

#### SCTD

*Value* is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 1.3.

#### G0

*Value* is the  $C_1^0$  constant. *Value* defaults to 3.6.

#### G1

*Value* is the  $C_1^1$  constant. *Value* defaults to 0.0.

#### G2

*Value* is the  $C_2$  constant. *Value* defaults to 0.8.

#### G3

*Value* is the  $C_3$  constant. *Value* defaults to 1.94.

#### G4

*Value* is the  $C_4$  constant. *Value* defaults to 1.16.

### *Value*

Turbulence model parameters values as explained above.

## Notes

Sets constants for the Nonlinear Turbulence Model of Girimaji (GIR), which is an extension of the Standard k- $\epsilon$  Model. Different values are applied to two of the Standard k- $\epsilon$  Model constants and five additional con-

stants are added. The values for the GIR Model are applied with the **FLDATA24C**,GIRT command and are separate from the Standard k- $\epsilon$  Model constants.

Applicable only if the Nonlinear Model of Girimaji (GIR) is activated [**FLDATA24**,TURB,MODL,5]. The default values are used most often. See [GIR Turbulence Model](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model

Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

## FLDATA24D, SZLT, Label, Value

Sets constants for the Shih, Zhu, Lumley Turbulence Model (SZL).

[PREP7: FLOTRAN Turbulence](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### SZLT

Enter the word SZLT in this field.

**FLDATA24D**,SZLT is the **FLDATA** command with its first argument set to SZLT. It can be entered into the program as either **FLDATA24D**,SZLT,*Label*,*Value* or **FLDATA**,SZLT,*Label*,*Value* where *Label* and *Value* are as described below.

### Label

Turbulence model parameter label (dictates the meaning of Value):

#### SCTK

*Value* is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.0.

#### SCTD

*Value* is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 1.3.

#### SZL1

*Value* is the numerator constant used in the calculation of  $C_\mu$ . It is the  $A_{szl1}$  constant. *Value* defaults to 0.666.

#### SZL2

*Value* is the denominator constant used in the calculation of  $C_\mu$ . It is the  $A_{szl2}$  constant. *Value* defaults to 1.25.

#### SZL3

*Value* is the strain rate multiplier. It is the  $A_{szl3}$  constant. *Value* defaults to 0.90.

### Value

Turbulence model parameters values as explained above.

## Notes

Sets constants for the Shih, Zhu, Lumley Turbulence Model (SZL), which is an extension of the Standard  $k-\epsilon$  Model. Different values are applied to two of the Standard  $k-\epsilon$  Model constants and three additional constants are added. The values for the SZL Model are applied with the **FLDATA24D,SZLT** command and are separate from the Standard  $k-\epsilon$  Model constants.

Applicable only if the Shih, Zhu, Lumley Model (SZL) is activated [**FLDATA24**,TURB,MODL,6]. The default values are used most often. See [SZL Turbulence Model](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model

Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

---

## FLDATA24E, SKWT, Label, Value

Sets constants for the  $k-\omega$  turbulence model.

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### SKWT

Enter the word SKWT in this field.

**FLDATA24E**,SKWT is the **FLDATA** command with its first argument set to SKWT. It can be entered into the program as either **FLDATA24E**,SKWT,*Label*,*Value* or **FLDATA**,SKWT,*Label*,*Value* where *Label* and *Value* are as described below.

### Label

Turbulence model parameter label (dictates the meaning of *Value*):

#### SCTK

*Value* is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 2.0.

#### SCTW

*Value* is the Schmidt number for the specific dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 2.0.

#### GAMA

*Value* is the GAMMA factor. *Value* defaults to 0.5555.

#### BETA

*Value* is the BETA factor. *Value* defaults to 0.075.

### Value

Turbulence model parameters values as explained above.

## Notes

Sets constants for the k- $\omega$  Turbulence Model. Values for the k- $\omega$  model are applied with the **FLDATA24E,SKWT** command.

Applicable only if the k- $\omega$  model is activated [**FLDATA24**,TURB,MODL,7]. The default values are used most often. See [Standard k-omega Model](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model  
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

**FLDATA24F**, *SST1*, *Label*, *Value*

Sets the turbulent production clip factor for the Shear Stress Transport (SST) turbulence model.

[PREP7: FLOTRAN Turbulence](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### SST1

Enter the word SST1 in this field.

**FLDATA24F**,*SST1* is the **FLDATA** command with its first argument set to SST1. It can be entered into the program as either **FLDATA24F**,*SST1*,*Label*,*Value* or **FLDATA**,*SST1*,*Label*,*Value* where *Label* and *Value* are as described below.

### Label

Enter the word CLMT in this field.

### Value

Turbulent production clip factor. Defaults to 1.0e15 (no clipping).

## Notes

Sets the turbulent production clip factor for the Shear Stress Transport (SST) turbulence model. Values for the k- $\omega$  regime are applied with the **FLDATA24F**,*SST1*,*CLMT* command.

Applicable only if the Shear Stress Transport model is activated [**FLDATA24**,TURB,MODL,8]. The default values are used most often. See [SST Turbulence Model](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model  
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

**FLDATA24G**, *SST1*, *Label*, *Value*

Sets constants in the  $k$ - $\omega$  regime for the Shear Stress Transport (SST) turbulence model.

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### SST1

Enter the word SST1 in this field.

**FLDATA24G**,*SST1* is the **FLDATA** command with its first argument set to *SST1*. It can be entered into the program as either **FLDATA24G**,*SST1*,*Label*,*Value* or **FLDATA**,*SST1*,*Label*,*Value* where *Label* and *Value* are as described below.

### Label

Turbulence model parameter label (dictates the meaning of *Value*):

#### SCTK

*Value* is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.176.

#### SCTW

*Value* is the Schmidt number for the specific dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 2.0.

#### GAMA

*Value* is the GAMMA factor. *Value* defaults to 0.5532.

#### BETA

*Value* is the BETA factor. *Value* defaults to 0.075.

### Value

Turbulence model parameter values as explained above.

## Notes

Sets constants in the  $k$ - $\omega$  regime for the Shear Stress Transport (SST) turbulence model. Values for the  $k$ - $\omega$  regime are applied with the **FLDATA24G**,*SST1* command.

Applicable only if the Shear Stress Transport model is activated [**FLDATA24**,TURB,MODL,8]. The default values are used most often. See *SST Turbulence Model* in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model

**FLDATA24H**, *SST2*, *Label*, *Value*

Sets constants in the  $k$ - $\epsilon$  regime for the Shear Stress Transport (SST) turbulence model.

[PREP7: FLOTRAN Turbulence](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

#### SST2

Enter the word SST2 in this field.

**FLDATA24H**,*SST2* is the **FLDATA** command with its first argument set to *SST2*. It can be entered into the program as either **FLDATA24H**,*SST2*,*Label*,*Value* or **FLDATA**,*SST2*,*Label*,*Value* where *Label* and *Value* are as described below.

#### Label

Turbulence model parameter label (dictates the meaning of *Value*):

##### SCTK

*Value* is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.0.

##### SCTW

*Value* is the Schmidt number for the specific dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 1.168.

##### GAMA

*Value* is the GAMMA factor. *Value* defaults to 0.4403.

##### BETA

*Value* is the BETA factor. *Value* defaults to 0.0828.

#### Value

Turbulence model parameter values as explained above.

## Notes

Sets constants in the  $k$ - $\epsilon$  regime for the Shear Stress Transport (SST) turbulence model. Values for the  $k$ - $\epsilon$  regime are applied with the **FLDATA24H**,*SST2* command.

Applicable only if the Shear Stress Transport model is activated [**FLDATA24**,TURB,MODL,8]. The default values are used most often. See the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model

Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model



**FLDATA25**, RELX, *Label*, *Value***Sets solution and property relaxation factors.**

PREP7: FLOTRAN Stability

MP &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; FL &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; &lt;&gt; &lt;&gt;

**RELX**

Enter the word RELX in this field.

**FLDATA25**,RELX is the **FLDATA** command with its first argument set to RELX. It can be entered into the program as either **FLDATA25**,RELX,*Label*,*Value* or **FLDATA**,RELX,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Solution and property relaxation factor labels:

**VX**

*Value* is the U Velocity relaxation factor.

**VY**

*Value* is the V Velocity relaxation factor.

**VZ**

*Value* is the W Velocity relaxation factor.

**PRES**

*Value* is the pressure relaxation factor.

**TEMP**

*Value* is the temperature relaxation factor.

**ENKE**

*Value* is the kinetic energy relaxation factor.

**ENDS**

*Value* is the dissipation rate relaxation factor.

**EVIS**

*Value* is the effective viscosity relaxation factor.

**ECON**

*Value* is the effective conductivity relaxation factor.

**DENS**

*Value* is the density update relaxation factor.

**VISC**

*Value* is the viscosity update relaxation factor.

**COND**

*Value* is the conductivity update relaxation factor.

**SPHT**

*Value* is the specific heat relaxation factor.

**Value**

Value of relaxation parameter as explained above (defaults to 0.5). Defaults to 1.0 for SPHT.

## Notes

For stability purposes the solution in FLOTRAN is under-relaxed between global iterations. When the solution of the equations for a given degree of freedom is completed, the actual set of values used is related to the calculated set and the previous values as follows (where RELX is the relaxation factor):

$$\Phi_{\text{new}} = (1 - \text{RELX}) \Phi_{\text{previous}} + \text{RELX} \Phi_{\text{calculated}}$$

See also the [FLDATA18](#),METH, [FLDATA19](#),TDMA, [FLDATA20](#),SRCH, [FLDATA21](#),CONV, [FLDATA22](#),MAXI, and [FLDATA23](#),DELT commands for other Solver Stability controls.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>DOF Relaxation**

**Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>Prop Relaxation**

**Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>DOF Relaxation**

**Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>Prop Relaxation**

**FLDATA26**, STAB, *Label*, *Value*

Sets stability controls.

[PREP7: FLOTRAN Stability](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### STAB

Enter the word STAB in this field.

**FLDATA26**,STAB is the **FLDATA** command with its first argument set to STAB. It can be entered into the program as either **FLDATA26**,STAB,*Label*,*Value* or **FLDATA**,STAB,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### *Label*

Control label (dictates the meaning of *Value*):

#### **TURB**

Turbulence inertial relaxation (*Value* defaults to  $1.0 \times 10^{15}$ ).

#### **MOME**

Momentum inertial relaxation (*Value* defaults to  $1.0 \times 10^{15}$ ).

#### **PRES**

Pressure inertial relaxation (*Value* defaults to  $1.0 \times 10^{15}$ ).

#### **TEMP**

Energy inertial relaxation (*Value* defaults to  $1.0 \times 10^{20}$ ).

#### **VISC**

Artificial viscosity (*Value* defaults to 0.0).

### *Value*

Inertial relaxation factor.

## Notes

These controls are used to make the sets of equations diagonally dominant through inertial relaxation. Making the matrix equations more diagonally dominant makes them easier to solve. More global iterations are required for convergence if inertial relaxation is used. See [Inertial Relaxation](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details.

Artificial viscosity can be activated to enhance stability in difficult compressible problems as well as in some incompressible problems. The artificial viscosity is added to the main diagonal and the forcing function of the momentum equations. The equations become more diagonally dominant locally in regions with high velocity gradients.

In compressible analyses, the artificial viscosity should be gradually removed as convergence is achieved since a nonzero value will affect the final solution. This is not necessary for incompressible analyses as the divergence of velocity should be zero. In practice, values of artificial viscosity should not exceed 1000 times the effective viscosity.

See also the [FLDATA18](#),[METH](#), [FLDATA19](#),[TDMA](#), [FLDATA20](#),[SRCH](#), [FLDATA21](#),[CONV](#), [FLDATA22](#),[MAXI](#), [FLDATA23](#),[DELT](#), [FLDATA24](#),[TURB](#), and [FLDATA25](#),[RELX](#) commands for other Solver Stability controls.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>Stability Parm**s

**Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>Stability Parm**s

---

**FLDATA27**, *PRIN*, *Label*, *Value*

**Controls dependent variable printing.**

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### PRIN

Enter the word PRIN in this field.

**FLDATA27**,*PRIN* is the **FLDATA** command with its first argument set to PRIN. It can be entered into the program as either **FLDATA27**,*PRIN*,*Label*,*Value* or **FLDATA**,*PRIN*,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### *Label*

Dependent variable tabulation is being specified for:

#### **VX**

U velocity.

#### **VY**

V velocity.

#### **VZ**

W velocity.

**PRES**

Pressure.

**TEMP**

Temperature.

**ENKE**

Kinetic energy.

**ENDS**

Turbulent kinetic energy dissipation rate.

**DENS**

Density.

**VISC**

Viscosity.

**COND**

Conductivity.

**SPHT**

Specific heat.

**EVIS**

Effective viscosity.

**ECON**

Effective conductivity.

**SFTS**

Surface tension coefficient.

**Value**

Tabulation key:

**T**

Turn tabulation on.

**F**

Turn tabulation off (default).

## Notes

When set to T, this control produces a tabulation of the values of the particular dependent variable at every node. It is set at the beginning of a load step. The printout can be large and is contained in the file `Job-name . PFL`.

To observe the values of a dependent variable at selected nodes during postprocessing, see the **PRNSOL** command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**FLDATA28**, MODR, *Label*, *Value*

Specifies that variable results are to be replaced.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**MODR**

Enter the word MODR in this field.

**FLDATA28**,MODR is the **FLDATA** command with its first argument set to MODR. It can be entered into the program as either **FLDATA28**,MODR,*Label*,*Value* or **FLDATA**,MODR,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Dependent variables to be replaced:

**VX**

U velocity.

**VY**

V velocity.

**VZ**

W velocity.

**PRES**

Pressure.

**TEMP**

Temperature.

**ENKE**

Kinetic energy.

**ENDS**

Turbulent kinetic energy dissipation rate.

**DENS**

Density.

**VISC**

Viscosity.

**COND**

Conductivity.

**SPHT**

Specific heat.

**EVIS**

Effective viscosity.

**ECON**

Effective conductivity.

**TTOT**

Total temperature.

**SFTS**

Surface tension coefficient.

**Value**

Variable replace flag:

**T**

Allow this variable to be re-initialized.

**F**

Do not allow this variable to be re-initialized (default).

## Notes

Results for any of the dependent variables available can be replaced with new values for the next load step. Only a single value may be specified for the entire solution domain. This is used for re-initialization of a variable such as temperature, which has diverged while other temperature independent calculations have been successful.

To re-initialize a variable, first use this command to set the replacement flag to T for the desired variable. Then use the **FLDATA29,MODV** command to put in the new variable value. Note that the flag(s) will be automatically reset to F after the new value has been inserted.

See also the **FLDATA29,MODV** command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Mod Res/Quad Ord>Modify Results**

**Main Menu>Solution>FLOTRAN Set Up>Mod Res/Quad Ord>Modify Results**

---

**FLDATA29,MODV,Label,Value**

**Re-initializes a results variable.**

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**MODV**

Enter the word MODV in this field.

**FLDATA29,MODV** is the **FLDATA** command with its first argument set to MODV. It can be entered into the program as either **FLDATA29,MODV,Label,Value** or **FLDATA,MODV,Label,Value** where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Dependent variables to be replaced:

**VX**

U velocity.

**VY**

V velocity.

**VZ**

W velocity.

**SP0<sub>*n*</sub>**

Mass fraction of species *n*, where *n* = 1 to 6 (FLOTRAN). If a species is given a user-defined name [**MSSPEC**], use that name instead of SP0<sub>*n*</sub>.

**PRES**

Pressure.

**TEMP**

Temperature.

**ENKE**

Kinetic energy.

**ENDS**

Turbulent kinetic energy dissipation rate.

**DENS**

Density.

**VISC**

Viscosity.

**COND**

Conductivity.

**SPHT**

Specific heat.

**EVIS**

Effective viscosity.

**ECON**

Effective conductivity.

**TTOT**

Total temperature.

**SFTS**

Surface tension coefficient.

**LMD<sub>*n*</sub>**

Laminar mass diffusion coefficient for species *n*, where *n* = 1 to 6.

**EMD<sub>*m*</sub>**

Effective mass diffusion coefficient for species *n*, where *n* = 1 to 6.

**Value**

New value of variable (defaults to 0.0). Specifying **FLDATA29,MODV,Label,Value** modifies the entire field variable to a constant value.

**Notes**

Results for any of the dependent variables available can be replaced with new values for the next load step. Only a single value may be specified for the entire solution domain. This is used for re-initialization of a variable such as temperature, which has diverged while other temperature independent calculations have been successful. Boundary conditions are reapplied upon restart.

Quantities which are not allowed to vary (e.g., CONSTANT properties) should not be modified. Also, modifying results should not be attempted on the initial run.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Mod Res/Quad Ord>Modify Results

Main Menu>Solution>FLOTRAN Set Up>Mod Res/Quad Ord>Modify Results

**FLDATA30**, *QUAD*, *Label*, *Value*

Controls the quadrature orders.

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### QUAD

Enter the word QUAD in this field.

**FLDATA30**,*QUAD* is the **FLDATA** command with its first argument set to QUAD. It can be entered into the program as either **FLDATA30**,*QUAD*,*Label*,*Value* or **FLDATA**,*QUAD*,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### *Label*

Element integral to change quadrature for:

#### **MOMD**

Momentum diffusion term (defaults to 0 integration points).

#### **MOMS**

Momentum source term (defaults to 0 integration points).

#### **PRSD**

Pressure diffusion term (defaults to 1 integration point).

#### **PRSS**

Pressure source term (defaults to 1 integration point).

#### **THRD**

Thermal diffusion term (defaults to 0 integration points).

#### **THRS**

Thermal source term (defaults to 0 integration points).

#### **TRBD**

Turbulent diffusion terms (defaults to 0 integration points).

#### **TRBS**

Turbulent source terms (defaults to 2 integration points).

### *Value*

Number of integration points.

## Notes

Controls the number of integration points used in the evaluation of element integrals. They are set at the optimum values by default. Values of 0 and 1 correspond to 1 point quadrature, but 0 means an average value of the diffusion coefficient has been used in the integrals. Values are automatically set to 2 for



axisymmetric and polar analyses. Using a value of 2 will improve the accuracy for analyses using distorted elements. Repeat the **FLDATA30** command as required.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Mod Res/Quad Ord>CFD Quad Orders**  
**Main Menu>Solution>FLOTRAN Set Up>Mod Res/Quad Ord>CFD Quad Orders**

## FLDATA31, CAPP, Label, Value

Specifies dependent variable caps.

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <>

### CAPP

Enter the word CAPP in this field.

**FLDATA31,CAPP** is the **FLDATA** command with its first argument set to CAPP. It can be entered into the program as either **FLDATA31,CAPP,Label,Value** or **FLDATA,CAPP,Label,Value** where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

### Label

Capping parameters. The first three are used to set the flags, and the rest are used to specify the cap values after the flag is set to T:

#### VELO

*Value* is the velocity capping flag (*Value* defaults to F).

#### TEMP

*Value* is the temperature capping flag (*Value* defaults to F).

#### PRES

*Value* is the pressure capping flag (*Value* defaults to F).

#### UMIN

*Value* is the minimum value of U velocity allowed (*Value* defaults to -1.E20).

#### UMAX

*Value* is the maximum value of U velocity allowed (*Value* defaults to +1.E20).

#### VMIN

*Value* is the minimum value of V velocity allowed (*Value* defaults to -1.E20).

#### VMAX

*Value* is the maximum value of V velocity allowed (*Value* defaults to +1.E20).

#### WMIN

*Value* is the minimum value of W velocity allowed (*Value* defaults to -1.E20).

#### WMAX

*Value* is the maximum value of W velocity allowed (*Value* defaults to +1.E20).

#### TMIN

*Value* is the minimum value of temperature allowed (*Value* defaults to 0.0).

**TMAX**

*Value* is the maximum value of temperature allowed (*Value* defaults to +1.E20).

**PMIN**

*Value* is the minimum value of pressure allowed (*Value* defaults to -1.E20).

**PMAX**

*Value* is the maximum value of pressure allowed (*Value* defaults to +1.E20).

**Value**

Capping flag (T or F) or capping parameter value.

**Notes**

These parameters are used to limit arbitrarily the values of the dependent variables. Capping helps prevent divergence in the early stages of analyses. Be careful when using caps to ensure that they have no impact on the final answers. You should remove capping as convergence is approached.

To use capping, you must first set the flag to T and then set the maximum and minimum caps.

The pressure value calculated by the solution of the pressure equation is capped, not the relaxed value. Therefore, if you introduce pressure capping upon restarting an analysis, pressure values may still be outside the caps.

Capping applies to relative values of pressure and absolute values of temperature.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>Results Capping**

**Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>Results Capping**

**FLDATA32**, REST, *Label*, *Value*, *Value2*, *Fname*, *Ext*, --

**Controls restart options.**

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**REST**

Enter the word REST in this field.

**FLDATA32**,REST is the **FLDATA** command with its first argument set to REST. It can be entered into the program as either **FLDATA32**,REST,*Label*,*Value*,*Value2* or **FLDATA**,REST,*Label*,*Value*,*Value2* where *Label*, *Value*, and *Value2* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Restart option (dictates the meaning of *Value* and *Value2*):

**NSET**

Identifies the restart point by results set number. *Value* is the results set number in the results file (Jobname .RFL) from which the analysis will be restarted. *Value* = 0 or blank indicates that the

restart will be from the last results set. *Value* will be reset to zero if the label ITER, LSTP, or TIME is subsequently set. *Value2* is not used.

**ITER**

Identifies the restart point by cumulative iteration number. *Value* is the cumulative iteration number in the results file (*Jobname*.RFL) from which the analysis will be restarted. If there are no results stored for this cumulative iteration, the results set with the next lowest cumulative iteration number will be used. *Value* = 0 or blank indicates that the restart will be from the last results set. *Value* will be reset to zero if the label NSET, LSTP, or TIME is subsequently set. *Value2* is not used.

**LSTP**

Identifies the restart point by load step and substep numbers. *Value* and *Value2* are the load step number and substep number in the results file (*Jobname*.RFL) from which the analysis will be restarted. *Value* = 0 or blank indicates that the restart will be from the last results set. *Value* and *Value2* will be reset to zero if the label ITER, NSET, or TIME is subsequently set.

**TIME**

Identifies the restart point by transient time. *Value* is the time point in the results file (*Jobname*.RFL) from which the analysis will be restarted. If there are no results stored for this time point, the results set with the next lowest time point will be used. *Value* = 0 or blank indicates that the restart will be from the last results set. *Value* will be reset to zero if the label NSET, LSTP, or ITER is subsequently set. *Value2* is not used.

**RFIL**

Specifies whether the CFD data structure restart file (*Jobname*.CFD) is to be read for the restart. Useful for large models where the creation of the data structures may take a long time. *Value* may be T (true) or F (false) and defaults to F. If the *Jobname*.CFD file does not exist, it will be created (if RFIL is set to true). Setting RFIL to true will toggle WFIL to false, and setting WFIL to true will toggle RFIL to false.

**WFIL**

Specifies whether the CFD data structure restart file (*Jobname*.CFD) is to be written. Useful for overwriting an existing restart file when changes in the model or boundary conditions have occurred. *Value* may be T (true) or F (false) and defaults to F. Setting WFIL to true will toggle RFIL to false, and setting RFIL to true will toggle WFIL to false.

**OVER**

Specifies whether to overwrite the set of results from which the restart occurs. *Value* may be -1, 0, or 1 (defaults to 0). If *Value* is -1, the previous set of results are overwritten. If *Value* is 1, the previous set of results is saved. If *Value* is 0 (default), the previous results are saved only if the results were written as a saved (converged) set of results. When this flag is used to change the status of the previous set of results, ANSYS sets it to 0 so that future sets of results are not affected.

**CLEAR**

Specifies whether to eliminate from the results file (*Jobname*.RFL) all results sets stored before and after the set used for the restart. *Value* may be T (true) or F (false) and defaults to F. The restart set is the last set or the set specified with another **FLDATA32,REST** command. Use a positive value of NSET, ITER, LSTP, or TIME to create a backup of the results file and use a negative value of NSET, ITER, LSTP or TIME if you do not desire a backup of the results file (see Notes below).

***Value, Value2***

Restart point or restart file flag (T or F) as described above.

***Fname***

File name and directory path of a results file to be used for the restart (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to `Jobname`. This field is valid only with `Label = NSET, ITER, LSTP, or TIME`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `RFL`, and is valid only with `Label = NSET, ITER, LSTP, or TIME`.

--

Unused field.

## Notes

If both `RFL` and `WFIL` are set to true through the GUI, the state of `WFIL` will prevail and a new `Jobname.CFD` file will be written.

If the value of `NSET, ITER, LSTP, or TIME` is positive, the original results file (`Jobname.RFL`) is moved to `Jobname.RFO`, and a new `Jobname.RFL` is created containing all the results sets stored prior to the restart point as well as new results. If the value is negative, the `Jobname.RFL` file will contain the prior results and the new results but the old file will be destroyed. This latter option is used if the results sets currently stored beyond the desired restart point are not worth saving.

If restarting from an existing file other than `Jobname.RFL`, no backup file (`Jobname.RFO`) is created because the existing file is not affected.

If a results file name (`Fname`) is entered for a restart, FLOTRAN interpolates those results onto the current mesh in the database, regardless of whether or not the mesh has changed. This causes the convergence monitors to start again from zero. This restart is different than a restart without a file name specification. However, the results quickly converge to the original solution.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>CFD Restart File**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Clear**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Iteration**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Load step**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Set**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Time**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Tag set status**  
**Main Menu>Solution>FLOTRAN Set Up>Restart Options>CFD Restart File**  
**Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Clear**  
**Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Iteration**  
**Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Load step**  
**Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Set**  
**Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Time**  
**Main Menu>Solution>FLOTRAN Set Up>Restart Options>Tag set status**

**FLDATA33**, *ADVM*, *Label*, *Value*

**Specifies the approach to discretize the advection term.**

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**ADVM**

Enter the word ADVM in this field.

**FLDATA33**,ADVM is the **FLDATA** command with its first argument set to ADVM. It can be entered into the program as either **FLDATA33**,ADVM,*Label*,*Value* or **FLDATA**,ADVM,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Specifies the transport equation.

**MOME**

Momentum equations.

**TURB**

Turbulence equations.

**PRES**

Compressible pressure equation.

**TEMP**

Energy equation.

**Value**

Choice of approach to discretize the advection term:

**MSU**

Monotone Streamline Upwind approach (default for PRES).

**SUPG**

Streamline Upwind / Petrov-Galerkin approach (default for MOME, TURB, and TEMP).

**COLG**

Collocated Galerkin (COLG) approach.

**Notes**

See [Using SUPG](#) in the *Fluids Analysis Guide* for more information on the SUPG approach.

See [Derivation of Fluid Flow Matrices](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the advection term.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Advection**

**Main Menu>Solution>FLOTRAN Set Up>Advection**

**FLDATA34**, *MIR, Label, Value*

**Sets modified inertial relaxation factors.**

[PREP7: FLOTRAN Stability](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**MIR**

Enter the word MIR in this field.

**FLDATA34**,MIR is the **FLDATA** command with its first argument set to MIR. It can be entered into the program as either **FLDATA34**,MIR,*Label*,*Value* or **FLDATA**,MIR,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Modified relaxation factor labels:

**MOME**

Momentum modified inertial relaxation.

**TURB**

Turbulence modified inertial relaxation.

**TEMP**

Energy modified inertial relaxation.

**Value**

Modified inertial relaxation factor. *Value* defaults to 0 (modified inertial relaxation off).

**Notes**

*Value* must be a positive real number. A *Value* between 0.1 and 1.0 is recommended. A larger *Value* provides a more robust scheme, but it may yield a slower convergence.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>MIR Stabilization**

**Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>MIR Stabilization**

**FLDATA35**, *VFTOL, Label, Value*

**Specifies tolerances for the lower and upper bound of the volume fraction.**

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**VFTOL**

Enter the word VFTOL in this field.

**FLDATA35**,VFTOL is the **FLDATA** command with its first argument set to VFTOL. It can be entered into the program as either **FLDATA35**,VFTOL,*Label*,*Value* or **FLDATA**,VFTOL,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Tolerance choices:

**VOFL**

Lower bound tolerance in the VOF advection algorithm (Value defaults to 1.0e-6).

**VOFU**

Upper bound tolerance in the VOF advection algorithm (Value defaults to 1.0e-6).

**LAML**

Lower bound tolerance in the solver for laminar flows (Value defaults to 1.0e-2).

**LAMU**

Upper bound tolerance in the solver for laminar flows (Value defaults to 1.0e-2).

**TRBL**

Lower bound tolerance in the solver for turbulent flows (Value defaults to 1.0e-1).

**TRBU**

Upper bound tolerance in the solver for turbulent flows (Value defaults to 1.0e-1).

**Value**

Tolerance value for Label above.

**Notes**

Volume fractions less than the lower bound tolerance are treated as 0. Volume fractions greater than 1 minus the upper bound tolerance are treated as 1.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>VOF Environment>VFRC Tolerance**

**Main Menu>Solution>FLOTRAN Set Up>VOF Environment>VFRC Tolerance**

**FLDATA36**, *AMBV*, *Label*, *Value*

**Specifies ambient reference values outside of the fluid for the volume of fluid (VOF) method.**

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**AMBV**

Enter the word AMBV in this field.

**FLDATA36**,*AMBV* is the **FLDATA** command with its first argument set to *AMBV*. It can be entered into the program as either **FLDATA36**,*AMBV*,*Label*,*Value* or **FLDATA**,*AMBV*,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Ambient Reference for which value is being specified:

**VX**

U velocity.

**VY**

V velocity.

**VZ**

W velocity.

**TEMP**

Temperature.

**ENKE**

Turbulent Kinetic Energy.

**ENDS**

Turbulent Dissipation Rate.

**Value**

Value for the ambient reference as described above.

**Notes**

PRES is used as a boundary condition at the free surface and for plotting purposes. VX, VY, VZ, TEMP, ENKE and ENDS are only used for plotting purposes.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>VOF Environment>Ambient Condit'n**

**Main Menu>Solution>FLOTRAN Set Up>VOF Environment>Ambient Condit'n**

---

**FLDATA37**, ALGR, *Label*, *Value*

**Specifies segregated solution or film coefficient algorithms.**

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**ALGR**

Enter the word ALGR in this field.

**FLDATA37**,ALGR is the **FLDATA** command with its first argument set to ALGR. It can be entered into the program as either **FLDATA37**,ALGR,*Label*,*Value* or **FLDATA**,ALGR,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Algorithm label:

**SEGR**

Segregated

**HFLM**

Film Coefficient

**Value**If *Label* = SEGR, *Value* specifies the segregated solution algorithm:



**SIMPLEF**

Original segregated algorithm (default)

**SIMPLEN**

Enhanced segregated algorithm

If *Label* = HFLM, *Value* specifies the film coefficient algorithm:

**MATX**

Conductivity matrix algorithm (default)

**TEMP**

Temperature field algorithm

## Notes

Settings automatically changed when SIMPLEN is chosen are not automatically reset if SIMPLEF is reselected. See [Coupling Algorithms](#) in the *Fluids Analysis Guide* for the settings.

The conductivity matrix algorithm uses the thermal conductivity matrix to calculate heat fluxes and film coefficients. The temperature field algorithm calculates film coefficients directly from thermal gradients.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTTRAN Set Up>Algorithm Control**

**Main Menu>Solution>FLOTTRAN Set Up>Algorithm Control**

**FLDATA38**, *MASS*, *Label*, *Value*

**Specifies the mass type for a fluid transient analysis.**

[PREP7: FLOTTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**MASS**

Enter the word MASS in this field.

**FLDATA38**,*MASS* is the **FLDATA** command with its first argument set to MASS. It can be entered into the program as either **FLDATA38**,*MASS*,*Label*,*Value* or **FLDATA**,*MASS*,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTTRAN CFD input choices.

**Label**

Degree of freedom set for which mass type is being specified:

**MOME**

Momentum equation.

**PRES**

Pressure equation.

**TURB**

Turbulent equation.

**TEMP**

Energy equation.

**ALL**

Momentum, pressure, turbulent, and energy equations.

**Value**

Mass type for fluid transient analysis:

**LUMP**

Lumped mass matrix (default).

**CONS**

Consistent mass matrix.

**Notes**

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>Preprocessor>FLOTRAN Set Up>Transient Ctrl>Mass Type

Main Menu>Solution>FLOTRAN Set Up>Transient Ctrl>Mass Type

**FLDATA39**, REMESH, *Label*, *Value*

Specifies remeshing parameters for transient fluid flow and fluid-solid interaction analyses.

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**REMESH**

Enter the word REMESH in this field.

**FLDATA39**,REMESH is the **FLDATA** command with its first argument set to REMESH. It can be entered into the program as either **FLDATA39**,REMESH,*Label*,*Value* or **FLDATA**,REMESH,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

**Label**

Remeshing fluid element label (dictates the meaning of *Value*):

**ELEM**

*Value* specifies the fluid elements included in remeshing. The following are valid values:

**NON**

Do not remesh (default).

**ALL**

Remesh all defined fluid elements if the quality of the worst defined element falls below any quality requirement. Required element qualities are specified by *Label* = ARMAX, VOCH, or ARCH as explained below.

**PAR**

Remesh defined fluid elements that have a quality below any quality requirement.

A component name may be specified for *Value*. All elements grouped into a component name are remeshed if the quality of the worst element falls below any quality requirement. The component name length is up to 8 characters.

**XBNE**

*Value* specifies the elements connected to the boundary nodes excluded from remeshing. The following are valid values:

**NON**

Do not exclude any elements connected to the boundary nodes (default).

**ALL**

Exclude all elements connected to the boundary nodes.

**FSI**

Exclude elements connected to fluid-solid interaction interfaces.

**RESIZ**

*Value* is the element size used for remeshing. *Value* defaults to 0 (the element size at the nearest boundary is used for remeshing).

**REXPN**

*Value* is the area expansion (or contraction) option for remeshing. (This option is the same as **SMRTSIZE,,,EXPND**.) This option is used to size internal elements in an area based on the size of the elements on the area's boundaries. *Value* is the expansion (or contraction) factor. For example, issuing **FLDATA39,REMESH,EXPND,2** before meshing an area will allow a mesh with elements that are approximately twice as large in the interior of an area as they are on the boundary. If *Value* is less than 1, a mesh with smaller elements on the interior of the area will be allowed. *Value* for this option should be greater than 0.5 but less than 4. *Value* defaults to 1, which does not allow expansion or contraction of internal element sizes. If *Value* = 0, the default value of 1 will be used. The actual size of the internal elements will also depend on RESIZ sizing, if used.

**ARMA**

*Value* is the maximum allowable [element generalized aspect ratio](#). *Value* defaults to 10.

**VOCH**

*Value* is the maximum allowable [change of element size](#) (area or volume). *Value* defaults to 3.

**ARCH**

*Value* is the maximum allowable [element aspect ratio change](#). *Value* defaults to 3.

**STEP**

*Value* is the element quality checking frequency based on time steps. Every *Value* time steps, a quality check takes place. *Value* defaults to 1 (a quality check at every step).

**TIME**

*Value* is the only element quality checking time. A quality check takes place at a time specified by *Value*. *Value* defaults to -1 (a quality check at every time point).

**Value**

*Value* as described for *Label* above.

**Notes**

See [Table 7.2: "Element Qualities"](#) in the *Fluids Analysis Guide* for definitions of element qualities.

Repeat command to set each *Label* as required.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Remesh Ctrl>Element size ctrl**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Remesh Ctrl>Elements for remesh**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Remesh Ctrl>Mesh qualities**  
**Main Menu>Preprocessor>FLOTRAN Set Up>Remesh Ctrl>Remesh frequency**  
**Main Menu>Solution>FLOTRAN Set Up>Remesh Ctrl>Element size ctrl**  
**Main Menu>Solution>FLOTRAN Set Up>Remesh Ctrl>Elements for remesh**  
**Main Menu>Solution>FLOTRAN Set Up>Remesh Ctrl>Mesh qualities**  
**Main Menu>Solution>FLOTRAN Set Up>Remesh Ctrl>Remesh frequency**

**FLDATA40**, *WADV*, *Label*, *Value*

**Controls activation of thermal stabilization near walls.**

[PREP7: FLOTRAN Turbulence](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### WADV

Enter the word WADV in this field.

**FLDATA40**,WADV is the **FLDATA** command with its first argument set to WADV. It can be entered into the program as either **FLDATA40**,WADV,*Label*,*Value* or **FLDATA**,WADV,*Label*,*Value* where *Label* and *Value* are as described below.

### Label

Enter the word TEMP in this field.

### Value

Value controlling *Label*:

#### TRUE or T

Turn this feature on.

#### FALSE or F

Turn this feature off.

## Notes

Thermal oscillations may occur for turbulent heat transfers in near-wall regions when using the SUPG or the COLG advection scheme with a coarse mesh. Use this command to minimize such spatial oscillations.

For the nodes that lie at the intersection of the wall or solid surface and the inlet, the thermal boundary conditions for those nodes must be the same as the wall or solid surface, not the inlet.

See the **FLDATA33**,ADVM command to see various advection discretization schemes.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>Thermal Stabilization  
Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>Thermal Stabilization

**FLLIST**, *NODE1*, *NODE2*, *NINC*

**Lists force loads on the nodes.**

SOLUTION: FE Forces

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

*NODE1*, *NODE2*, *NINC*

List forces for nodes *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If ALL, list for all selected nodes [**NSEL**] and *NODE2* and *NINC* are ignored (default). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*.

## Notes

Listing applies to the selected nodes [**NSEL**] and the selected force labels [**DOFSEL**].

### Caution

A list containing a node number that is larger than the maximum defined node (*NODE2*), could deplete the system memory and produce unpredictable results.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>Forces>On All Nodes  
Utility Menu>List>Loads>Forces>On Picked Nodes

**FLLIST**, *NLOC1*, *NLOC2*, *NINC*

**Lists the fatigue location parameters.**

POST1: Fatigue

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

*NLOC1*, *NLOC2*, *NINC*

List location parameters from *NLOC1* (defaults to 1) to *NLOC2* (defaults to *NLOC1*) in steps of *NINC* (defaults to 1). If *NLOC1* = ALL, *NLOC2* and *NINC* are ignored and all locations are listed.

## Menu Paths

Main Menu>General Postproc>Fatigue>List Stress Loc

## FLOCHECK, *Key*

**Sets up and runs a zero-iteration FLOTRAN analysis.**

[SOLUTION: FLOTRAN Checkout](#)

MP <> <> <> <> <> <> FL <> <> <> <> <> <>

### Key

Determines whether the FLOTRAN analysis is initialized and whether boundary condition status now becomes "old."

**0**

(Default) Initialization is not performed.

**1**

Initialization is performed

**2**

No initialization performed, boundary condition status now becomes "old."

## Notes

The **FLOCHECK** command will provide a results summary for the Zeroth iteration (*KEY* = 1) or the current iteration (*KEY* = 0). The results summary provides max/min/average values for each property and DOF. Also, mass flow boundaries are identified and all thermal energy transfer information is summarized. All input information is summarized in the `Jobname.PFL` file (i.e., print file). If you are running the ANSYS program from the GUI, the FLOTRAN print file is echoed to the output window. The **FLOCHECK** command aids verification of boundary condition and property specification.

Initialization [**FLOCHECK,1**] *deletes* any existing `Jobname.PFL` and `Jobname.RFL` files. Run **FLOCHECK** initialization only when you are sure you no longer need the existing results files.

The **FLOCHECK,2** command changes boundary conditions to the "old" state. It has no effect on the `Jobname.RFL` file. For information on changing fluid boundary conditions, see [Applying Transient Boundary Conditions](#) in *Fluids Analysis Guide*.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Flocheck**

**Main Menu>Solution>FLOTRAN Set Up>Flocheck**

---

## FLOTRAN

Specifies "FLOTRAN data settings" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL <> <> DY PP <> EME MFS

### Notes

This is a status [\[STAT\]](#) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu > List > Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the [STAT](#) command should immediately follow this command.

### Menu Paths

**Utility Menu > List > Status > General Postproc > FLOTRAN Module**

---

**FLREAD**, *Fname*, *Ext*, --

Reads the residual file written by the FLOTRAN CFD option.

[POST1: FLOTRAN Processing](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

#### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

#### *Ext*

Filename extension (8 character maximum).

The extension defaults to RDF (residual file) if *Fname* is blank.

--

Unused field.

### Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Menu Paths

**Main Menu > General Postproc > Read Results > FLOTRAN 2.1A**

---

**FLST**, *NFIELD*, *NARG*, *TYPE*, *Otype*, *LENG*

**Specifies data required for a picking operation (GUI).**

DATABASE: Picking

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NFIELD***

Field number on the command which uses the picking data. (Count the command name as a field, so that a 2 indicates the first command argument, 3 for the second command argument, etc.) The corresponding field on the command will have a P51X label.

***NARG***

Number of items in the picked list.

***TYPE***

Type of items picked:

- 1** Node numbers
- 2** Element numbers
- 3** Keypoint numbers
- 4** Line numbers
- 5** Area numbers
- 6** Volume numbers
- 7** Trace points
- 8** Coordinate locations (in Global Cartesian coordinates)
- 9** Screen picks (in X, Y screen coordinates (-1 to 1))

***Otype***

Data order:

**NOOR**

Data is not ordered (default).

**ORDER**

Data is in an ordered list (such as for the **E**,P51X and **A**,P51X commands, in which the order of the data items is significant for the picking operation).

***LENG***

Length of number of items describing the list (should equal *NARG* if *Otype* = NOOR; default).



## Notes

Specifies data required for the **FITEM** command during a picking operation. This is a command generated by the GUI and will appear in the log file (`Jobname.LOG`) if graphical picking is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

On the log file, **FLST** will always be followed by one or more **FITEM** commands which in turn are followed by the ANSYS command that contains a P51X label in one of its fields. This set of commands should not be edited.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

## FLUXV

**Calculates the flux passing through a closed contour.**

POST1: Magnetics Calculations

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

## Notes

**FLUXV** invokes an ANSYS macro which calculates the flux passing through a closed contour (path) predefined by **PATH**. The calculated flux is stored in the parameter FLUX. In a 2-D analysis, at least two nodes must be defined on the path. In 3-D, a path of nodes describing a closed contour must be specified (i.e., the first and last node in the path specification must be the same). A counterclockwise ordering of nodes on the **PPATH** command will give the correct sign on flux. Path operations are used for the calculations, and all path items are cleared upon completion. This macro is only available for vector potential formulations.

## Menu Paths

**Main Menu>General Postproc>Elec&Mag Calc>Path Based>Path Flux**

**FMAGBC**, *Cnam1*, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*, *Cnam9*

**Applies force and torque boundary conditions to an element component.**

SOLUTION: Misc Loads

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

*Cnam1*, *Cnam2*, *Cnam3*, . . . , *Cnam9*

Names of existing element components (**CM** command). Must be enclosed in single quotes (e.g., `'Cnam1'`) when the command is manually typed in.

## Notes

**FMAGBC** invokes a predefined ANSYS macro to apply Maxwell and virtual work force and torque boundary conditions to an element component. These boundary conditions are used for subsequent force and torque calculations during solution. Magnetic virtual displacements (MVDI = 1) are applied to nodes of elements in the components, and Maxwell surface flags (MXWF) are applied to air elements adjoining the element components. Incorrect force and torque calculations will occur for components sharing adjacent air elements. Companion macros **FMAGSUM** and **TORQSUM** can be used in POST1 to summarize the force and torque calculations, respectively. Torque calculations are valid for 2-D planar analysis only. For 2-D harmonic analysis, force and torque represent time-average values.

If using elements **SOLID117**, **PLANE121**, **SOLID122**, **SOLID123**, **PLANE233**, **SOLID236** and **SOLID237** (static analyses only), use **EMFT** to summarize electromagnetic force and torque. If you do use **FMAGSUM**, you do not need to first set either the Maxwell or the virtual work force flags via **FMAGBC**.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>Comp. Force**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>Comp. Force/Torque**

**Main Menu>Solution>Define Loads>Apply>Electric>Flag>Comp. Force**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>Comp. Force/Torque**

---

**FMAGSUM**, *Cnam1*, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*, *Cnam9*

Summarizes electromagnetic force calculations on element components.

POST1: Magnetics Calculations

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

*Cnam1*, *Cnam2*, *Cnam3*, . . . , *Cnam9*

Names of existing element components for which Maxwell or virtual work boundary conditions were applied in the preprocessor. Must be enclosed in single quotes (e.g., `Cnam1`) when the command is manually typed in.

## Notes

**FMAGSUM** invokes an ANSYS macro that summarizes the Maxwell and virtual work forces. The element components must have had appropriate Maxwell or virtual work boundary conditions established in the preprocessor prior to solution in order to retrieve forces (see **FMAGBC**, **SF**, **BF** commands). The forces are also stored on a per-element basis for the adjacent air layer surrounding the components in the element table [**ETABLE**]. Maxwell forces are stored as items FMX\_X, FMX\_Y, and FMX\_Z, and virtual work forces are stored as items FVW\_X, FVW\_Y, and FVW\_Z. Use the **PLETAB** and **PRETAB** commands to plot and list the element table items.

If using elements **SOLID117**, **PLANE121**, **SOLID122**, **SOLID123**, **PLANE233**, **SOLID236**, and **SOLID237**, use **EMFT** to summarize electromagnetic force and torque.

**FMAGSUM** can also be used to summarize time-average forces from a 2-D harmonic analysis.

## Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Component Based>Force

**/FOCUS**, *WN, XF, YF, ZF, KTRANS*

Specifies the focus point (center of the window).

GRAPHICS:Views

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **WN**

Window number (or ALL) to which command applies (defaults to 1).

### **XF, YF, ZF**

Location of the object to be at the focus point (center of the window) in the global Cartesian coordinate system. If  $XF = \text{AUTO}$ , allow automatic location calculation. If  $XF = \text{USER}$ , use focus location of last display (useful when last display had auto focus).

### **KTRANS**

Translate key:

**0**

Interpret numerical  $XF, YF, ZF$  values as described above.

**1**

Interpret  $XF, YF, ZF$  values as multiples of half-screens to translate from the current position in the screen coordinate system. Example:  $XF$  of 2.4 translates the display approximately 2.4 half-screens to the left in the screen X (horizontal) direction.

**2**

Interpret  $XF, YF, ZF$  values as multiples of half-screens to translate from the current position in the global Cartesian coordinate system. Example:  $XF$  of 1.5 translates the display approximately 1.5 half-screens in the global Cartesian X direction of the model.

## Command Default

Focus location is automatically calculated to be at the geometric center of the object (modified for centering within the window, depending upon the view).

## Notes

Specifies the location on (or off) the model which is to be located at the focus point (center of the window). For section and capped displays, the cutting plane is also assumed to pass through this location (unless the working plane is used via **/CPLANE**). See also **/AUTO** and **/USER** commands.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate  
Utility Menu>PlotCtrls>View Settings>Focus Point

## FOR2D

Calculates magnetic forces on a body.

POST1: Magnetics Calculations

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

### Notes

**FOR2D** invokes an ANSYS macro which calculates magnetic forces on a body that is completely surrounded by air (symmetry permitted). The calculated forces are stored in the parameters FX and FY. In interactive mode, a node plot is produced with the integration path highlighted. A predefined closed path [**PATH**], passing through the air elements surrounding the body, must be available for this calculation. A counter-clockwise ordering of nodes on the **PPATH** command will give the correct sign on the forces. Forces are calculated using a Maxwell stress tensor approach. The macro is valid for 2-D planar or axisymmetric analysis. Path operations are used for the calculations, and all path items are cleared upon completion.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>Mag Forces

## FORCE, Lab

Selects the element nodal force type for output.

POST1: Controls

POST26: Controls

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### Lab

Type of force to be associated with the force items:

#### TOTAL

Total forces (static, damping, and inertia).

#### STATIC

Static forces.

#### DAMP

Damping forces.

#### INERT

Inertia forces.

### Command Default

Use the total forces.

### Notes

**FORCE** selects the element nodal force type for output with the POST1 **PRESOL**, **PLESOL**, **PRRFOR**, **NFORCE**, **FSUM**, etc. commands, the POST26 **ESOL** command, and reaction force plotting [**/PBC**]. For example,

**FORCE,STATIC** causes item F of the **PRESOL** command to be the static forces for the elements processed. Element member forces (such as those available for beams and shells, which are processed by Item and Sequence number) are *not* affected by this command. Damping and inertia forces are only available for full transient and harmonic analyses.

The **PRRSOL** command is not valid with **FORCE**. Use the **PRRFOR** command, which provides the same functionality as **PRRSOL**, instead.

In POST26, the **ESOL** data stored is based on the active **FORCE** specification at the time the data is stored. To store data at various specifications (for example, static and inertia forces), issue a **STORE** command before each new specification.

The **FORCE** command cannot be used to extract static, damping, and inertial forces for **MPC184** joint elements.

## Menu Paths

**Main Menu>General Postproc>Options for Outp**

**Main Menu>TimeHist Postpro>Define Variables**

**Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables**

**Utility Menu>List>Results>Options**

---

## FORM, Lab

Specifies the format of the file dump.

[AUX2: Binary Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Lab

Format:

#### RECO

Basic record description only (minimum output) (default).

#### TEN

Same as RECO plus the first ten words of each record.

#### LONG

Same as RECO plus all words of each record.

## Command Default

Basic record description (RECO).

## Notes

Specifies the format of the file dump (from the **DUMP** command).

## Menu Paths

**Utility Menu>File>List>Binary Files**

**Utility Menu>List>Files>Binary Files**

**/FORMAT**, *NDIGIT*, *Ftype*, *NWIDTH*, *DSIGNF*, *LINE*, *CHAR*

**Specifies format controls for tables.**

[POST1: Listing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### **NDIGIT**

Number of digits (3 to 32) in first table column (usually the node or element number). Initially defaults to 7.

#### **Ftype**

FORTRAN format types (initially defaults to G):

##### **G**

$G_{xx}.yy.xx$  and  $yy$  are described below.

##### **F**

$F_{xx}.yy$

##### **E**

$E_{xx}.yy$

#### **NWIDTH**

Total width (9 to 32) of the field (the  $xx$  in *Ftype*). Initially defaults to 12.

#### **DSIGNF**

Number of digits after the decimal point ( $yy$  in F or E format) or number of significant digits in G format. Range is 2 to  $xx-7$  for *Ftype* = G or E; and 0 to  $xx-4$  for *Ftype* = F. Initially defaults to 5.

#### **LINE**

Number of lines (11 minimum) per page. Defaults to *ILINE* or *BLINE* from the **/PAGE** command.

#### **CHAR**

Number of characters (41 to 240, system-dependent) per line before wraparound. Defaults to *ICHAR* or *BCHAR* from the **/PAGE** command.

## Command Default

Program determines format for the data.

## Notes

Specifies various format controls for tables printed with the POST1 **PRNSOL**, **PRESOL**, **PRETAB**, **PRRSOL**, and **PRPATH** commands. A blank (or out-of-range) field on the command retains the current setting. Issue **/FORMAT,STAT** to display the current settings. Issue **/FORMAT,DEFA** to reestablish the initial default specifications.

For the POST26 **PRVAR** command, the *Ftype*, *NWIDTH*, and *DSIGNF* fields control the time output format.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

**FP**, *STITM*, *C1*, *C2*, *C3*, *C4*, *C5*, *C6*

**Defines the fatigue S vs. N and Sm vs. T tables.**

POST1: Fatigue

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

### ***STITM***

Starting item number for entering properties (defaults to 1). If 1, data input in field *C1* of this command is entered as the first item in the list; if 7, data input in field *C1* of this command is entered as the seventh item in the list; etc. If the item number is negative, *C1-C6* are ignored and the item is deleted. If -ALL, the table is erased. Items are as follows (items 41-62 are required only if simplified elastic-plastic code calculations are to be performed):

#### **1,2,...20**

N1, N2, ... N20

#### **21,22,...40**

S1, S2, ... S20

#### **41,42,...50**

T1, T2, ... T10

#### **51,52,...60**

Sm1, Sm2, ... Sm10

#### **61**

M (first elastic-plastic material parameter)

#### **62**

N (second elastic-plastic material parameter)

### ***C1*, *C2*, *C3*, ..., *C6***

Data inserted into six locations starting with *STITM*. If a value is already in one of these locations, it will be redefined. A blank retains the previous value.

## **Notes**

Defines the fatigue alternating stress (S) vs. cycles (N) table and the design stress-intensity value (Sm) vs. temperature (T) table. May also be used to modify any previously stored property tables. Log-log interpolation is used in the S vs. N table and linear interpolation is used in the Sm vs. T table. Cycles and temperatures must be input in ascending order; S and Sm values in descending order. Table values must be supplied in pairs, i.e., every N entry must have a corresponding S entry, etc. Not all property pairs per curve need be used. If no S vs. N table is defined, the fatigue evaluation will not produce usage factor results. See the [Structural Analysis Guide](#) for details.

## **Menu Paths**

**Main Menu>General Postproc>Fatigue>Property Table>Elas-plas Par**

**Main Menu>General Postproc>Fatigue>Property Table>Erase Tables**

**Main Menu>General Postproc>Fatigue>Property Table>S-N Table**

**Main Menu>General Postproc>Fatigue>Property Table>Sm\_T Table**

## FPLIST

Lists the property table stored for fatigue evaluation.

[POST1: Fatigue](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

## Menu Paths

Main Menu>General Postproc>Fatigue>Property Table>List Tables

---

**\*FREE**, *Name*,

Deletes a matrix or a solver object and frees its memory allocation.

[APDL: Matrix Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

*Name*

Name of the matrix or solver object to delete. Use *Name* = ALL to delete all APDL Math matrices and solver objects.

## Notes

A **/CLEAR** command will automatically delete all the current APDL Math objects.

## Menu Paths

This command cannot be accessed from a menu.

---

**FREQ**, *FREQ1*, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*, *FREQ8*, *FREQ9*

Defines the frequency points for the **SV vs. FREQ** tables.

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

*FREQ1*, *FREQ2*, *FREQ3*, . . . , *FREQ9*

Frequency points for SV vs. FREQ tables. Values must be in ascending order. *FREQ1* should be greater than zero. Units are cycles/time.

## Command Default

No frequency table.

## Notes

Repeat the **FREQ** command for additional frequency points (100 maximum). Values are added after the last nonzero frequency. If all fields (*FREQ1* -- *FREQ9*) are blank, erase SV vs. FREQ tables.



Frequencies must be in ascending order.

Spectral values are input with the **SV** command and interpreted according to the **SVTYP** command. Applies only to the SPRS (single-point) option of the **SPOPT** command. See the **SPFREQ** command for frequency input in MPRS (multi-point) analysis.

Use the **STAT** command to list current frequency points.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Erase Table**

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Freq Table**

**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Erase Table**

**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Freq Table**

---

## FRQSCL, *Scaling*

**Turns on automatic scaling of the entire mass matrix and frequency range for modal analyses using the Block Lanczos, PCG Lanczos, or Supernode mode extraction method.**

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> <> EH <> PP <> EME MFS

### *Scaling*

#### **Off**

Do not use automatic scaling of the mass matrix and frequency range.

#### **On**

Use automatic scaling of the mass matrix and frequency range.

## **Command Default**

ANSYS will use automatic scaling if appropriate.

## **Notes**

Use this command to deactivate or force activation of automatic scaling of the entire mass matrix and frequency range for modal analyses where the entire mass matrix is significantly different (i.e., orders of magnitude difference) than the entire stiffness matrix (for example, due to the particular unit system being used). Where the mass matrix is significantly smaller compared to the stiffness matrix, the eigenvalues will tend to approach very large numbers (>10e12), making the Block Lanczos, PCG Lanczos, or Supernode mode extraction method less efficient and more likely to miss modes.

ANSYS uses scaling (if appropriate) by default. However, you can issue **FRQSCL,ON** to force the entire mass matrix and frequency range to be scaled to bring the stiffness and mass matrices closer together in terms of orders of magnitude, improving efficiency and reducing the likelihood of missed modes. The resulting eigenvalues are then automatically scaled back to the original system. If you are using micro MKS units, where the density is typically very small compared to the stiffness, you may want to issue **FRQSCL,ON** to force scaling on.

If the stiffness and mass are on the same scale, **FRQSCL,ON** has no effect.

This command is available only for modal analyses using the Block Lanczos, PCG Lanczos, or Supernode mode extraction method (**MODOPT,LANB, LANPCG, or SNODE**).

This command is not valid and has no effect when used in conjunction with the **MSAVE,ON** command in a modal analysis with the PCG Lanczos mode extraction method.

## Menu Paths

**This command cannot be accessed from a menu.**

**FS, NODE, NEV, NLOD, STITM, C1, C2, C3, C4, C5, C6**

**Stores fatigue stress components at a node.**

POST1: Fatigue

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### **NODE**

Node number corresponding to this location. Used only to associate a node with a new location or to find an existing location.

### **NEV**

Event number to be associated with these stresses (defaults to 1).

### **NLOD**

Loading number to be associated with these stresses (defaults to 1).

### **STITM**

Starting item number for entering stresses (defaults to 1). If 1, data input in field *C1* of this command is entered as the first item in the list; if 7, data input in field *C1* of this command is entered as the seventh item in the list; etc. Items are as follows:

#### **1-6**

SX, SY, SZ, SXY, SYZ, SXZ total stress components

#### **7**

Temperature

#### **8-13**

SX, SY, SZ, SXY, SYZ, SXZ membrane-plus-bending stress components.

### **C1, C2, C3, . . . , C6**

Stresses assigned to six locations starting with *STITM*. If a value is already in one of these locations, it will be redefined. A blank retains the previous value (except in the *C1* field, which resets the *STITM* item to zero).

## Notes

Stores fatigue stress components at a node as input on this command instead of from the current data in the database. Stresses are stored according to the event number and loading number specified. The location is associated with that previously defined for this node [**FL**] or else it is automatically defined. May also be used to modify any previously stored stress components. Stresses input with this command should be consistent with the global coordinate system for any **FSNODE** or **FSSECT** stresses used at the same location.

## Menu Paths

Main Menu>General Postproc>Fatigue>Store Stresses>Specified Val

### **FSCALE**, *RFACT*, *IFACT*

Scales force load values in the database.

SOLUTION: FE Forces

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### *RFACT*

Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

#### *IFACT*

Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

## Notes

Scales force load (force, heat flow, etc.) values in the database. Scaling applies to the previously defined values for the selected nodes [**NSEL**] and the selected force labels [**DOFSEL**]. Issue **FLIST** command to review results. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

### Note

Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

**FSCALE** does not work for tabular boundary conditions.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Forces

Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Forces

### **FSDELE**, *NLOC*, *NEV*, *NLOD*

Deletes a stress condition for a fatigue location, event, and loading.

POST1: Fatigue

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### *NLOC*

Delete stresses associated with location *NLOC*. Defaults to zero.

**NEV**

Delete stresses associated with event *NEV*. Defaults to zero.

**NLOD**

Delete stresses associated with loading *NLOD*. Defaults to zero.

**Notes**

Deletes a stress condition stored for a particular fatigue location, event, and loading. Use **FE** command to delete all stresses for a particular event or **FL** command to delete all stresses for a particular location.

**Menu Paths**

Main Menu>General Postproc>Fatigue>Store Stresses>Dele Stresses

---

**FSLIST**, *NLOC1*, *NLOC2*, *NINC*, *NEV*, *NLOD*

**Lists the stresses stored for fatigue evaluation.**

POST1:Fatigue

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**NLOC1**, **NLOC2**, **NINC**

List stresses from *NLOC1* (defaults to 1) to *NLOC2* (defaults to *NLOC1*) in steps of *NINC* (defaults to 1). If *NLOC1* = ALL, *NLOC2* and *NINC* are ignored and stresses for all locations are listed.

**NEV**

Event number for stress listing (defaults to ALL).

**NLOD**

Loading number for stress listing (defaults to ALL).

**Notes**

Stresses may be listed per location, per event, per loading, or per stress condition. Use **FELIST** and **FLLIST** if only event and location parameters (no stresses) are to be listed.

**Menu Paths**

Main Menu>General Postproc>Fatigue>Store Stresses>List Stresses

---

**FSNODE**, *NODE*, *NEV*, *NLOD*

**Calculates and stores the stress components at a node for fatigue.**

POST1:Fatigue

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**NODE**

Node number for which stress components are stored.

**NEV**

Event number to be associated with these stresses (defaults to 1).

**NLOD**

Loading number to be associated with these stresses (defaults to 1).

## Notes

Calculates and stores the total stress components at a specified node for fatigue. Stresses are stored according to the event number and loading number specified. The location is associated with that previously defined for this node [FL] or else it is automatically defined. Stresses are stored as six total components (SX through SYZ). Temperature is also stored along with the total stress components. Calculations are made from the stresses currently in the database (last \*SET or LCASE command). Stresses stored are in global Cartesian coordinates, regardless of the active results coordinate system [RSYS]. The FSLIST command may be used to list stresses. The FS command can be used to modify stored stresses.

## Menu Paths

**Main Menu>General Postproc>Fatigue>Store Stresses>From rst File**

### FSPLOT, NLOC, NEV, ITEM

**Displays a fatigue stress item for a fatigue location and event.**

POST1:Fatigue

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**NLOC**

Display stresses associated with location *NLOC*.

**NEV**

Display stresses associated with event *NEV*.

**ITEM**

Display stresses associated with item number *ITEM*. Items are as follows:

**16**

SX, SY, SZ, SXY, SYZ, SXZ total stress components

**7**

Temperature

**813**

SX, SY, SZ, SXY, SYZ, SXZ membrane-plus-bending stress components.

## Notes

Displays a fatigue stress item as a function of loading number for a particular fatigue location and event.

## Menu Paths

**Main Menu>General Postproc>Fatigue>Store Stresses>Plot Stresses**

**FSSECT**, *RHO*, *NEV*, *NLOD*, *KBR*

**Calculates and stores total linearized stress components.**

POST1: Fatigue

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

**RHO**

In-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If zero (or blank), a plane or 3-D structure is assumed. If nonzero, an axisymmetric structure is assumed. Use a suitably large number (see the *Theory Reference for the Mechanical APDL and Mechanical Applications*) or use -1 for an axisymmetric straight section.

**NEV**

Event number to be associated with these stresses (defaults to 1).

**NLOD**

Loading number to be associated with these stresses (defaults to 1).

**KBR**

For an axisymmetric analysis ( $RHO \neq 0$ ):

**0**

Include the thickness-direction bending stresses

**1**

Ignore the thickness-direction bending stresses

**2**

Include the thickness-direction bending stress using the same formula as the Y (axial direction) bending stress. Also use the same formula for the shear stress.

## Notes

Calculates and stores the total linearized stress components at the ends of a section path **[PATH]** (as defined by the first two nodes with the **PPATH** command). The path must be entirely within the selected elements (that is, there must not be any element gaps along the path). Stresses are stored according to the fatigue event number and loading number specified. Locations (one for each node) are associated with those previously defined for these nodes **[FL]** or else they are automatically defined. Stresses are separated into six total components (SX through SXZ) and six membrane-plus-bending (SX through SXZ) components. The temperature at each end point is also stored along with the total stress components. Calculations are made from the stresses currently in the database (last **\*SET** or **LCASE** command). Stresses are stored as section coordinate components if axisymmetric or as global Cartesian coordinate components otherwise, regardless of the active results coordinate system **[RSYS]**. The **FSLIST** command may be used to list stresses. The **FS** command can be used to modify stored stresses. See also the **PRSECT** and **PLSECT** commands for similar calculations.

## Menu Paths

**Main Menu>General Postproc>Fatigue>Store Stresses>At Cross Sect**

**FSSPARM**, *PORT1*, *PORT2*

Calculates reflection and transmission properties of a frequency selective surface.

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**PORT1**

Port number of input port. Defaults to 1.

**PORT2**

Port number of output port. Defaults to 1.

**Notes**

**FSSPARM** calculates reflection and transmission coefficients, power reflection and transmission coefficients, and return and insertion losses of a frequency selective surface.

**Menu Paths**

Main Menu>General Postproc>Elec&Mag Calc>Port>FSS Parameters

**FSUM**, *LAB*, *ITEM*

Sums the nodal force and moment contributions of elements.

POST1: Special Purpose

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**LAB**

Coordinate system in which to perform summation.

**(blank)**

Sum all nodal forces in global Cartesian coordinate system (default).

**RSYS**

Sum all nodal forces in the currently active RSYS coordinate system.

**ITEM**

Selected set of nodes.

**(blank)**

Sum all nodal forces for all selected nodes (default), excluding contact elements.

**CONT**

Sum all nodal forces for contact nodes only.

**BOTH**

Sum all nodal forces for all selected nodes, including contact elements.

**Notes**

Sums and prints, in each component direction for the total selected node set, the nodal force and moment contributions of the selected elements attached to the node set. Selecting a subset of nodes [**NSEL**] and then issuing this command will give the total force acting on that set of nodes (default), excluding surface-to-surface, node-to-surface, line-to-line, and line-to-surface contact elements (**TARGE169**, **TARGE170**, **CONTA171**,

**CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, and CONTA177**). Setting *ITEM* = **CONT** sums the nodal forces and moment contributions of the selected contact elements (**CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, and CONTA177**). Setting *ITEM* = **BOTH** sums the nodal forces for all selected nodes, including contact elements. Nodal forces associated with surface loads are not included. The effects of nodal coupling and constraint equations are ignored. Moment summations are about the global origin unless another point is specified with the **SPOINT** command. This vector sum is printed in the global Cartesian system unless it is transformed [**RSYS**] and a point is specified with the **SPOINT** command. By default, the sum is done in global Cartesian, and the resulting vector is transformed to the requested system.

The *LAB* = **RSYS** option transforms each of the nodal forces into the active coordinate system before summing and printing. The **FORCE** command can be used to specify which component (static, damping, inertia, or total) of the nodal load is to be used. This command output is included in the **NFORCE** command.

The **FSUM** command should not be used with axisymmetric elements because it might calculate a moment where none exists. Consider, for example, the axial load on a pipe modeled with an axisymmetric shell element. The reaction force on the end of the pipe is the total force (for the full 360 degrees) at that location. The net moment about the centerline of the pipe would be zero, but the program would incorrectly calculate a moment at the end of the element as the force multiplied by the radius.

The **FSUM** command should also not be used with the **SOLID117** element.

## Using FSUM with the NLGEOM Command

If you have activated large deflection (via the **NLGEOM,ON** command), the **FSUM** command generates the following message:

```
Summations based on final geometry and  
will not agree with solution reactions.
```

The message warns that the moment reactions are incorrect. When computing moment reactions, the command assumes that the summation of rotations applies; however, it does *not* apply for large rotations, which require pseudovector representation to sum the rotations.

In contrast, the results for force reactions will be correct because they depend upon linear displacement vectors (which can be added).

## Using FSUM in a Spectrum or PSD Analysis (ANTYPE, SPECTR)

When using **FSUM** in a spectrum analysis after the combination file has been input (**/INPUT,,MCOM**), or in a PSD analysis when postprocessing 1-sigma results (loadstep 3, 4, or 5), the following message will display in the printout header:

```
(Spectrum analysis summation is used)
```

This message means that the summation of the element nodal forces is performed prior to the combination of those forces. In this case, **RSYS** does not apply. The forces are in the nodal coordinate systems, and the vector sum is always printed in the global coordinate system.

Because modal displacements cannot be used to calculate contact element nodal forces, *ITEM* does not apply to spectrum and PSD analyses.



## Menu Paths

Main Menu>General Postproc>Nodal Calcs>Total Force Sum

---

### FTCALC, NLOC, NODE

Performs fatigue calculations for a particular node location.

POST1: Fatigue

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### NLOC

Location number of stress conditions to be used for fatigue calculation.

#### NODE

Node number (used only for convenience if *NLOC* is not input).

## Menu Paths

Main Menu>General Postproc>Fatigue>Calculate Fatig

---

### FTRAN

Transfers solid model forces to the finite element model.

SOLUTION: Solid Forces

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

## Notes

Forces are transferred only from selected keypoints to selected nodes. The **FTRAN** operation is also done if the **SBCTRAN** command is issued or automatically done upon initiation of the solution calculations [**SOLVE**].

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Forces

Main Menu>Solution>Define Loads>Operate>Transfer to FE>Forces

---

### FTSIZE, MXLOC, MXEV, MXLOD

Defines the fatigue data storage array.

POST1: Fatigue

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### MXLOC

Maximum number of fatigue locations (defaults to 5).

***MXEV***

Maximum number of fatigue events (defaults to 10).

***MXLOD***

Maximum number of loadings in each event (defaults to 3).

## Command Default

5 locations, 10 events, 3 loadings.

## Notes

Defines the size and erases the stress conditions for the fatigue data storage array. A stress condition is a loading (stresses) at a particular location (node) for a particular event. Size is defined in terms of the maximum number of locations, events, and loadings. The array size cannot be changed once data storage has begun (without erasing all previously stored data). If a size change is necessary, see the **FTWRITE** command.

## Menu Paths

**Main Menu>General Postproc>Fatigue>Size Settings**

---

**FTWRITE**, *Fname*, *Ext*, --

**Writes all currently stored fatigue data on a file.**

[POST1: Fatigue](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `FATG` if *Fname* is blank.

--

Unused field.

## Notes

- Data are written in terms of the equivalent POST1 fatigue commands [**FTSIZE**, **FL**, **FS**, etc.] which you can then edit and resubmit to POST1 (with a **/INPUT** command).
- Once you have created a fatigue data file, each subsequent use of the **FTWRITE** command overwrites the contents of that file.

## Menu Paths

Main Menu>General Postproc>Fatigue>Write Fatig Data

## FVMESH, *KEEP*

Generates nodes and tetrahedral volume elements from detached exterior area elements (facets).

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *KEEP*

Specifies whether to keep the area elements after the tetrahedral meshing operation is complete.

0

Delete area elements (default).

1

Keep area elements.

## Notes

The **FVMESH** command generates a tetrahedral volume mesh from a selected set of detached exterior area elements (facets). (Detached elements have no solid model associativity.) The area elements can be triangular-shaped, quadrilateral-shaped, or a mixture of the two.

The **FVMESH** command is in contrast to the **VMESH** command, which requires a volume to be input.

The main tetrahedra mesher [**MOPT**,**VMESH**,**MAIN**] is the only tetrahedra mesher that supports the **FVMESH** command. The alternate tetrahedra mesher [**MOPT**,**VMESH**,**ALTERNATE**] does not support **FVMESH**. **MESH200** elements do not support **FVMESH**.

Tetrahedral mesh expansion [**MOPT**,**TETEXPND**,*Value*] is supported for both the **FVMESH** and **VMESH** commands. Tet-mesh expansion is the only mesh control supported by **FVMESH**.

Triangle- or quadrilateral-shaped elements may be used as input to the **FVMESH** command. Where quadrilaterals are used, the default behavior is for the pyramid-shaped elements to be formed at the boundary when the appropriate element type is specified. See the **MOPT**,**PYRA** command for details.

The **FVMESH** command does not support multiple "volumes." If you have multiple volumes in your model, select the surface elements for one "volume," while making sure that the surface elements for the other volumes are deselected. Then use **FVMESH** to generate a mesh for the first volume. Continue this procedure by selecting one volume at a time and meshing it, until all of the volumes in the model have been meshed.

If an error occurs during the meshing operation, the area elements are kept even if *KEEP* = 0.

## Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Tet Mesh From>Area Elements



---

## G Commands

### GAP

Specifies "Reduced transient gap conditions" as the subsequent status topic.

[SOLUTION: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This is a status [\[STAT\]](#) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the [STAT](#) command should immediately follow this command.

### Menu Paths

**Utility Menu>List>Status>Solution>Gap Conditions**

---

**GAPF**, *NVAR*, *NUM*, *Name*

Defines the gap force data to be stored in a variable.

[POST26: Set Up](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### *NVAR*

Arbitrary reference number assigned to this variable (2 to NV [[NUMVAR](#)]). Overwrites any existing results for this variable.

#### *NUM*

Number identifying gap number for which the gap force is to be stored. Issue the [GPLIST](#) command to display gap numbers.

#### *Name*

Thirty-two character name for identifying the item on the printout and displays (defaults to the name **GAPF**).

### Notes

Defines the gap force data to be stored in a variable. Applicable only to the expansion pass of the reduced or mode superposition linear transient dynamic ([ANTYPE,TRANS](#)) analysis. The data usually on *Fname*.RDSP.

## Menu Paths

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

## GAUGE, *Opt*, *FREQ*

Gauges the problem domain for a magnetic edge-element formulation.

[SOLUTION: Analysis Options](#)

MP ME <> <> <> <> <> <> EM <> <> PP <> EME MFS

### *Opt*

Type of gauging to be performed:

#### ON

Perform tree gauging of the edge values (default).

#### OFF

Gauging is off. (You must specify custom gauging via APDL specifications.)

#### STAT

Gauging status (returns the current *Opt* and *FREQ* values)

### *FREQ*

The following options are valid when *Opt* = ON:

#### 0

Generate tree-gauging information once, at the first load step. Gauging data is retained for subsequent load steps. (This behavior is the default.)

#### 1

Repeat gauging for each load step. Rewrites the gauging information at each load step to accommodate changing boundary conditions on the AZ degree of freedom (for example, adding or deleting AZ constraints via the **D** or **CE** commands).

## Notes

The **GAUGE** command controls the tree-gauging procedure required for electromagnetic analyses using an edge-based magnetic formulation (elements [SOLID117](#), [SOLID236](#) and [SOLID237](#)).

Gauging occurs at the solver level for each solution (**SOLVE**). It sets additional zero constraints on the edge-flux degrees of freedom AZ to produce a unique solution; the additional constraints are removed after solution.

Use the *FREQ* option to specify how the command generates gauging information for multiple load steps.

Access the gauging information via the `_TGAUGE` component of gauged nodes. The program creates and uses this component internally to remove and reapply the AZ constraints required by gauging. If *FREQ* = 0, the `_TGAUGE` component is created at the first load step and is used to reapply the tree gauge constraints at subsequent load steps. If *FREQ* = 1, the tree-gauging information and the `_TGAUGE` component are generated at every load step

If gauging is turned off (**GAUGE,OFF**), you must specify your own gauging at the APDL level.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>Gauging**  
**Main Menu>Solution>Load Step Opts>Magnetics>Options Only>Gauging**

**/GCMD**, *WN, Lab1, Lab2, Lab3, Lab4, Lab5, Lab6, Lab7, Lab8, Lab9, Lab10, Lab11, Lab12*

**Controls the type of element or graph display used for the **G**PLOT command.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WN**

Window number (or ALL) to which this command applies (defaults to 1)

**Lab1, Lab2, Lab3, . . . , Lab12**

Command labels (for example, **PLNSOL,S,X**)

## Notes

This command controls the type of element or graph display that appears when you issue the **G**PLOT command when the **/GTYPE**, (ELEM or GRPH) entity type is active. If you have multiple plotting windows enabled, you can also use **/GCMD** to select one window when you wish to edit its contents.

For related information, see the descriptions of the **G**PLOT and **/GTYPE** commands in this manual.

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Multi-Plot Contrls**

**/GCOLUMN**, *CURVE, STRING*

**Allows the user to apply a label to a specified curve.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**CURVE**

Curve number on which label will be applied (integer value between 1 and 10).

**STRING**

Name or designation that will be applied to the curve (8 characters max).

## Notes

This command is used for an array parameter plot (a plot created by the **\*V**PLOT command). Normally the label for curve 1 is "COL 1", the label for curve 2 is "COL 2" and so on; the column number is the field con-

taining the dependent variables for that particular curve. Issuing `/GCOLUMN,CURVE`, with no string value specified resets the label to the original value.

## Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Curve

---

## GENOPT

Specifies "General options" as the subsequent status topic.

SOLUTION: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status `[STAT]` topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a `STAT` command, which will report the status for the specified topic.

If entered directly into the program, the `STAT` command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Solution>General Options

---

## GEOM, *K2D*, *NDIV*

Defines the geometry specifications for the radiation matrix calculation.

AUX12: Radiation Matrix Method

MP ME ST PR <> <> <> <> <> <> PP <> EME MFS

### *K2D*

Dimensionality key:

**0**

3-D geometry

**1**

2-D geometry (plane or axisymmetric)

### *NDIV*

Number of divisions in an axisymmetric model. Used only with  $K2D = 1$ . Defaults to 0 (2-D plane). The 2-D model is internally expanded to a 3-D model based on the number of divisions specified ( $6 \leq NDIV \leq 90$ ). For example, *NDIV* of 6 is internally represented by six 60° sections.

## Command Default

3-D geometry.



## Menu Paths

Main Menu>Radiation Opt>Matrix Method>Other Settings

## GEOMETRY

Specifies "Geometry" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Preprocessor>Solid Model

**\*GET**, *Par*, *Entity*, *ENTNUM*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*

Retrieves a value and stores it as a scalar parameter or part of an array parameter.

[APDL: Parameters](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Par*

The name of the resulting parameter. See **\*SET** for name restrictions.

### *Entity*

Entity keyword. Valid keywords are NODE, ELEM, KP, LINE, AREA, VOLU, PDS, etc., as shown for *Entity* = in the tables below.

### *ENTNUM*

The number or label for the entity (as shown for *ENTNUM* = in the tables below). In some cases, a zero (or blank) *ENTNUM* represents all entities of the set.

### *Item1*

The name of a particular item for the given entity. Valid items are as shown in the *Item1* columns of the tables below.

### *IT1NUM*

The number (or label) for the specified *Item1* (if any). Valid *IT1NUM* values are as shown in the *IT1NUM* columns of the tables below. Some *Item1* labels do not require an *IT1NUM* value.

### *Item2, IT2NUM*

A second set of item labels and numbers to further qualify the item for which data are to be retrieved. Most items do not require this level of information.

## Notes

**\*GET** retrieves a value for a specified item and stores the value as a scalar parameter, or as a value in a user-named array parameter. An item is identified by various keyword, label, and number combinations. Usage is similar to the **\*SET** command except that the parameter values are retrieved from previously input or calculated results. For example, , A, ELEM, 5, CENT, X returns the centroid x-location of element 5 and stores the result as parameter A. **\*GET** command operations, along with the associated Get functions return values in the active coordinate system unless stated otherwise. A Get function is an alternative in-line function that can be used to retrieve a value instead of the **\*GET** command (see [Using In-line Get Functions](#) for more information).

Both **\*GET** and **\*VGET** retrieve information from the active data stored in memory. The database is often the source, and sometimes the information is retrieved from common memory blocks that ANSYS uses to manipulate information. Although POST1 and POST26 operations use a \*.rst file, **\*GET** data is accessed from the database or from the common blocks. Get operations do not access the \*.rst file directly. For repeated gets of sequential items, such as from a series of elements, see the **\*VGET** command.

Most items are stored in the database after they are calculated and are available anytime thereafter. Items are grouped according to where they are usually first defined or calculated. Preprocessing data will often not reflect the calculated values generated from section data. You should not use **\*GET** to obtain data from elements that use calculated section data, such as beams or shells. Most of the GENERAL items listed below are available from all modules. Each of the sections for accessing **\*GET** parameters are shown in the following order:

- [\\*GET General Entity Items \(p. 718\)](#)
- [\\*GET Preprocessing Entity Items \(p. 722\)](#)
- [\\*GET Solution Entity Items \(p. 748\)](#)
- [\\*GET Postprocessing Entity Items \(p. 750\)](#)
- [\\*GET Optimization and Probabilistic Design Entity Items \(p. 764\)](#)

The **\*GET** command is valid in any processor.

## General Items

### **\*GET General Entity Items**

- [Table 150: \\*GET General Items, Entity = ACTIVE \(p. 719\)](#)
- [Table 151: \\*GET General Items, Entity = CMD \(p. 720\)](#)
- [Table 152: \\*GET General Items, Entity = COMP \(p. 720\)](#)
- [Table 153: \\*GET General Items, Entity = GRAPH \(p. 721\)](#)

- Table 154: \*GET General Items, Entity = PARM (p. 722)

**Table 150 \*GET General Items, Entity = ACTIVE**

<i>Entity = ACTIVE, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, ACTIVE, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
INT		Current interactive key: 0=off, 2=on.
IMME		Current immediate key: 0=off, 1=on.
MENU		Current menu key: 0=off, 1=on.
PRKEY		Printout suppression status: 0= <b>/NOPR</b> , 1= <b>/GOPR</b> or <b>/GO</b>
UNITS		Units specified by <b>/UNITS</b> command: 0 = USER, 1 = SI, 2 = CGS, 3 = BFT, 4 = BIN, 5 = MKS, 6 = MPA, 7 = uMKS.
ROUT		Current routine: 0 = Begin level, 17 = PREP7, 21 = SOLUTION, 31 = POST1, 36 = POST26, 41 = OPT, 52 = AUX2, 62 = AUX12, 65 = AUX15, 71 = RUNSTAT.
TIME	WALL,CPU	Current wall clock or CPU time. Current wall clock will continue to accumulate during an ANSYS run and is NOT reset to zero at midnight.
DBASE	LDATE	Date of first modification of any database quantity required for POST1 operation. The parameter returned is <i>Par</i> = YEAR*10000 + MONTH*100 + DAY.
DBASE	LTIME	Time of last modification of any database quantity required for POST1 operation. The parameter returned is <i>Par</i> = HOURS*10000 + MINUTES*100 + SECONDS.
REV		ANSYS minor revision number (5.6, 5.7, 6.0 etc.). Letter notation (e.g., 5.0A) is not included.
TITLE	0,1,2,3,4	<b>Item2:</b> START <b>IT2NUM:</b> <i>N</i> Current title string of the main title ( <i>IT1NUM</i> =0 or blank) or subtitle 1, 2, 3, or 4 ( <i>IT1NUM</i> =1,2,3, or 4). A character parameter of up to 8 characters, starting at position <i>N</i> , is returned.
JOBNAM		<b>Item2:</b> START <b>IT2NUM:</b> <i>N</i> Current Jobname. A character parameter of up to 8 characters, starting at position <i>N</i> , is returned. Use <b>*DIM</b> and <b>*DO</b> to get all 32 characters.
PLATFORM		The current platform.
NPROC	CURR, MAX	The number of processors being used for the current session, or the maximum number of processors available on the machine. This only applies to shared memory parallelism.

<i>Entity = ACTIVE, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, ACTIVE, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NUMCPU		Number of Distributed ANSYS processes being used (distributed memory parallel).

**Table 151 \*GET General Items, Entity = CMD**

<i>Entity = CMD, ENTNUM = 0 (or blank)</i>		
The following items are valid for all commands except star (*) commands and non-graphics slash (/) commands.		
<b>*GET, Par, CMD, 0, Item1, IT1NUM, Item2, IT2NUM.</b>		
Item1	IT1NUM	Description
STAT		Status of previous command: 0=found, 1=not found (unknown).
NARGS		Field number of last nonblank field on the previous command.
FIELD	2,3... <i>N</i>	Numerical value of the <i>N</i> th field on the previous command. Field 1 is the command name (not available)

**Table 152 \*GET General Items, Entity = COMP**

<i>Entity = COMP, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, COMP, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NCOMP		Total number of components and assemblies currently defined.
<i>Entity = COMP, ENTNUM = n (nth component)</i>		
<b>*GET, Par, COMP, n, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NAME		Name of the <i>N</i> th item (component or assembly) in the list of components and assemblies. A character parameter is returned.
<i>Entity = COMP, ENTNUM = Cname (component or assembly name)</i>		
<b>*GET, Par, COMP, Cname, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
TYPE		Type of component <i>Cname</i> : 1=Nodes, 2=Elements, 6=Keypoints, 7=Lines, 8=Areas, 9=Volumes, 11-15=Subcomponents (11=subcomponent at level 1, 12=subcomponent at level 2, etc.).
NSCOMP		Number of subcomponents (for assemblies).

<i>Entity</i> = COMP, <i>ENTNUM</i> = <i>Cname</i> (component or assembly name)		
*GET, <i>Par</i> , COMP, <i>Cname</i> , <i>Item1</i> , <i>IT1NUM</i> , <i>Item2</i> , <i>IT2NUM</i>		
Item1	IT1NUM	Description
SNAME	<i>N</i>	Name of <i>N</i> th subcomponent of assembly <i>Cname</i> . A character parameter is returned.

Table 153 \*GET General Items, Entity = GRAPH

<i>Entity</i> =GRAPH, <i>ENTNUM</i> = <i>N</i> (window number)		
*GET, <i>Par</i> , GRAPH, <i>N</i> , <i>Item1</i> , <i>IT1NUM</i> , <i>Item2</i> , <i>IT2NUM</i>		
Item1	IT1NUM	Description
ACTIVE		<b>/WINDOW</b> status: 0=off, 1=on.
ANGLE		<b>/ANGLE</b> <i>THETA</i> angle.
CONTOUR	<i>Name</i>	<b>/CONTOUR</b> value for <i>Name</i> , where <i>Name</i> = VMIN, VINC, or NCONT.
DIST		<b>/DIST</b> <i>DVAL</i> value.
DSCALE	DMULT	<b>/DSCALE</b> <i>DMULT</i> value.
EDGE		<b>/EDGE</b> <i>KEY</i> value.
FOCUS	X, Y, Z	<b>/FOCUS</b> <i>XF</i> , <i>YF</i> , or <i>ZF</i> value.
GLINE		<b>/GLINE</b> <i>STYLE</i> value.
MODE		<b>/USER</b> or <b>/AUTO</b> setting: 0=user, 1=auto.
NORMAL		<b>/NORMAL</b> <i>KEY</i> value.
RANGE	XMIN, XMAX, YMIN, YMAX	<b>/WINDOW</b> <i>XMIN</i> , <i>XMAX</i> , <i>YMIN</i> , or <i>YMAX</i> screen coordinates.
RATIO	X, Y	<b>/RATIO</b> <i>RATOX</i> or <i>RATOY</i> value.
SSCALE	SMULT	<b>/SSCALE</b> <i>SMULT</i> value.
TYPE		<b>/TYPE</b> <i>Type</i> value.
VCONE	ANGLE	<b>/VCONE</b> <i>PHI</i> angle.
VIEW	X, Y, Z	<b>/VIEW</b> <i>XV</i> , <i>YV</i> , or <i>ZV</i> value.
VSCALE	VRATIO	<b>/VSCALE</b> <i>VRATIO</i> value.
<i>Entity</i> =GRAPH, <i>ENTNUM</i> = 0 (or blank)		
*GET, <i>Par</i> , GRAPH, 0, <i>Item1</i> , <i>IT1NUM</i> , <i>Item2</i> , <i>IT2NUM</i>		
Item1	IT1NUM	Description
DISPLAY		<b>/SHOW</b> <i>VECT</i> setting: 0=raster, 1=vector.
ERASE		<b>/ERASE</b> or <b>/NOERASE</b> setting: 0=no erase, 1=erase.
NDIST		Largest nodal range for current model (DX, DY, or DZ of the model).
NUMBER		<b>/NUMBER</b> <i>NKEY</i> value.
PLOPTS	<i>Name</i>	<b>/PLOPTS</b> setting of <i>Name</i> , where <i>Name</i> =LEG1, LEG2, LEG3, INFO, FRAM, TITL, MINM, or VERS.
SEG		Segment capability of graphics driver: 0=no segments available, 1=erasable segments available, 2=non-erasable segments available.

<i>Entity</i> =GRAPH, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, GRAPH, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
SHRINK		/SHRINK <i>RATIO</i> value.

**Table 154 \*GET General Items, Entity = PARM**

<i>Entity</i> , = PARM <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, PARM, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
MAX		Total number of parameters currently defined.
BASIC		Number of scalar parameters (excluding parameters beginning with an underscore _, array parameters, and character parameters).
LOC	<i>Num</i>	Name of the parameter at the <i>Num</i> location in the parameter table. A character parameter is returned.
<i>Entity</i> = PARM, <i>ENTNUM</i> = Name (parameter name)		
<b>*GET, Par, PARM, Name, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
TYPE		Parameter type: 0 = scalar, 1 = array, 2 = table, 3 = character scalar, 4 = character array, -1 = undefined
DIM	X(1), Y(2), Z(3), (4), (5)	Row (X or 1), Column (Y or 2), Plane (Z or 3), Book (4), or Shelf (5) dimension of array parameter.
CSYS		Local coordinate system number
VAR	1, 2, 3, 4, 5	Name of primary variables 1-5. Primary variable names are character strings.
If Item1 = DIM and ITNUM refers to a dimension that does not exist, ANSYS assigns a value of -1 to <i>Par</i> .		
If Item1 = CSYS and no local coordinate system number was assigned to the array parameter ( <i>Name</i> ), ANSYS assigns a value of ZERO to <i>Par</i> .		
If Item1 = VAR and if IT1NUM refers to a primary variable that does not exist, ANSYS assigns a value of BLANK to <i>Par</i> .		

## Preprocessing Items

### \*GET Preprocessing Entity Items

- [Table 155: \\*GET Preprocessing Items, Entity = ACTIVE \(p. 724\)](#)
- [Table 156: \\*GET Preprocessing items, Entity = AREA \(p. 724\)](#)
- [Table 157: \\*GET Preprocessing Items, Entity = AXIS \(p. 725\)](#)
- [Table 158: \\*GET Preprocessing Items, Entity = CDSY \(p. 726\)](#)
- [Table 159: \\*GET Preprocessing Items, Entity = CE \(p. 726\)](#)
- [Table 160: \\*GET Preprocessing Items, Entity = CMPB \(p. 727\)](#)

- *Table 161: \*GET Preprocessing Items, Entity = CP (p. 727)*
- *Table 162: \*GET Preprocessing Items, Entity = CSEC (p. 728)*
- *Table 163: \*GET Preprocessing Items, Entity = EDCC (p. 728)*
- *Table 164: \*GET Preprocessing Items, Entity = ELEM (p. 729)*
- *Table 165: \*GET Preprocessing Items, Entity = ETYP (p. 730)*
- *Table 166: \*GET Preprocessing Items, Entity = FLDATA (p. 731)*
- *Table 167: \*GET Preprocessing Items, Entity = GENB (p. 731)*
- *Table 168: \*GET Preprocessing Items, Entity = GENS (p. 732)*
- *Table 169: \*GET Preprocessing Items, Entity = KP (p. 733)*
- *Table 170: \*GET Preprocessing Items, Entity = LINE (p. 734)*
- *Table 171: \*GET Preprocessing Items, Entity = MAT (p. 735)*
- *Table 172: \*GET Preprocessing Items, Entity = MPLAB (p. 736)*
- *Table 173: \*GET Preprocessing Items, Entity = MSCAP (p. 736)*
- *Table 174: \*GET Preprocessing Items, Entity = MSDATA (p. 736)*
- *Table 175: \*GET Preprocessing Items, Entity = MSMETH (p. 737)*
- *Table 176: \*GET Preprocessing Items, Entity = MSNOMF (p. 737)*
- *Table 177: \*GET Preprocessing Items, Entity = MSPROP (p. 737)*
- *Table 178: \*GET Preprocessing Items, Entity = MSRELAX (p. 737)*
- *Table 179: \*GET Preprocessing Items, Entity = MSSOLU (p. 738)*
- *Table 180: \*GET Preprocessing Items, Entity = MSSPEC (p. 738)*
- *Table 181: \*GET Preprocessing Items, Entity = MSVARY (p. 738)*
- *Table 182: \*GET Preprocessing Items, Entity = NODE (p. 738)*
- *Table 185: \*GET Preprocessing Items, Entity = PART (p. 741)*
- *Table 184: \*GET Preprocessing Items, Entity = PIPE (p. 740)*
- *Table 186: \*GET Preprocessing Items, Entity = RCON (p. 741)*
- *Table 187: \*GET Preprocessing Items, Entity = REIN (p. 742)*
- *Table 188: \*GET Preprocessing Items, Entity = SCTN (p. 742)*
- *Table 189: \*GET Preprocessing Items, Entity = SECP (p. 743)*
- *Table 190: \*GET Preprocessing Items, Entity = SHEL (p. 744)*
- *Table 191: \*GET Preprocessing Items, Entity = TBFT (p. 745)*
- *Table 192: \*GET Preprocessing Items, Entity = TBLAB (p. 746)*
- *Table 193: \*GET Preprocessing Items, Entity = VOLU (p. 746)*

- Table 194: \*GET Preprocessing Items, Entity = WELD (p. 747)

**Table 155 \*GET Preprocessing Items, Entity = ACTIVE**

<i>Entity = ACTIVE, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, ACTIVE, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
SEG		Segment capability of graphics driver: 0=no segments available, 1=erasable segments available, 2=non-erasable segments available.
CSYS		Active coordinate system.
DSYS		Active display coordinate system.
MAT		Active material.
TYPE		Active element type.
REAL		Active real constant set.
ESYS		Active element coordinate system.
SECT		Active section.
CP		Maximum coupled node set number in the model (includes merged and deleted sets until compressed out).
CE		Maximum constraint equation set number in the model (includes merged and deleted sets until compressed out).
WFRONT	MAX, RMS	Current maximum or RMS wavefront. Zero if no reordering done.

**Table 156 \*GET Preprocessing items, Entity = AREA**

<i>Entity = AREA, ENTNUM = N (area number)</i>		
<b>*GET, Par, AREA, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
ATTR	<i>Name</i>	Number assigned to the attribute, <i>Name</i> , where <i>Name</i> =MAT, TYPE, REAL, ESYS, KB,KE,SECN, NNOD, NELM, or ESIZ. (NNOD=number of nodes, NELM=number of elements, ESIZ=element size.)
ASEL		Select status of area <i>N</i> : -1=unselected, 0=undefined, 1=selected. Alternative get function: ASEL( <i>N</i> ).
NXTH		Next higher area number above <i>N</i> in selected set (or zero if none found).
NXTL		Next lower area number below <i>N</i> in selected set (or zero if none found).
AREA		Area of area <i>N</i> . ( <b>ASUM</b> or <b>GSUM</b> must have been performed sometime previously with at least this area <i>N</i> selected).
LOOP	1,2,...,I	<i>Item2</i> : LINE, <i>IT2NUM</i> : 1,2,...,p Line number of position <i>p</i> of loop <i>I</i>



<i>Entity = AREA, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, AREA, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
NUM	MAX, MIN	Highest or lowest area number in the selected set.
NUM	MAXD, MIND	Highest or lowest area number defined.
COUNT		Number of areas in the selected set.
AREA		Combined areas (from last <b>ASUM</b> or <b>GSUM</b> ).
VOLU		Combined volume of areas (from last <b>ASUM</b> or <b>GSUM</b> . For 3-D area elements, thickness is determined from area attributes [ <b>AATT</b> ]. For 2-D elements, area attributes are ignored and unit thickness is assumed.
CENT	X, Y, Z	Centroid X, Y, or Z location of areas (from last <b>ASUM</b> or <b>GSUM</b> ).
IOR	X, Y, Z, XY, YZ, ZX	Moments of inertia about origin (from last <b>ASUM</b> or <b>GSUM</b> ).
IMC	X, Y, Z, XY, YZ, ZX	Moments of inertia about mass centroid (from last <b>ASUM</b> or <b>GSUM</b> ).
IPR	X, Y, Z	Principal moments of inertia (from last <b>ASUM</b> or <b>GSUM</b> ).
IXV	X, Y, Z	Principal orientation X vector components (from last <b>ASUM</b> or <b>GSUM</b> ).
IYV	X, Y, Z	Principal orientation Y vector components (from last <b>ASUM</b> or <b>GSUM</b> ).
IZV	X, Y, Z	Principal orientation Z vector components (from last <b>ASUM</b> or <b>GSUM</b> ).

**Table 157 \*GET Preprocessing Items, Entity = AXIS**

<i>Entity = AXIS, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, AXIS, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
COUNT	--	Number of defined sections.
NUM	MAX	Largest section number defined.
<i>Entity = AXIS, ENTNUM = ID (axis section identifier)</i>		
<b>*GET, Par, AXIS, ID, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
TYPE	--	Section type, for <i>ID</i> -- <b>SECTYPE</b> command (always <b>AXIS</b> for axis sections).
NAME	--	Name defined for the given section ID number.

<i>Entity</i> = <b>AXIS</b> , <i>ENTNUM</i> = <i>ID</i> (axis section identifier)		
<b>*GET</b> , <i>Par</i> , <b>AXIS</b> , <i>ID</i> , <i>Item1</i> , <i>IT1NUM</i> , <i>Item2</i> , <i>IT2NUM</i>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
DATA	<i>nnn</i>	Where <i>nnn</i> is the location in the <b>SECDATA</b> command for the given section ID number.

**Table 158 \*GET Preprocessing Items, Entity = CDSY**

<i>Entity</i> = <b>CDSY</b> , <i>ENTNUM</i> = <i>N</i> (coordinate system number)		
<b>*GET</b> , <i>Par</i> , <b>CDSY</b> , <i>N</i> , <i>Item1</i> , <i>IT1NUM</i> , <i>Item2</i> , <i>IT2NUM</i>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
LOC	X, Y, Z	X, Y, or Z origin location in global Cartesian system.
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle (in degrees) relative to the global Cartesian coordinate system.
ATTR	<i>Name</i>	Number assigned to <i>Name</i> , where <i>Name</i> =KCS, KTHET, KPHI, PAR1, or PAR2. The value -1.0 is returned for KCS if the coordinate system is undefined.
NUM	MAX	The maximum coordinate system number

**Table 159 \*GET Preprocessing Items, Entity = CE**

<i>Entity</i> = <b>CE</b> , <i>ENTNUM</i> = <i>N</i> (constraint equation set)		
<b>*GET</b> , <i>Par</i> , <b>CE</b> , <i>N</i> , <i>Item1</i> , <i>IT1NUM</i> , <i>Item2</i> , <i>IT2NUM</i>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
If <i>N</i> = 0, then		
MAX		Maximum constraint equation number
NUM		Number of constraint equations
If <i>N</i> > 0, then		
NTERM		Number of terms in this constraint equation
CONST		Constant term for this constraint equation
TERM	number	Item2 = NODE: Gives the node for this position in the constraint equation.  Item2 = DOF: Gives the DOF number for this position in the constraint equation. (1-UX, 2-UY, 3-UZ, 4-ROTX, etc.)

<i>Entity = CE, ENTNUM = N (constraint equation set)</i>		
<b>*GET, Par, CE, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
		Item2 = COEF: Gives the coefficient for this position in the constraint equation.

**Table 160 \*GET Preprocessing Items, Entity = CMPB**

<i>Entity = CMPB, ENTNUM = N (composite beam section identification number)</i>		
<b>*GET, Par, CMPB, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
COUNT		Number of defined sections. If Item1 = COUNT, then N is blank.
NUM	MAX	Largest section number defined. If IT1NUM = MAX, then N is blank.
EXIS		Returns a 1 if the section exists and if it is a CMPB section.
NAME		The 8-character section name defined via the <b>SECTYPE</b> command.
One of the following:  CBMX CBTE CBMD		Item2 = NTEM (the number of temperatures for <b>CBMX</b> , <b>CBTE</b> , or <b>CBMD</b> data).
One of the following:  CBMX CBTE CBMD		Item2 = TVAL; IT2NUM = <i>nnn</i>  where <i>nnn</i> is the temperature value ( $\leq$ NTEM).
One of the following:  CBMX CBTE CBMD	<i>nnn</i>	Item2 = TEMP; IT2NUM = <i>tval</i>  Where <i>nnn</i> is the location in the <b>CBMX</b> , <b>CBTE</b> , or <b>CBMD</b> command for the given coefficient number, and <i>tval</i> is the temperature value.

**Table 161 \*GET Preprocessing Items, Entity = CP**

<i>Entity = CP, ENTNUM = N (coupled node set)</i>		
<b>*GET, Par, CP, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
If N = 0, then		
MAX		Maximum coupled set number
NUM		Number of coupled sets
If N > 0, then		
DOF		The degree of freedom for this set (1–UX, 2–UY, 3–UZ, 4–ROTX, etc.)

<i>Entity = CP, ENTNUM = N (coupled node set)</i>		
<b>*GET, Par, CP, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NTERM		Number of nodes in this set.
TERM	number	Item2 = NODE: Gives the node for this position number in the coupled set.

**Table 162 \*GET Preprocessing Items, Entity = CSEC**

<i>Entity = CSEC, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, CSEC, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
COUNT	--	Number of defined sections.
NUM	MAX	Largest section number defined.
<i>Entity = CSEC, ENTNUM = ID (contact section identifier)</i>		
<b>*GET, Par, CSEC, ID, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
TYPE	--	Section type, for <i>ID</i> -- <b>SECTYPE</b> command (always CONT for contact sections).
NAME	--	Name defined for the given section ID number.
DATA	<i>nnn</i>	Where <i>nnn</i> is the location in the <b>SECDATA</b> command for the given section ID number.

**Table 163 \*GET Preprocessing Items, Entity = EDCC**

<i>Entity = EDCC, ENTNUM = N (contact entity number, obtained by issuing the EDCLIST command)</i>		
<b>*GET, Par, EDCC, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
COMP	1, 2	Component name for the contact (1) or target (2) surface of contact entity <i>N</i> . A character parameter is returned.
PART	1, 2	PART number for contact (1) or target (2) surface of contact entity <i>N</i> .

<i>Entity = EDCC, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, EDCC, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
COUNT		Total number of contact definitions.

**Table 164 \*GET Preprocessing Items, Entity = ELEM**

<i>Entity = ELEM, ENTNUM = N (element number)</i>		
<b>*GET, Par, ELEM, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
NODE	1, 2, ... 20	Node number at position 1,2,... or 20 of element $N$ . Alternative get function: NELEM( $n, npos$ ), where $npos$ is 1,2,...20.
CENT	X, Y, Z	Centroid X, Y, or Z location (based on shape function) in the active coordinate system. The original locations is used even if large deflections are active. Alternative get functions: CENTRX( $N$ ), CENTRY( $N$ ), and CENTRZ( $N$ ) always retrieve the element centroid in global Cartesian coordinates, and are determined from the selected nodes on the elements.
ADJ	1, 2, ... 6	Element number adjacent to face 1,2,...6. Alternative get function: ELADJ( $N, face$ ). Only elements (of the same dimensionality) adjacent to lateral faces are considered.
ATTR	<i>Name</i>	Number assigned to the attribute <i>Name</i> , where <i>Name</i> = MAT, TYPE, REAL, ESYS, PSTAT, LIVE, or SECN. Returns a zero if the element is unselected. If <i>Name</i> = LIVE, returns a 1 if the element is selected and active, and a -1 if it is selected and inactive. <i>Name</i> = SECN returns the section number of the selected beam element.
LENG		Length of line element (straight line between ends).
LPROJ	X, Y, Z	Projected line element length (in the active coordinate system). X is x-projection onto y-z plane, Y is y projection onto z-x plane, and Z is z-projection onto x-y plane.
AREA		Area of area element.
APROJ	X, Y, Z	Projected area of area element area (in the active coordinate system). X is x-projection onto y-z plane, Y is y projection onto z-x plane, and Z is z-projection onto x-y plane.
VOLU		Element volume. Based on unit thickness for 2-D plane elements (unless the thickness option is used) and on the full 360 degrees for 2-D axisymmetric elements.  <b>Note</b>  If results data are in the database, the volume returned is the volume calculated during solution.
ESEL		Select status of element $N$ : -1 = unselected, 0 = undefined, 1 = selected. Alternative get function: ESEL( $N$ ).

<i>Entity</i> = ELEM, <i>ENTNUM</i> = <i>N</i> (element number)		
<b>*GET, Par, ELEM, <i>N</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NXTH		Next higher element number above <i>N</i> in selected set (or zero if none found). Alternative get function: ELNEXT( <i>N</i> )
NXTL		Next lower element number below <i>N</i> in selected set (or zero if none found).
HGEN		Heat generation on selected element <i>N</i> .
HCOE	face	Heat coefficient for selected element <i>N</i> on specified face. Returns the value at the first node that forms the face.
TBULK	face	Bulk temperature for selected element <i>N</i> on specified face. Returns the value at the first node that forms the face.
PRES	face	Pressure on selected element, <i>N</i> on specified face. Returns the value at the first node that forms the face.
SHPAR	<i>Test</i>	Element shape test result for selected element <i>N</i> , where <i>Test</i> = ANGD (SHELL28 corner angle deviation), ASPE (aspect ratio), JACR (Jacobian ratio), MAXA (maximum corner angle), PARA (deviation from parallelism of opposite edges), or WARP (warping factor).
<i>Entity</i> = ELEM, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, ELEM, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NUM	MAX,MIN	Highest or lowest element number in the selected set.
NUM	MAXD, MIND	Highest or lowest element number defined.
COUNT		Number of elements in the selected set.

Table 165 \*GET Preprocessing Items, Entity = ETYP

<i>Entity</i> = ETYP, <i>ENTNUM</i> = <i>N</i> (element type number)		
<b>*GET, Par, ETYP, <i>N</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
ATTR	<i>Name</i>	Number assigned to the attribute <i>Name</i> , where <i>Name</i> =ENAM, KOP1, KOP2, ..., KOP9, KO10, KO11, etc.

<i>Entity</i> = ETYP, <i>ENTNUM</i> = 0 (or blank)		
*GET, <i>Par</i> , ETYP, 0, <i>Item1</i> , IT1NUM, <i>Item2</i> , IT2NUM		
Item1	IT1NUM	Description
NUM	MAX	Maximum element type.

**Table 166 \*GET Preprocessing Items, Entity = FLDATA**

<i>Entity</i> = FLDATA, <i>ENTNUM</i> = <i>Name</i> ( <i>Name</i> is a valid label on the <i>Name</i> field of the FLDATA command.)		
<p>The value returned is the numerical value for numeric items, 0 or 1 for logical items (off/on or false/true), and a character parameter for items that require a character string. For example, *GET,X,FLDATA,TERM,PRES returns X=convergence monitor value for pressure [FLDATA3], *GET,X,FLDATA,SOLU,TURB returns X=1 if the turbulence model is ON [FLDATA1], and *GET,X,FLDATA,PROT,DENS returns X='CONSTANT' if density is specified as a constant property type [FLDATA7].</p>		
*GET, <i>Par</i> , FLDATA, <i>Name</i> , <i>Item1</i> , IT1NUM, <i>Item2</i> , IT2NUM		
Item1	IT1NUM	Description
Lab	(Blank)	Value of Lab, where Lab is a valid label from the <i>Label</i> field of the FLDATA command.

**Table 167 \*GET Preprocessing Items, Entity = GENB**

<i>Entity</i> = GENB, <i>ENTNUM</i> = <i>N</i> (nonlinear beam general section identification number)		
*GET, <i>Par</i> , GENB, <i>N</i> , <i>Item1</i> , IT1NUM, <i>Item2</i> , IT2NUM		
Item1	IT1NUM	Description
COUNT	(Blank)	Number of defined sections. If <i>Item1</i> = COUNT, then <i>N</i> is blank.
NUM	MAX	Largest section number defined. If <i>IT1NUM</i> = MAX, then <i>N</i> is blank.
EXIS	(Blank)	Returns a 1 if the section exists <i>and</i> if it is a GENB section.
SUBTYPE	(Blank)	Section subtype for the section ID specified via the SECTYPE command.
NAME	(Blank)	The 8-character section name defined via the SECTYPE command.
<i>One of the following:</i>  BSAX BSM1 BSM2 BSTQ BSS1 BSS2 BSMD BSTE	(Blank)	<i>Item2</i> = NTEM, the number of temperatures for BSAX, BSM1, BSM2, BSTQ, BSS1, BSS2, BSMD, or BSTE data.
<i>One of the following:</i>  BSAX	(Blank)	<i>Item2</i> = TVAL; <i>IT2NUM</i> = <i>nnn</i>  Where <i>nnn</i> is the temperature value (<= NTEM).

<i>Entity</i> = GENB, <i>ENTNUM</i> = <i>N</i> (nonlinear beam general section identification number)		
<b>*GET, Par, GENB, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
BSM1 BSM2 BSTQ BSS1 BSS2 BSMD BSTE		
One of the following:  BSAX BSM1 BSM2 BSTQ BSS1 BSS2 BSMD BSTE	<i>nnn</i>	<i>Item2</i> = TEMP; <i>IT2NUM</i> = <i>tval</i>  Where <i>nnn</i> is the location in the <b>BSAX, BSM1, BSM2, BSTQ, BSS1, BSS2, BSMD, or BSTE</b> command for the given coefficient number, and <i>tval</i> is the temperature value.  Examples for <i>nnn</i> :  <i>nnn</i> = 1 for STRAIN(1) <i>nnn</i> = 2 for STRESS(1) <i>nnn</i> = 3 for STRAIN(2) <i>nnn</i> = 4 for STRESS(2) <i>nnn</i> = 5 for STRAIN(3) ...
One of the following:  BSAX BSM1 BSM2 BSTQ BSS1 BSS2 BSMD BSTE	(Blank)	<i>Item2</i> = TEMP; <i>IT2NUM</i> = <i>tval</i> ; <i>Item3</i> = NCONST  The number of constants at <i>tval</i> .

Table 168 \*GET Preprocessing Items, Entity = GENS

<i>Entity</i> = GENS, <i>ENTNUM</i> = <i>N</i> (preintegrated shell general section identification number)		
<b>*GET, Par, GENS, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
COUNT	(Blank)	Number of defined sections. If <i>Item1</i> = COUNT, then <i>N</i> is blank.
NUM	MAX	Largest section number defined. If <i>IT1NUM</i> = MAX, then <i>N</i> is blank.
EXIS	(Blank)	Returns a 1 if the section exists <i>and</i> if it is a GENS section.
NAME	(Blank)	The 8-character section name defined via the <b>SECTYPE</b> command.
One of the following:	(Blank)	<i>Item2</i> = NTEM, the number of temperatures for <b>SSPA, SSPB, SSPD, SSPE, SSMT, SSBT, or SSPM</b> data.



<i>Entity</i> = GENS, <i>ENTNUM</i> = <i>N</i> (preintegrated shell general section identification number)		
<b>*GET, Par, GENS, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
SSPA SSPB SSPD SSPE SSMT SSBT SSPM		
One of the following:  SSPA SSPB SSPD SSPE SSMT SSBT SSPM	(Blank)	<i>Item2</i> = TVAL; <i>IT2NUM</i> = <i>nnn</i>  Where <i>nnn</i> is the temperature value ( $\leq$ NTEM).
One of the following:  SSPA SSPB SSPD SSPE SSMT SSBT SSPM	<i>nnn</i>	<i>Item2</i> = TEMP; <i>IT2NUM</i> = <i>tval</i>  Where <i>nnn</i> is the location in the <b>SSPA, SSPB, SSPD, SSPE, SSMT, SSBT, or SSPM</b> command for the given coefficient number, and <i>tval</i> is the temperature value.

**Table 169 \*GET Preprocessing Items, Entity = KP**

<i>Entity</i> = KP, <i>ENTNUM</i> = <i>N</i> (keypoint number)		
<b>*GET, Par, KP, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
LOC	X, Y, Z	X, Y, or Z location in the active coordinate system. Alternative get functions: KX( <i>N</i> ), KY( <i>N</i> ), KZ( <i>N</i> ). Inverse get function: KP( <i>x, y, z</i> ) returns the number of the selected keypoint nearest the <i>x, y, z</i> location (in the active coordinate system, lowest number for coincident keypoints).
ATTR	<i>Name</i>	Number assigned to the attribute <i>Name</i> , where <i>Name</i> = MAT, TYPE, REAL, ESYS, NODE, or ELEM.
KSEL		Select status of keypoint <i>N</i> : -1 = unselected, 0 = undefined, 1 = selected. Alternative get function: KSEL( <i>N</i> ).
NXTH		Next higher keypoint number above <i>N</i> in selected set (or zero if none found). Alternative get function: KPNEXT( <i>N</i> ).

<i>Entity</i> = KP, <i>ENTNUM</i> = <i>N</i> (keypoint number)		
<b>*GET, Par, KP, <i>N</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NXTL		Next lower keypoint number below <i>N</i> in selected set (or zero if none found).
DIV		Divisions (element size setting) from <b>KESIZE</b> command.
<i>Entity</i> = KP, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, KP, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NUM	MAX, MIN	Highest or lowest keypoint number in the selected set.
NUM	MAXD, MIND	Highest or lowest keypoint number defined
COUNT		Number of keypoints in the selected set.
CENT	X, Y, Z	Centroid X, Y, or Z location of keypoints (from last <b>KSUM</b> or <b>GSUM</b> ).
IOR	X, Y, Z, XY, YZ, ZX	Moments of inertia about origin (from last <b>KSUM</b> or <b>GSUM</b> ).
IMC	X, Y, Z, XY, YZ, ZX	Moments of inertia about mass centroid (from last <b>KSUM</b> or <b>GSUM</b> ).
IPR	X, Y, Z	Principal moments of inertia (from last <b>KSUM</b> or <b>GSUM</b> ).
IXV	X, Y, Z	Principal orientation X vector components (from last <b>KSUM</b> or <b>GSUM</b> ).
IYV	X, Y, Z	Principal orientation Y vector components (from last <b>KSUM</b> or <b>GSUM</b> ).
IZV	X, Y, Z	Principal orientation Z vector components (from last <b>KSUM</b> or <b>GSUM</b> ).
MXLOC	X, Y, Z	Maximum X, Y, or Z keypoint coordinate in the selected set (in the active coordinate system).
MNLOC	X, Y, Z	Minimum X, Y, or Z keypoint coordinate in the selected set (in the active coordinate system).
NRELM	m	Keypoint number of meshed region nearest centroid of element <i>m</i> .

Table 170 \*GET Preprocessing Items, Entity = LINE

<i>Entity</i> = LINE, <i>ENTNUM</i> = <i>N</i> (line number)		
<b>*GET, Par, LINE, <i>N</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
KP	1,2	Keypoint number at position 1 or 2.
ATTR	<i>Name</i>	Number assigned to the attribute, <i>Name</i> , where <i>Name</i> =MAT, TYPE, REAL, ESYS, NNOD, NELM, NDIV, NDNX, SPAC, SPNX, KYND, KYSP, LAY1, or LAY2. (NNOD=number of nodes, returns --1 for meshed line with no internal nodes, NELM=number of elements, NDIV=number of divisions in an existing mesh, NDNX=number of

<b>Entity = LINE, ENTNUM = N (line number)</b>		
<b>*GET, Par, LINE, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
		divisions assigned for next mesh, SPAC=spacing ratio in an existing mesh, SPNX=spacing ratio for next mesh, KYND=soft key for NDNX, KYSP=soft key for SPNX, LAY1=LAYER1 setting, LAY2=LAYER2 setting.)
LSEL		Select status of line <i>N</i> : -1=unselected, 0=undefined, 1=selected. Alternative get function: LSEL( <i>N</i> ).
NXTH		Next higher line number above <i>N</i> in the selected set (or zero if none found). Alternative get function: LSNEXT( <i>N</i> )
NXTL		Next lower line number below <i>N</i> in selected set (or zero if none found).
LENG		Length. A get function LX( <i>n</i> , <i>frac</i> ) also exists to return the X coordinate location of line <i>N</i> at the length fraction <i>frac</i> (0.0 to 1.0). Similar LY and LZ functions exist.
<b>Entity = LINE, ENTNUM = 0 (or blank)</b>		
<b>*GET, Par, LINE, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
NUM	MAX, MIN	Highest or lowest line number in the selected set.
NUM	MIND, MAXD	Highest or lowest line number defined.
COUNT		Number of lines in the selected set.
LENG		Combined length of lines (from last <b>LSUM</b> or <b>GSUM</b> ).
CENT	X, Y, Z	Centroid X, Y, or Z location of lines (from last <b>LSUM</b> or <b>GSUM</b> ).
IOR	X, Y, Z, XY, YZ, ZX	Moments of inertia about origin (from last <b>LSUM</b> or <b>GSUM</b> ).
IMC	X, Y, Z, XY, YZ, ZX	Moments of inertia about mass centroid (from last <b>LSUM</b> or <b>GSUM</b> ).
IPR	X, Y, Z	Principal moments of inertia (from last <b>LSUM</b> or <b>GSUM</b> ).
IXV	X, Y, Z	Principal orientation X vector components (from last <b>LSUM</b> or <b>GSUM</b> ).
IYV	X, Y, Z	Principal orientation Y vector components (from last <b>LSUM</b> or <b>GSUM</b> ).
IZV	X, Y, Z	Principal orientation Z vector components (from last <b>LSUM</b> or <b>GSUM</b> ).

**Table 171 \*GET Preprocessing Items, Entity = MAT**

<b>Entity = MAT, ENTNUM = 0 (or BLANK)</b>		
<b>*GET, Par, MAT, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
COUNT		Number of materials.

<i>Entity</i> = MAT, <i>ENTNUM</i> = 0 (or BLANK)		
<b>*GET</b> , <i>Par</i> , MAT, 0, Item1, IT1NUM, Item2, IT2NUM		
Item1	IT1NUM	Description
NUM	MAX	Largest material number defined.

Table 172 \*GET Preprocessing Items, Entity = MPLAB

<i>Entity</i> = MPLab, <i>ENTNUM</i> = <i>N</i> ( <i>MPLab</i> = material property label from <b>MP</b> command; <i>N</i> = material number.)		
<b>*GET</b> , <i>Par</i> , MPLab, <i>N</i> , Item1, IT1NUM, Item2, IT2NUM		
Item1	IT1NUM	Description
TEMP	val	Material property value at temperature of <i>val</i> . For temperature dependent materials, the program interpolates the property at temperature input for <i>val</i> .

Table 173 \*GET Preprocessing Items, Entity = MSCAP

<i>Entity</i> = MSCAP, <i>ENTNUM</i> = <i>n</i> (species number)		
<b>*GET</b> , <i>Par</i> , MSCAP, <i>n</i> , Item1, IT1NUM, Item2, IT2NUM		
Item1	IT1NUM	Description
KEY		Status of mass fraction capping for species <i>n</i> : 0=off, 1=on.
UPP		Upper bound of mass fraction.
LOW		Lower bound of mass fraction.

Table 174 \*GET Preprocessing Items, Entity = MSDATA

<i>Entity</i> = MSDATA, <i>ENTNUM</i> = 0		
<b>*GET</b> , <i>Par</i> , MSDATA, 0, Item1, IT1NUM, Item2, IT2NUM		
Item1	IT1NUM	Description
SPEC		The algebraic species number.

<i>Entity = MSDATA, ENTNUM = 0</i>		
<b>*GET, Par, MSDATA, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
UGAS		Value of the universal gas constant.

**Table 175 \*GET Preprocessing Items, Entity = MSMETH**

<i>Entity = MSMETH, ENTNUM = n (species number)</i>		
<b>*GET, Par, MSMETH, n, Item1, IT1NUM, Item2, IT2NUM</b>		
(Blank)	(Blank)	Solution method for species <i>n</i> : 0=no solution, 1=TDMA method, 2=conjugate residual method, 3=preconditioned conjugate residual method.

**Table 176 \*GET Preprocessing Items, Entity = MSNOMF**

<i>Entity = MSNOMF, ENTNUM = n (species number)</i>		
<b>*GET, Par, MSNOMF, n, Item1, IT1NUM, Item2, IT2NUM</b>		
(Blank)	(Blank)	Initial mass fraction of species <i>N</i> .

**Table 177 \*GET Preprocessing Items, Entity = MSPROP**

<i>Entity = MSPROP, ENTNUM = n (species number)</i>		
<b>*GET, Par, MSPROP, n, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
<i>Lab</i>	TYPE	The type of fluid property variation being used for <i>Lab</i> , where <i>Lab</i> is a valid property label as described on the <b>MSPROP</b> command (DENS, VISC, COND, etc.). A character parameter is returned: CONSTANT, GAS, LIQUID, or a property name from the floprp.ans file.
"	NOMI	Value of property <i>Lab</i> : nominal value for a CONSTANT fluid property, value at temperature given by <i>COF1</i> for other property types.
"	COF1, COF2, COF3	Coefficients in the equation of state for property <i>Lab</i> .

**Table 178 \*GET Preprocessing Items, Entity = MSRELAX**

<i>Entity = MSRELAX, ENTNUM = n (species number)</i>		
<b>*GET, Par, MSRELAX, n, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
CONC		Mass fraction concentration relaxation factor.
MDIF		Mass diffusion coefficient relaxation factor.
EMDI		Effective mass diffusion coefficient relaxation factor.

<i>Entity</i> = MSRELAX, <i>ENTNUM</i> = <i>n</i> (species number)		
<b>*GET, Par, MSRELAX, <i>n</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
STAB		Transport equation inertial relaxation factor.

**Table 179 \*GET Preprocessing Items, Entity = MSSOLU**

<i>Entity</i> = MSSOLU, <i>ENTNUM</i> = <i>n</i> (species number)		
<b>*GET, Par, MSSOLU, <i>n</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NSWE		Number of TDMA sweeps.
MAXI		Maximum number of iterations allowed for semi-direct methods.
NSRC		Number of search vectors used for semi-direct methods.
CONV		Convergence criterion for semi-direct methods.
DELT		Maximum normalized rate of change which will permit the semi-direct solution to continue.

**Table 180 \*GET Preprocessing Items, Entity = MSSPEC**

<i>Entity</i> = MSSPEC, <i>ENTNUM</i> = <i>n</i> (species number)		
<b>*GET, Par, MSSPEC, <i>n</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NAME		Name of species <i>n</i> . A character parameter is returned.
MOLW		Molecular weight of species <i>n</i> .
SCHM		Turbulent Schmidt number of species <i>n</i> .

**Table 181 \*GET Preprocessing Items, Entity = MSVARY**

<i>Entity</i> = MSVARY, <i>ENTNUM</i> = <i>n</i> (species number)		
<b>*GET, Par, MSVARY, <i>n</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
<i>Lab</i>		Variability status of property <i>Lab</i> (where <i>Lab</i> =DENS, VISC, COND, or MDIF): 0=off, 1=on.

**Table 182 \*GET Preprocessing Items, Entity = NODE**

<i>Entity</i> = NODE, <i>ENTNUM</i> = <i>N</i> (node number)		
<b>*GET, Par, NODE, <i>N</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
LOC	X, Y, Z	X, Y, Z location in the active coordinate system. Alternative get functions: NX( <i>N</i> ), NY( <i>N</i> ), NZ( <i>N</i> ). Inverse get function. NODE( <i>x</i> , <i>y</i> , <i>z</i> ) returns the number of the selected node nearest the <i>x</i> , <i>y</i> , <i>z</i> location (in the active coordinate system, lowest number for coincident nodes).
ANG	XY, YZ, ZX	THXY, THYZ, THZX rotation angle.

<b>Entity = NODE, ENTNUM = N (node number)</b>		
<b>*GET, Par, NODE, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
NSEL		Select status of node <i>N</i> : -1=unselected, 0=undefined, 1=selected. Alternative get function: NSEL( <i>N</i> ).
NXTH		Next higher node number above <i>N</i> in selected set (or zero if none found). Alternative get function: NDNEXT( <i>N</i> ).
NXTL		Next lower node number below <i>N</i> in selected set (or zero if none found).
F	FX, MX, ...	Applied force at selected node <i>N</i> in direction <i>IT1NUM</i> (returns 0.0 if no force is defined, if node is unselected, or if the DOF is inactive). If <i>ITEM2</i> is IMAG, return the imaginary part.
D	UX, ROTX, ...	Applied constraint force at selected node <i>N</i> in direction <i>IT1NUM</i> (returns a large number, such as 2e100, if no constraint is specified, if the node is unselected, or if the DOF is inactive). If <i>ITEM2</i> is IMAG, return the imaginary part.
HGEN		Heat generation on selected node <i>N</i> (returns 0.0 if node is unselected, or if the DOF is inactive).
NTEMP		Temperature on selected node <i>N</i> (returns 0.0 if node is unselected)
CPS	Lab	Couple set number with direction Lab = any active DOF, which contains the node <i>N</i> .
<b>Entity = NODE, ENTNUM = 0 (or blank)</b>		
<b>*GET, Par, NODE, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
NUM	MAX, MIN	Highest or lowest node number in the selected set.
NUM	MAXD, MIND	Highest or lowest node number defined.
COUNT		Number of nodes in the selected set.
MXLOC	X, Y, Z	Maximum X, Y, or Z node coordinate in the selected set (in the active coordinate system).
MNLOC	X, Y, Z	Minimum X, Y, or Z node coordinate in the selected set (in the active coordinate system).
<p><b>Note:</b> If ANSYS creates internal nodes during solution, the internal nodes will not be included. You can include them by using <i>KINTERNAL</i>, a seventh <b>*GET</b> command argument specific to <i>Entity = NODE</i>. The command syntax is:</p>		
<b>*GET, Par, NODE, 0, Item1, IT1NUM, Item2, IT2NUM, KINTERNAL</b>		

<i>Entity</i> = NODE, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, NODE, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
The options for the <i>KINTERNAL</i> key are ( <i>blank</i> ), - count only external nodes, and <i>INTERNAL</i> , - count all nodes, including internal nodes.		

**Table 183 \*GET Preprocessing Items, Entity = OCEAN**

<i>Entity</i> = OCEAN, <i>ENTNUM</i> = <i>N</i> (ocean identification number)		
<b>*GET, Par, OCEAN, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
SUBTYPE		Subtype for ID ( <b>OCTYPE</b> command)
NAME		Name defined for a given id number.
DATA	1	Depth when Subtype = BASI
	2	Material ID when Subtype = BASI
	3	Ocean current ID when Subtype = BASI
	4	Ocean wave ID when Subtype = BASI
	8	KFLOOD when Subtype = BASI
	9	Ci when Subtype = BASI
	10	Cb when Subtype = BASI
	1	KWAVE when Subtype = WAVE
	2	THETA when Subtype = WAVE
	3	WAVELOC when Subtype = WAVE
	4	KCRC when Subtype = WAVE
	5	KMF when Subtype = WAVE
	6	PRKEY when Subtype = WAVE
	PROP	NROW
TABL	<i>i</i>	Data in table defined by <b>OCTABLE</b> command <i>i</i> = row number; Item2 = column number

**Table 184 \*GET Preprocessing Items, Entity = PIPE**

<i>Entity</i> = PIPE, <i>ENTNUM</i> = 0 (or BLANK)		
<b>*GET, Par, PIPE, NUM, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
COUNT		Number of defined sections
NUM	MAX	Largest section number defined



<b>Entity = PIPE, ENTNUM = id (pipe section identification number)</b>		
<b>*GET, Par, PIPE, id, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
TYPE		Section type, for id - <b>SECTYPE</b> command (always PIPE for pipe sections)
SUBTYPE		Section type for id - <b>SECTYPE</b> command
NAME		Name defined for the given section id number
DATA	<i>nnn</i>	Where <i>nnn</i> is the location in the <b>SECDATA</b> command for the given section id number
SOCEAN	<i>nnn</i>	The ID number of the ocean environment surrounding the given section ID number
SFLEX	<i>nnn</i>	Where <i>nnn</i> is the location in the <b>SFLEX</b> command for the given section ID number
PROP	AREA	Area value
	IYY, IYZ, IZZ	Moments of inertia
	TORS	Torsion constant
	SCYY, SCYZ, SCZZ	Shear correction factors
	OFFY	Section offset in the Y-direction.
	OFFZ	Section offset in the Z-direction.
	ADDMAS	Added mass per unit length

**Table 185 \*GET Preprocessing Items, Entity = PART**

<b>Entity = PART, ENTNUM = N (PART number)</b>		
<b>*GET, Par, PART, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
TYPE		Element type number assigned to PART <i>N</i> .
MAT		Material number assigned to PART <i>N</i> .
REAL		Real constant number assigned to PART <i>N</i> .
<b>Entity = PART, ENTNUM = 0 (or blank)</b>		
<b>*GET, Par, PART, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
NUMP		Total number of parts in the model.

**Table 186 \*GET Preprocessing Items, Entity = RCON**

<b>Entity = RCON, ENTNUM = N (real constant set number)</b>		
<b>*GET, Par, RCON, N, Item1, IT1NUM, Item2, IT2NUM</b>		
CONST	1, 2, ..., <i>m</i>	Value of real constant number <i>m</i> in set <i>N</i> .
<b>*GET, Par, RCON, 0, Item1, IT1NUM, Item2, IT2NUM</b>		

<i>Entity</i> = <b>RCON</b> , <i>ENTNUM</i> = <i>N</i> (real constant set number)		
<b>*GET</b> , <i>Par</i> , <b>RCON</b> , <i>N</i> , <i>Item1</i> , <b>IT1NUM</b> , <i>Item2</i> , <b>IT2NUM</b>		
NUM	MAX	The maximum real constant set number defined

**Table 187 \*GET Preprocessing Items, Entity = REIN**

<i>Entity</i> = <b>REIN</b> , <i>ENTNUM</i> = <i>N</i> (reinforcing section identification number)		
<b>*GET</b> , <i>Par</i> , <b>REIN</b> , <i>N</i> , <i>Item1</i> , <b>IT1NUM</b> , <i>Item2</i> , <b>IT2NUM</b>		
Item1	IT1NUM	Description
TYPE		Section type, for ID -- <b>SECTYPE</b> command (always REIN for reinforcing sections).
SUBTYPE		Section subtype for ID -- <b>SECTYPE</b> command.
NAME		Name defined for a given ID number.
NREIN		Number of reinforcing fibers.
TABL	<i>ReinfNumber, I</i>	Reinforcing fiber data, as defined by the <b>SECDATA</b> command.

**Table 188 \*GET Preprocessing Items, Entity = SCTN**

<i>Entity</i> = <b>SCTN</b> , <i>ENTNUM</i> = <i>N</i> (pretension section ID number)		
<b>*GET</b> , <i>Par</i> , <b>SCTN</b> , <i>N</i> , <i>Item1</i> , <b>IT1NUM</b> , <i>Item2</i> , <b>IT2NUM</b>		
Item1	IT1NUM	Description
1		Section ID number.
2		Section type (always 5 for pretension section).
3		Pretension node number.
4	Coordinate system number.	Section normal NX.
5	Coordinate system number.	Section normal NY.
6	Coordinate system number.	Section normal NZ.
7 or 8		Eight character section name.
9		Initial action key. Returns 0 or 1 for lock, 2 for "free-to-slide," or 3 for tiny.
10		Force displacement key. Returns 0 or 1 for force, or 2 for displacement.
11		First preload value.
12		Load step in which first preload value is to be applied.
13		Load step in which first preload value is to be locked.

<b>Entity = SCTN, ENTNUM = N (pretension section ID number)</b>		
<b>*GET, Par, SCTN, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
14...		14 through 17 is a repeat of 10 through 13, but for the second preload value; 18 through 21 is for the third preload value; and so forth.

**Table 189 \*GET Preprocessing Items, Entity = SECP**

<b>Entity = SECP, ENTNUM = 0 (or BLANK)</b>		
<b>*GET, Par, SECP, NUM, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
COUNT		Number of defined sections
NUM	MAX	Largest section number defined
<b>Entity = SECP, ENTNUM = id (beam section identification number)</b>		
<b>*GET, Par, SECP, id, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
TYPE		Section type, for id - <b>SECTYPE</b> command (always BEAM for beam sections)
SUBTYPE		Section type for id - <b>SECTYPE</b> command
NAME		Name defined for the given section id number
DATA	<i>nnn</i>	Where <i>nnn</i> is the location in the <b>SECDATA</b> command for the given section id number
PROP	AREA	Area value
	IYY, IYZ, IZZ	Moments of inertia
	WARP	Warping constant
	TORS	Torsion constant
	CGY, CGZ	Y or Z coordinate center of gravity
	SHCY, SHCZ	Y or Z coordinate shear center
	SCYY, SCYZ, SCZZ	Shear correction factors
	OFFSET	Offset location: 1 = Centroid 2 = Shear Center 3 = Origin 0 = User Defined
	OFFY	Section offset in the Y-direction.

<i>Entity</i> = SECP, <i>ENTNUM</i> = <i>id</i> (beam section identification number)		
*GET, <i>Par</i> , SECP, <i>id</i> , <i>Item1</i> , <i>IT1NUM</i> , <i>Item2</i> , <i>IT2NUM</i>		
Item1	IT1NUM	Description
	OFFZ	Section offset in the Z-direction.

Table 190 \*GET Preprocessing Items, Entity = SHEL

<i>Entity</i> = SHEL, <i>ENTNUM</i> = <i>N</i> (shell section identification number)		
*GET, <i>Par</i> , SHEL, <i>N</i> , <i>Item1</i> , <i>IT1NUM</i> , <i>Item2</i> , <i>IT2NUM</i>		
Item1	IT1NUM	Description
TYPE		Section type, for <i>id</i> — <b>SECTYPE</b> command. (always SHEL for shell sections)
NAME		Name defined for a given <i>id</i> number.
PROP	TTHK	Total thickness.
"	NLAY	Number of layers.
"	NSP	Number of section integration points.
"	POS	Node position (as defined by <b>SECOFFSET</b> ).
"	"	0 = User Defined.
"	"	1 = Middle.
"	"	2 = Top.
"	"	3 = Bottom.
"	OFFZ	User-defined section offset (POS = 0).
"	TS11	Transverse shear stiffness factors.
"	TS22	Transverse shear stiffness factors.
"	TS12	Transverse shear stiffness factors.
"	HORC	Homogeneous or complete section flag.
"	"	0 = Homogeneous.
"	"	1 = Composite.
"	FUNC	Tabular function name for total thickness.
"	UT11	User transverse shear stiffness 11.
"	UT22	User transverse shear stiffness 22.
"	UT12	User transverse shear stiffness 12.
"	AMAS	Added mass.
"	MSCF	Hourglass control membrane scale factor.
"	BSCF	Hourglass control bending scale factor.
"	DSTF	Drill stiffness scale factor.
"	LDEN	Laminate density.
"	FKCN	<i>KCN</i> field value from the <b>SECFUNCTION</b> command, in which the array or table is interpreted.
LAYD	LayerNumber, THIC	Layer thickness.

<i>Entity</i> = SHEL, <i>ENTNUM</i> = <i>N</i> (shell section identification number)		
<b>*GET, Par, SHEL, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
"	LayerNumber,MAT	Layer material.
"	LayerNumber,ANGL	Layer orientation angle.
"	LayerNumber,NINT	Number of layer integration points.

**Table 191 \*GET Preprocessing Items, Entity = TBFT**

<i>Entity</i> = TBFT, <i>ENTNUM</i> = <i>BLANK</i>		
<b>*GET, Par, TBFT, , Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
nmat		Number of defined material models.
matnum	<i>index</i>	Material number in array (index varies for 1 to num materials).
<i>Entity</i> = TBFT, <i>ENTNUM</i> = <i>matid</i> (For getting names of constitutive function, <i>matid</i> = the material ID number)		
<b>*GET, Par, TBFT, matid, nfun, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
nfun		Number of constitutive functions for this material.
<i>Entity</i> = TBFT, <i>ENTNUM</i> = <i>matid</i> (To query constitutive function data, <i>matid</i> = the material ID number)		
<b>*GET, Par, TBFT, matid, func, fname, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
func	<i>index</i>	if <i>Item2</i> = <i>fname</i> , the name of the constitutive function is returned.
func	<i>function name</i>	If <i>Item2</i> = <i>ncon</i> , the number of constants is returned for the function specified in <i>IT1NUM</i> by the constitutive function name.
"	"	If <i>Item2</i> = <i>cons</i> , set <i>Item2num</i> to <i>index</i> to return the value of the constant.
"	"	If <i>Item2</i> = <i>fixe</i> , set <i>Item2num</i> to <i>index</i> to return the fix flag status.
"	"	If <i>Item2</i> = <i>RESI</i> , returns the residual error while fitting the data.
"	"	If <i>Item2</i> = <i>type</i> , returns the category of the constitutive model (moon, poly, etc.)
"	"	If <i>Item2</i> = <i>sord</i> , returns the shear order of the prony visco model.
"	"	If <i>Item2</i> = <i>bord</i> , returns the bulk order of the prony visco model.
"	"	If <i>Item2</i> = <i>shif</i> , returns the shift function name of the prony visco model.
<i>Entity</i> = TBFT, <i>ENTNUM</i> = <i>matid</i> (To query experimental data, <i>matid</i> = the material ID number)		
<b>*GET, Par, TBFT, matid, func, fname, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
expe	(blank)	If <i>Item2</i> = <i>nexp</i> , returns number of experiments in a material model.

<i>Entity</i> = TBFT, <i>ENTNUM</i> = BLANK		
*GET, <i>Par</i> , TBFT, , <i>Item1</i> , IT1NUM, <i>Item2</i> , IT2NUM		
Item1	IT1NUM	Description
"	expindex	If <i>Item2</i> = type, returns index of experiment.
"	"	If <i>Item2</i> = numrow, returns number of rows in the data.
"	"	If <i>Item2</i> = numcol, returns the number of cols in a row (set <i>Item2num</i> = Row index)
"	"	If <i>Item2</i> = data, returns the value of the data in <i>row</i> , <i>col</i> of exp <i>expindex</i> (set <i>item2Num</i> = row index and <i>item3</i> = column index. All indices vary from 1 to the maximum value.
"	"	If <i>Item2</i> = natt, returns the number of attributes.
"	"	If <i>Item2</i> = attrname, returns the attribute name (set <i>Item2Num</i> = Attr index).
"	"	If <i>Item2</i> = attvald, returns double value of attribute (set <i>Item2Num</i> = Attr index).
"	"	If <i>Item2</i> = attvali, returns integer valud of attribute (set <i>Item2Num</i> = Attr index).
"	"	If <i>Item2</i> = attvals, returns the string value of the attribute (set <i>Item2Num</i> = Attr index).

Table 192 \*GET Preprocessing Items, Entity = TBLAB

<i>Entity</i> = TBlab, <i>ENTNUM</i> = <i>N</i> . ( <i>TBlab</i> = data table label from the TB command; <i>N</i> = material number.)		
*GET, <i>Par</i> , TBlab, <i>N</i> , <i>Item1</i> , IT1NUM, <i>Item2</i> , IT2NUM, TBOPT		
Item1	IT1NUM	Description
TEMP	<i>T</i>	<b>Item2: CONST IT2NUM:</b> <i>Num</i> Value of constant number <i>Num</i> in the datatable at temperature <i>T</i> (see <a href="#">Data Tables - Implicit Analysis</a> in the <a href="#">Element Reference</a> ). For constants input a X,Y point, the constant numbers are consecutive with the X constants being the odd numbers, beginning with one.

Table 193 \*GET Preprocessing Items, Entity = VOLU

<i>Entity</i> = VOLU, <i>ENTNUM</i> = <i>N</i> (volume number)		
*GET, <i>Par</i> , VOLU, <i>N</i> , <i>Item1</i> , IT1NUM, <i>Item2</i> , IT2NUM		
Item1	IT1NUM	Description
ATTR	<i>Name</i>	Number assigned to the attribute <i>Name</i> , where <i>Name</i> =MAT, TYPE, REAL, ESYS, NNOD, or NELM. (NNOD=number of nodes, NELM=number of elements.)
VSEL		Select status of volume <i>N</i> : -1=unselected, 0=undefined, 1=selected. Alternative get function: VSEL( <i>N</i> ).
NXTH		Next higher volume number above <i>N</i> in selected set (or zero if none found). Alternative get function: VLNEXT( <i>N</i> ).

<i>Entity</i> = VOLU, <i>ENTNUM</i> = <i>N</i> (volume number)		
<b>*GET, Par, VOLU, <i>N</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NXTL		Next lower volume number below <i>N</i> in selected set (or zero if none found).
VOLU		Volume of volume <i>N</i> . ( <b>VSUM</b> or <b>GSUM</b> must have been performed sometime previously with at least this volume <i>N</i> selected).
SHELL	1, 2, ..., <i>m</i>	<b>Item2:</b> AREA <b>IT2NUM:</b> 1,2,..., <i>p</i> Line number of position <i>p</i> of shell <i>m</i>
<i>Entity</i> = VOLU, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, VOLU, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
NUM	MAX, MIN	Highest or lowest volume number in the selected set.
NUM	MAXD, MIND	Highest or lowest volume number defined.
COUNT		Number of volumes in the selected set.
VOLU		Combined volumes (from last <b>VSUM</b> or <b>GSUM</b> ).
CENT	X, Y, Z	Centroid X, Y, or Z location of volumes (from last <b>VSUM</b> or <b>GSUM</b> ).
IOR	X, Y, Z, XY, YZ, ZX	Moments of inertia about origin (from last <b>VSUM</b> or <b>GSUM</b> ).
IMC	X, Y, Z, XY, YZ, ZX	Moments of inertia about mass centroid (from last <b>VSUM</b> or <b>GSUM</b> ).
IPR	X, Y, Z	Principal moments of inertia (from last <b>VSUM</b> or <b>GSUM</b> ).
IXV	X, Y, Z	Principal orientation X vector components (from last <b>VSUM</b> or <b>GSUM</b> ).
IYV	X, Y, Z	Principal orientation Y vector components (from last <b>VSUM</b> or <b>GSUM</b> ).
IZV	X, Y, Z	Principal orientation Z vector components (from last <b>VSUM</b> or <b>GSUM</b> ).

Table 194 \*GET Preprocessing Items, Entity = WELD

<i>Entity</i> = WELD, <i>ENTNUM</i> = <i>N</i> (weld number)		
<b>*GET, Par, WELD, <i>N</i>, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NODE	1, 2	First or second node number for spot weld <i>N</i> .
NXTH		Next higher spotweld number above <i>N</i> (or 0 if none found).
<i>Entity</i> = WELD, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, WELD, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NUM	MAX, MIN	Highest or lowest spotweld number.
COUNT		Total number of spotwelds in model.

## Solution Items

### \*GET Solution Entity Items

- Table 195: \*GET Solution Items, Entity = ACTIVE (p. 748)
- Table 196: \*GET Solution Items, Entity = ELEM (p. 749)
- Table 197: \*GET Solution Items, Entity = MODE (p. 749)

**Table 195 \*GET Solution Items, Entity = ACTIVE**

<i>Entity = ACTIVE, ENTNUM = 0 (or blank)</i>		
<i>*GET, Par, ACTIVE, 0, Item1, IT1NUM, Item2, IT2NUM</i>		
Item1	IT1NUM	Description
ANTY		Current analysis type.
SOLU	DTIME	Time step size.
"	NCMLS	Cumulative number of load steps.
"	NCMSS	Number of substeps. NOTE: Used only for static and transient analyses.
"	EQIT	Number of equilibrium iterations.
"	NCMIT	Cumulative number of iterations.
"	CNVG	Convergence indicator: 0=not converged, 1=converged.
"	MXDVL	Maximum degree of freedom value.
"	RESFRQ	Response frequency for 2nd order systems.
"	RESEIG	Response eigenvalue for 1st order systems.
"	DSPRM	Descent parameter.
"	FOCV	Force convergence value.
"	MOCV	Moment convergence value.
"	HFCV	Heat flow convergence value.
"	MFCV	Magnetic flux convergence value.
"	CSCV	Current segment convergence value.
"	CUCV	Current convergence value.
"	FFCV	Fluid flow convergence value.
"	DICV	Displacement convergence value.
"	ROCV	Rotation convergence value.
"	TECV	Temperature convergence value.
"	VMCV	Vector magnetic potential convergence value.
"	SMCV	Scalar magnetic potential convergence value.
"	VOCV	Voltage convergence value.
"	PRCV	Pressure convergence value.
"	VECV	Velocity convergence value.
"	CRPRAT	Maximum creep ratio.



<i>Entity</i> = ACTIVE, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, ACTIVE, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
"	PSINC	Maximum plastic strain increment.
"	CGITER	Number of iterations in the PCG and symmetric JCG (non-complex version) solvers.

**Table 196 \*GET Solution Items, Entity = ELEM**

<i>Entity</i> = ELEM, <i>ENTNUM</i> = 0 (or blank) (Available only after inertia relief solution [IRLF,1] or pre-calculation of masses [IRLF,-1])		
<b>*GET, Par, ELEM, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
MTOT	X, Y, Z	Total mass components.
MC	X, Y, Z	Mass centroid components.
IOR	X, Y, Z, XY, YZ, ZX	Moment of inertia about origin.
IMC	X, Y, Z, XY, YZ, ZX	Moment of inertia about mass centroid.
IPRIN	X, Y, Z	Principal centroidal moments of inertia.
IANG	XY, YZ, ZX	Angles of the principal axes.
FMC	X, Y, Z	Force components at mass centroid.
MMOR	X, Y, Z	Moment components at origin.
MMMC	X, Y, Z	Moment components at mass centroid.

**Table 197 \*GET Solution Items, Entity = MODE**

<i>Entity</i> = MODE, <i>ENTNUM</i> = <i>N</i> (mode number)		
<b>*GET, Par, MODE, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
FREQ	(IMAG)	Frequency of mode <i>N</i> . For modal solutions that create complex frequencies, the real part (for DAMP and QRDAMP eigensolves), or the imaginary part (for UNSYM eigensolver) of the frequencies is returned unless IT1NUM = IMAG.
PFACT		Participation factor of mode <i>N</i> . <ul style="list-style-type: none"> <li>If retrieved after a modal analysis, the real part of the participation factor is returned unless IT1NUM = IMAG. The direction is specified using Item2 = DIREC and IT2NUM = X, Y, Z, ROTX, ROTY, or ROTZ</li> <li>If retrieved after a spectrum analysis, the spectrum number <i>M</i> is specified using Item2 = SPECT and IT2NUM = <i>M</i>.</li> </ul>

<i>Entity</i> = <b>MODE</b> , <i>ENTNUM</i> = <i>N</i> (mode number)		
<b>*GET</b> , <i>Par</i> , <b>MODE</b> , <i>N</i> , <b>Item1</b> , <b>IT1NUM</b> , <b>Item2</b> , <b>IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
		For a PSD analysis with spatial correlation or wave excitation, the retrieved participation factors will correspond to the first degree of freedom that is excited.
MCOEF		Mode coefficient of mode <i>N</i> .  Returned values are valid only after a spectrum analysis has been solved. The spectrum number <i>M</i> is specified using <i>Item2</i> = SPECT and <i>IT2NUM</i> = <i>M</i> . In a SPRS analysis, the values returned are based on the curve with the lowest damping.  After a PSD analysis, the diagonal of the dynamic modal covariance matrix is retrieved for the displacement solution.
DAMP		Damping ratio of mode <i>N</i> . If retrieved after a modal analysis that creates complex solutions (DAMP, QRDAMP, or UNSYM eigensolvers) returned value is calculated from the complex frequencies.  If retrieved after a spectrum analysis, returned value is the effective damping ratio.  Not a function of direction. Also retrievable following a Harmonic Response Analysis or Transient Analysis with mode superposition.

For all items except PFACT and MCOEF (as noted above), only the first 10000 values corresponding to significant modes will be returned.

The MODE file must be available to retrieve items PFACT and MCOEF with specified *Item2*. If *Item2* is not specified, the last calculated value will be returned.

## Postprocessing Items

### **\*GET Postprocessing Entity Items**

- *Table 198: \*GET Postprocessing Items, Entity = ACTIVE* (p. 751)
- *Table 199: \*GET Postprocessing Items, Entity = CAMP* (p. 751)
- *Table 200: \*GET Postprocessing Items, Entity = CINT* (p. 752)
- *Table 201: \*GET Postprocessing Items, Entity = ELEM* (p. 752)
- *Table 202: \*GET Postprocessing Items, Entity = ETAB* (p. 753)
- *Table 203: \*GET Postprocessing Items, Entity = FSUM* (p. 754)
- *Table 204: \*GET Postprocessing Items, Entity = GSRESULT* (p. 754)
- *Table 205: \*GET Postprocessing Items, Entity = INTSRF* (p. 754)
- *Table 206: \*GET Postprocessing Items, Entity = KCALC* (p. 755)
- *Table 207: \*GET Postprocessing Items, Entity = NODE* (p. 755)
- *Table 208: \*GET Postprocessing Items, Entity = PATH* (p. 758)

- Table 209: \*GET Postprocessing Items, Entity = PLNSOL (p. 759)
- Table 210: \*GET Postprocessing Items, Entity = PRERR (p. 760)
- Table 211: \*GET Postprocessing Items, Entity = RAD (p. 760)
- Table 212: \*GET Postprocessing Items, Entity = SECR (p. 760)
- Table 213: \*GET Postprocessing Items, Entity = SECTION (p. 762)
- Table 214: \*GET Postprocessing Items, Entity = SORT (p. 763)
- Table 215: \*GET Postprocessing Items, Entity = SSUM (p. 763)
- Table 216: \*GET Postprocessing Items, Entity = TREF (p. 763)
- Table 217: \*GET Postprocessing Items, Entity = VARI (p. 764)

**Table 198 \*GET Postprocessing Items, Entity = ACTIVE**

<i>Entity</i> = ACTIVE, <i>ENTNUM</i> = 0 (or blank)		
*GET,Par, ACTIVE, 0, Item1, IT1NUM, Item2, IT2NUM		
Item1	IT1NUM	Description
SET	LSTP	Current load step number.
"	SBST	Current substep number.
"	TIME	Time associated with current results in the database.
"	FREQ	Frequency (for ANTYPE=MODAL, HARMIC, SPECTR; load factor for ANTYPE=BUCKLE).
"	NSET	If Item2 is blank, number of data sets on result file.  If Item2 = FIRST, IT2NUM = Loadstep, get set number of first substep of loadstep If Item2 = LAST, IT2NUM = Loadstep, get set number of last substep of loadstep
RSYS		Active results coordinate system.

**Table 199 \*GET Postprocessing Items, Entity = CAMP**

<b>Note</b>		
Available after <b>PLCAMP</b> or <b>PRCAMP</b> command is issued.		
<i>Entity</i> = CAMP, <i>ENTNUM</i> = N (mode number)		
*GET,Par, CAMP, N, Item1, IT1NUM, Item2, IT2NUM		
Item1	IT1NUM	Description
NBMO		Number of modes in the Campbell diagram ( <i>ENTNUM</i> not required). This value is the maximum value for <i>N</i> .
NBST		Number of steps in the Campbell diagram: modal load steps with rotational velocity ( <i>ENTNUM</i> not required). This value is the maximum value for <i>M</i> (see Item1 = FREQ).

<b>Note</b>		
Available after <b>PLCAMP</b> or <b>PRCAMP</b> command is issued.		
<i>Entity</i> = <b>CAMP</b> , <i>ENTNUM</i> = <i>N</i> (mode number)		
<b>*GET,Par, CAMP, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
WHRL		Whirl of mode <i>N</i> : -1 is backward whirl, 1 is forward whirl, and 0 is undetermined.
VCRI		Critical speed for mode <i>N</i> . This value is available if an excitation is defined via the <b>PLCAMP</b> or <b>PRCAMP</b> command's <i>SLOPE</i> argument. (The unit of speed depends upon the <i>UNIT</i> value specified in those commands.)
FREQ	<i>M</i>	Natural frequency of mode <i>N</i> at step <i>M</i> .

**Note**

Only the first 200 modes (NBMO) and the first 100 steps (NBST) are available.

**Table 200 \*GET Postprocessing Items, Entity = CINT**

<i>Entity</i> = <b>CINT</b> <i>ENTNUM</i> = <i>CrackId</i> (Crack ID number)		
<b>*GET,Par, CINT, CrackId, Item1, IT1NUM, Item2, IT2NUM, Item3, IT3NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
CTIP	CTNUM	Item1 = Label identifying crack tip node. IT1Num = Crack tip node number for fracture parameter data. Item2 = Label identifying contour information. IT2NUM = Contour number. Item3 = Label identifying data type. IT3NUM = Data type (JINT,IIN1,IIN2,IIN3,K1,K2,K3,G1,G2,G3,GT).
Depending on how the crack is defined, some fields are not be used. In all cases, however, the following values are required: the crack ID number and the crack tip node number.		

**Table 201 \*GET Postprocessing Items, Entity = ELEM**

<i>Entity</i> = <b>ELEM</b> , <i>ENTNUM</i> = <i>N</i> (element number)		
<b>*GET,Par, ELEM, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
SERR[1]		Structural error energy.
SDSG[1]		Absolute value of the maximum variation of any nodal stress component.

<i>Entity = ELEM, ENTNUM = N (element number)</i>		
<b>*GET,Par, ELEM, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
TERR[1]		Thermal error energy.
TDSG[1]		Absolute value of the maximum variation of any nodal thermal gradient component.
SENE		"Stiffness" energy or thermal heat dissipation. Same as TENE.
TENE		Thermal heat dissipation or "stiffness" energy. Same as SENE.
KENE		Kinetic energy.
JHEAT		Element Joule heat generation (coupled-field calculation).
JS	X, Y, Z	Source current density (coupled-field calculation) in the global Cartesian coordinate system.
HS	X, Y, Z	Average element magnetic field intensity from current sources.
VOLU		Element volume, as calculated during solution.
ETAB	<i>Lab</i>	Value of element table item <i>Lab</i> for element <i>N</i> (see <b>ETABLE</b> command).
SMISC	<i>Snum</i>	Value of element summable miscellaneous data at sequence number <i>Snum</i> (as used on <b>ETABLE</b> command).
NMISC	<i>Snum</i>	Value of element non-summable miscellaneous data at sequence number <i>Snum</i> (as used on <b>ETABLE</b> command).

1. Some element- and material-type limitations apply. For more information, see the documentation for the **PRERR** command.

**Table 202 \*GET Postprocessing Items, Entity = ETAB**

<i>Entity = ETAB, ENTNUM = N (column number)</i>		
<b>*GET,Par, ETAB, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
LAB		Label for column <i>N</i> of the element table [ <b>ETABLE</b> ]. Returns a character parameter.
ELEM	<i>E</i>	Value in <b>ETABLE</b> column <i>N</i> for element number <i>E</i> .
<i>Entity = ETAB, ENTNUM = 0 (or blank)</i>		
<b>*GET,Par,ETAB,0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
NCOL	MAX	Total number of <b>ETABLE</b> columns.

<i>Entity</i> = ETAB, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET,Par,ETAB,0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NLENG	MAX	Largest element number defined.

**Table 203 \*GET Postprocessing Items, Entity = FSUM**

<i>Entity</i> = FSUM, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET,Par,FSUM,0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
ITEM	<i>Lab</i>	Value of item <i>Lab</i> from last <b>FSUM</b> command. Valid labels are FX, FY, FZ, MX, MY, MZ, FLOW, HEAT, FLUX, etc.

**Table 204 \*GET Postprocessing Items, Entity = GSRESULT**

<i>Entity</i> = GSRESULT, <i>ENTNUM</i> = 0 (or blank) for generalized plane strain results in fiber direction		
<b>*GET,Par,GSRESULT,0, Item1, IT1NUM</b>		
Item1	IT1NUM	Description
LFIBER		Fiber length change at ending point.
ROT	X,Y	Rotation angle of end plane about X or Y axis.
F		Reaction force at ending point.
M	X,Y	Reaction moment on ending plane.

**Table 205 \*GET Postprocessing Items, Entity = INTSRF**

<i>Entity</i> = INTSRF, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET,Par,INTSRF,0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
PRES	<i>Lab</i>	Value of item <i>Lab</i> from last <b>INTSRF,PRES</b> command. Valid labels are FX, FY, FZ, MX, MY, and MZ.

<i>Entity</i> = <b>INTSRF</b> , <i>ENTNUM</i> = 0 (or blank)		
<b>*GET</b> , <i>Par</i> , <b>INTSRF</b> , 0, <i>Item1</i> , <b>IT1NUM</b> , <i>Item2</i> , <b>IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
TAUW	<i>Lab</i>	Value of item <i>Lab</i> from last <b>INTSRF</b> ,TAUW command. Valid labels are FX, FY, FZ, MX, MY, and MZ.

**Table 206 \*GET Postprocessing Items, Entity = KCALC**

<i>Entity</i> = <b>KCALC</b> , <i>ENTNUM</i> = 0 (or blank)		
<b>*GET</b> , <i>Par</i> , <b>KCALC</b> , 0, <i>Item1</i> , <b>IT1NUM</b> , <i>Item2</i> , <b>IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
K	1, 2, 3	Value of KI, KII, or KIII stress intensity factor from last <b>KCALC</b> command.

**Table 207 \*GET Postprocessing Items, Entity = NODE**

<i>Entity</i> = <b>NODE</b> , <i>ENTNUM</i> = <i>N</i> (node number) for nodal degree of freedom results:		
<b>*GET</b> , <i>Par</i> , <b>NODE</b> , <i>N</i> , <i>Item1</i> , <b>IT1NUM</b> , <i>Item2</i> , <b>IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum. Alternative get functions: UX( <i>N</i> ), UY( <i>N</i> ), UZ( <i>N</i> ).
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum. Alternative get functions: ROTX( <i>N</i> ), ROTY( <i>N</i> ), ROTZ( <i>N</i> ).
TEMP		Temperature. For <b>SHELL131</b> and <b>SHELL132</b> elements with KEYOPT(3) = 0 or 1, use TBOT, TE2, TE3, . . . , TTOP instead of TEMP. Alternative get functions: TEMP( <i>N</i> ), TBOT( <i>N</i> ), TE2( <i>N</i> ), etc.
PRES		Pressure. Alternative get function: PRES( <i>N</i> ).
VOLT		Electric potential. Alternative get function: VOLT( <i>N</i> ).
MAG		Magnetic scalar potential. Alternative get function: MAG( <i>N</i> ).
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum in a fluid analysis. X, Y, or Z nodal velocity or vector sum in a structural transient analysis (LS-DYNA analysis or ANSYS analysis with <b>ANTYPE</b> ,TRANS). Alternative get functions: VX( <i>N</i> ), VY( <i>N</i> ), VZ( <i>N</i> ).
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum in an electromagnetic analysis. X, Y, or Z nodal acceleration or vector sum in a structural transient analysis (LS-DYNA analysis or ANSYS analysis with <b>ANTYPE</b> ,TRANS). Alternative get functions: AX( <i>N</i> ), AY( <i>N</i> ), AZ( <i>N</i> ).
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy (FLOTRAN). Alternative get function: ENKE( <i>N</i> ).
ENDS		Turbulent energy dissipation (FLOTRAN). Alternative get function: ENDS( <i>N</i> ).

<b>Entity = NODE, ENTNUM = N (node number) for nodal degree of freedom results:</b>		
<b>*GET, Par, NODE, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
RF	FX, FY, FZ, MX, MY, MZ	Nodal reaction forces in the nodal coordinate system.
ORBT	A, B, PSI, PHI, YMAX, ZMAX	Whirl orbit characteristics:  A is the semi-major axis. B is the semi-minor axis. PSI is the angle between the local axis y and the major axis Y. PHI is the angle between initial position (t = 0) and major axis. YMAX is the maximum displacement along local y axis. ZMAX is the maximum displacement along local z axis.  Angles PSI and PHI are in degrees and within the range of -180 through +180.  Orbits are available only <i>after</i> issuing a <b>PRORB</b> command.
<b>Note</b>		
This command should be used very carefully when N represents an internal node because the nodal DOFs may have different physical meanings.		
<b>Entity = NODE, ENTNUM = N (node number) for averaged nodal results based on selected elements:</b>		
<b>*GET, Par, NODE, N, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.
"	MAXF	Maximum stress failure criterion.
"	TWSI	Tsai-Wu strength failure criterion.
"	TWSR	Inverse of Tsai-Wu strength ratio index failure criterion.
EPTO	X, Y, Z, XY, YZ, XZ,	Component total strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
"	MAXF	Maximum strain failure criterion.



*Entity = NODE, ENTNUM = N (node number) for averaged nodal results based on selected elements:*

**\*GET, Par, NODE, N, Item1, IT1NUM, Item2, IT2NUM**

Item1	IT1NUM	Description
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
CDM	DMG	Damage variable
"	LM	Maximum previous strain energy for virgin material
GKS	X, XY, XZ	Gasket component stress.
GKD	X, XY, XZ	Gasket component total closure.
GKDI	X, XY, XZ	Gasket component total inelastic closure.
GKTH	X, XY, XZ	Gasket component thermal closure.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
FAIL	MAX	Maximum of all failure criterion defined for this node.
"	EMAX	Maximum Strain failure criterion.
"	SMAX	Maximum Stress failure criterion.
"	TWSI	<a href="#">Tsai-Wu</a> Failure Criterion Strength Index failure criterion.
"	TWSR	Inverse of <a href="#">Tsai-Wu</a> Strength Ratio Index failure criterion.
"	USR1, USR2, ..., USRN	User-defined failure criteria.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable or plastic work/volume.
"	PLWK	Plastic work/volume.
TG	X, Y, Z, SUM	Component thermal gradient and sum.
TF	X, Y, Z, SUM	Component thermal flux and sum.
PG	X, Y, Z, SUM	Component pressure gradient and sum.
EF	X, Y, Z, SUM	Component electric field and sum.
D	X, Y, Z, SUM	Component electric flux density and sum.

*Entity* = NODE, *ENTNUM* = N (node number) for averaged nodal results based on selected elements:

**\*GET, Par, NODE, N, Item1, IT1NUM, Item2, IT2NUM**

Item1	IT1NUM	Description
H	X, Y, Z, SUM	Component magnetic field intensity and sum.
B	X, Y, Z, SUM	Component magnetic flux density and sum.
FMAG	X, Y, Z, SUM	Component electromagnetic force and sum.
HS	X, Y, Z	Component magnetic field intensity from current sources (in the global Cartesian coordinate system).
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution.
FICT	TEMP	Fictive temperature.

Element nodal results are the average nodal value of the selected elements.

*Entity* = NODE, *ENTNUM* = N (node number) for FLOTRAN results:

**\*GET, Par, NODE, N, Item1, IT1NUM, Item2, IT2NUM**

Item1	IT1NUM	Description
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function (2-D applications only).
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.

**Table 208 \*GET Postprocessing Items, Entity = PATH**

Entity = PATH, ENTNUM = 0 (or blank)		
<b>*GET, Par, PATH, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
MAX	<i>Lab</i>	Maximum value of path item <i>Lab</i> from last path operation. Valid labels are the user-defined labels on the <b>PDEF</b> or <b>PCALC</b> command.
MAXPATH		Returns the maximum path number defined.

<b>Entity = PATH, ENTNUM = 0 (or blank)</b>		
<b>*GET, Par, PATH, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
MIN	<i>Lab</i>	Minimum value of path item <i>Lab</i> from last path operation. Valid labels are the user-defined labels on the <b>PDEF</b> or <b>PCALC</b> command.
LAST	<i>Lab</i>	Last value of path item <i>Lab</i> from last path operation. Valid labels are the user-defined labels on the <b>PDEF</b> or <b>PCALC</b> command.
NODE		Value providing the number of nodes defining the path referenced in the last path operation.
ITEM	<i>Lab</i>	<i>Item2</i> = PATHPT, <i>IT2NUM</i> = <i>n</i> The value of <i>Lab</i> at the <i>n</i> th data point from the last path operation.
POINT	<i>n</i>	<i>Item2</i> = X,Y,Z, or CSYS. Returns information about the <i>n</i> th point on the current path.
NVAL		The number of path data points (the length of the data table) from the last path operation.
SET	<i>n</i>	<i>Item2</i> = NAME. Returns the name of the <i>n</i> th data set on the current path.
NUMPATH		Returns the number of paths defined.
<b>Entity = PATH, ENTNUM = n (path number)</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
NAME		Returns the name of the <i>n</i> th path.
<b>Entity = PATH, ENTNUM = 0 (or blank)</b>		
<b>*GET, Par, KCALC, 0, ...</b>		
K	1, 2, 3	Value of KI, KII, or KIII stress intensity factor from last <b>KCALC</b> command.

**Table 209 \*GET Postprocessing Items, Entity = PLNSOL**

<b>Entity = PLNSOL, ENTNUM = 0 (or blank)</b>		
<b>*GET, Par, PLNSOL, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
MAX		Maximum value of item in last contour display [ <b>PLNSOL</b> , <b>PLESOL</b> ].
MIN		Minimum value of item in last contour display [ <b>PLNSOL</b> , <b>PLESOL</b> ].
BMAX		Maximum bound value of item in last contour display [ <b>PLNSOL</b> , <b>PLESOL</b> ].

<i>Entity</i> = <b>PLNSOL</b> , <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, PLNSOL, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
BMIN		Minimum bound value of item in last contour display [ <b>PLNSOL</b> , <b>PLESOL</b> ].

**Table 210 \*GET Postprocessing Items, Entity = PRERR**

<i>Entity</i> = <b>PRERR</b> , <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, PRERR, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
SEPC[1]		Structural percent error in energy norm [ <b>PRERR</b> ].
TEPC[1]		Thermal percent error in energy norm [ <b>PRERR</b> ].
SERSM[1]		Structural error energy summation [ <b>PRERR</b> ].
TERSMS[1]		Thermal error energy summation [ <b>PRERR</b> ].
SENSM[1]		Structural energy summation [ <b>PRERR</b> ].
TENSM[1]		Thermal energy summation [ <b>PRERR</b> ].

1. Some element- and material-type limitations apply. For more information, see the documentation for the **PRERR** command.

**Table 211 \*GET Postprocessing Items, Entity = RAD**

<i>Entity</i> = <b>RAD</b> , <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, RAD, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
VFAVG		Value of the average view factor computed from the previous VFQUERY command.
<i>Entity</i> = <b>RAD</b> , <i>ENTNUM</i> = n (enclosure number)		
<b>*GET, Par, RAD, n, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NETHF		Value of the net heat rate lost by an enclosure.

**Table 212 \*GET Postprocessing Items, Entity = SECR**

<i>Entity</i> = <b>SECR</b> , <i>ENTNUM</i> = n (element number)				
<b>For beam and pipe (including elbow) section results, return values for all elements if the element number (n) is blank or ALL.</b>				
<b>*GET, Par, SECR, n, Item1, IT1NUM, Item2, IT2NUM</b>				
Item1	IT1NUM	Description	Item2	IT2NUM
S	X, Y, Z, XY, YZ, XZ	Component total stress	MAX – Returns maximum	These values are applicable only when

*Entity* = SECR, *ENTNUM* = *n* (element number)

For beam and pipe (including elbow) section results, return values for *all* elements

if the element number (*n*) is blank or ALL.

\*GET, *Par*, SECR, *n*, Item1, IT1NUM, Item2, IT2NUM

Item1	IT1NUM	Description	Item2	IT2NUM
	1, 2, 3	Principal stress value	MIN – Returns minimum	Item2 = IVAL, JVAL, or KVAL:
	INT	Stress intensity value		
	EQV	Equivalent stress value		
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain	MAXY – Returns section Y location of maximum	When KEY-OPT(15) = 0, this value is the section node number.
	1, 2, 3	Principal total strain value	MAXZ – Returns section Z location of maximum	
	INT	Total strain intensity value		
	EQV	Equivalent total strain value		
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain	MINY – Returns section Y location of minimum	When KEY-OPT(15) = 1 (or when using elbow elements), this value is the integration point number.
	1, 2, 3	Principal elastic strain value	MINZ – Returns section Z location of minimum	
	INT	Elastic strain intensity value		
	EQV	Equivalent elastic strain value		
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain	IVAL – Returns value at node or integration point at element I node	
	1, 2, 3	Principal thermal strain value	JVAL – Returns value at node or integration point at element J node	
	INT	Thermal strain intensity value		
	EQV	Equivalent thermal strain value		
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain	KVAL – Returns value at node or integration point at element K node (ELBOW290)	
	1, 2, 3	Principal plastic strain value		
	INT	Plastic strain intensity value		
	EQV	Equivalent plastic strain value		
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain	---	<b>For IVAL, JVAL, and KVAL:</b> The ALL (or blank) option for the element number is not valid. You must specify an element ( <i>n</i> ).
	1, 2, 3	Principal component creep strain value		
	INT	Component creep strain intensity value		
	EQV	Equivalent component creep strain value		
EPTT	X, Y, Z, XY, YZ, XZ	Component total mechanical and thermal strain		

<i>Entity</i> = SECR, <i>ENTNUM</i> = <i>n</i> (element number)				
For beam and pipe (including elbow) section results, return values for <i>all</i> elements if the element number ( <i>n</i> ) is blank or ALL.				
<b>*GET, Par, SECR, n, Item1, IT1NUM, Item2, IT2NUM</b>				
Item1	IT1NUM	Description	Item2	IT2NUM
	1, 2, 3	Principal total mechanical <i>and</i> thermal strain value		
	INT	Total mechanical <i>and</i> thermal strain intensity value		
	EQV	Equivalent total mechanical <i>and</i> thermal strain value		

Table 213 \*GET Postprocessing Items, Entity = SECTION

<i>Entity</i> = SECTION, <i>ENTNUM</i> = <i>component</i> (listed below).				
Generate data for section stress results, using <b>PRSECT</b> before retrieving these items. Valid labels for <i>ENTNUM</i> are MEMBRANE, BENDING, SUM (Membrane+Bending) , PEAK, and TOTAL. (The following items are not stored in the database and the values returned reflect the last quantities generated by <b>PRSECT</b> or <b>PLSECT</b> .) Only MEMBRANE, BENDING, and SUM data are available after a <b>PLSECT</b> command. The MEMBRANE label is only valid with <i>Item1</i> = INSIDE.				
<b>*GET, Par, SECTION, component, Item1, IT1NUM, Item2, IT2NUM</b>				
Item1	IT1NUM	Item2	Description	
INSIDE	S	X, Y, Z, XY, YZ, XZ	Stress component at beginning of path.	
"	"	1, 2, 3	Principal stress at beginning of path.	
"	"	INT, EQV	Stress intensity or equivalent stress at beginning of path.	
CENTER	S	X, Y, Z, XY, YZ, XZ	Stress component at midpoint of path.	
"	"	1, 2, 3	Principal stress at midpoint of path.	
"	"	INT, EQV	Stress intensity or equivalent stress at midpoint of path.	
OUTSIDE	S	X, Y, Z, XY, YZ, XZ	Stress component at end of path.	
"	"	1, 2, 3	Principal stress at end of path.	

<i>Entity</i> = SECTION, <i>ENTNUM</i> = <i>component</i> (listed below).			
Generate data for section stress results, using <b>PRSECT</b> before retrieving these items. Valid labels for <i>ENTNUM</i> are MEMBRANE, BENDING, SUM (Membrane+Bending) , PEAK, and TOTAL. (The following items are not stored in the database and the values returned reflect the last quantities generated by <b>PRSECT</b> or <b>PLSECT</b> .) Only MEMBRANE, BENDING, and SUM data are available after a <b>PLSECT</b> command. The MEMBRANE label is only valid with <i>Item1</i> = INSIDE.			
<b>*GET, Par, SECTION, component, Item1, IT1NUM, Item2, IT2NUM</b>			
Item1	IT1NUM	Item2	Description
"	"	INT, EQV	Stress intensity or equivalent stress at end of path.

Table 214 \*GET Postprocessing Items, Entity = SORT

<i>Entity</i> = SORT, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, SORT, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
MAX		Maximum value of last sorted item ( <b>NSORT</b> or <b>ESORT</b> command).
MIN		Minimum value of last sorted item ( <b>NSORT</b> or <b>ESORT</b> command).
IMAX		Node/Element number where maximum value occurs.
IMIN		Node/Element number where minimum value occurs.

Table 215 \*GET Postprocessing Items, Entity = SSUM

<i>Entity</i> = SSUM, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, SSUM, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
ITEM	<i>Lab</i>	Value of item <i>Lab</i> from last <b>SSUM</b> command. Valid labels are the user-defined labels on the <b>ETABLE</b> command.

Table 216 \*GET Postprocessing Items, Entity = TREF

<i>Entity</i> = TREF, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, TREF, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
ENER		Stored energy.
ENUM		Trefftz element number.
CEMIN		First (or minimum) constraint equation number associated with the Trefftz domain.
CEMAX		Last (or maximum) constraint equation number associated with the Trefftz domain.
NTZN		Number of Trefftz DOFs.
NSFN		Number of nodes on exterior surface.

<i>Entity</i> = TREF, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, TREF, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NSFE		Number of faces on exterior element.

**Table 217 \*GET Postprocessing Items, Entity = VARI**

<i>Entity</i> = VARI, <i>ENTNUM</i> = <i>N</i> (variable number after POST26 data storage) (for complex values, only the real portion is returned)		
<b>*GET, Par, VARI, N, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
EXTREM	VMAX	Maximum extreme value
"	TMAX	Time corresponding to VMAX.
"	VMIN	Minimum extreme value (after POST26 data storage).
"	TMIN	Time corresponding to VMIN.
"	VLAST	Last value (after POST26 data storage).
"	TLAST	Time corresponding to VLAST.
"	CVAR	Covariance
RTIME	t	Real value of variable <i>N</i> at time= <i>t</i> .
ITIME	t	Imaginary value of variable <i>N</i> at time= <i>t</i> .
RSET	<i>Snum</i>	Real value of variable <i>N</i> at location <i>Snum</i> .
ISET	<i>Snum</i>	Imaginary value of variable <i>N</i> at location <i>Snum</i> .
<i>Entity</i> = VARI, <i>ENTNUM</i> = 0 (or blank) (after POST26 data storage)		
<b>*GET, Par, VARI, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NSETS		Number of data sets stored (after POST26 data storage).

## Optimization and Probabilistic Design

### **\*GET Optimization and Probabilistic Design Entity Items**

- [Table 218: \\*GET Optimization and Probabilistic Design Items, Entity = OPT](#) (p. 764)
- [Table 219: \\*GET Optimization and Probabilistic Design Items, Entity = TOPO](#) (p. 765)
- [Table 220: \\*GET Optimization and Probabilistic Design Items, Entity = PDS \(pre\)](#) (p. 766)
- [Table 221: \\*GET Optimization and Probabilistic Design Items, Entity = PDS \(post\)](#) (p. 768)

**Table 218 \*GET Optimization and Probabilistic Design Items, Entity = OPT**

<i>Entity</i> = OPT, <i>ENTNUM</i> = 0 (or blank)		
<b>*GET, Par, OPT, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
TOTAL		Total number of analysis loops that have been executed.



<i>Entity = OPT, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, OPT, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
ITER		Total number of iterations for the optimization method or tool (i.e., per <b>OPEXE</b> command).
FEAS	<i>N</i>	Feasibility of design set <i>N</i> : 0=infeasible, 1=feasible.
TERM		Termination condition. For first order [ <b>OPTYPE</b> ,FIRST] or subproblem approximation [ <b>OPTYPE</b> ,SUBP] optimization: -1=not converged or not finished (still looping), 0=converged, 1=not converged due to too many sequential infeasible designs; 2=not converged due to too many iterations. For all other optimization methods: -1=not finished (still looping), 3=optimization complete.
BEST		Best design set. If design is feasible, best design is the one with the lowest value of the objective function. If infeasible, the best design is the one that is closest to being feasible.

**Table 219 \*GET Optimization and Probabilistic Design Items, Entity = TOPO**

<i>Entity = TOPO, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, TOPO, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
ACT		Status of topological optimization: 0=off, 1=on
TOELEM		Total number of elements used for topological optimization.
LOADS		Number of load cases specified in the <b>TOCOMP</b> command.
ITER		Current number of iterations performed. The iteration counter retrieved is always one greater than the actual iterations performed because the densities are also one iteration ahead (more current than those you are seeing).
MXIT		Maximum number of topological optimization iterations allowed.
CONV		Termination/convergence indicator: 0=not converged, 1=converged.
DIM		Dimensionality of the topological optimization problem: 0=2D, 1=3D, 2=Shell.
KAXI		Axisymmetric option within 2-D topological optimization: 0=off, 1=on.
POWP		Power of power law within axisymmetric option.
NEV		Total number of eigenvalues considered in topological optimization.
TOAC		Termination/convergence accuracy.
LOWD		Lower bound for element densities.
NTOC		Total number of constraints defined for topological optimization problem.
SFLAG		Solution approach active: 0=OC, 1=SCP.
COMP		Compliance value for current iteration.
PORV		Porous volume value for current iteration.

<i>Entity = TOPO, ENTNUM = n</i>		
<b>*GET, Par, TOPO, n, Item1, IT1NUM, Item2, IT2NUM</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
DENS		Element density used for topological optimization: low value (near 0)=material to be removed, high value (near 1)=material to keep. <i>n</i> = element number
FRQI		Individual frequencies for current topological optimization iteration. <i>n</i> =frequency ID
TCBO	FLAG	Constraint bounds for topological optimization: FLAG=1 - Lower bound. FLAG=2 - Upper bound. <i>n</i> = constraint ID
TCBF		Indicate whether bound should be treated as percentage of initial design or as an absolute value: 0=percentage, 1=absolute. <i>n</i> = constraint ID
TOHO		Value of topological objective for specified iteration. <i>n</i> = iteration counter
TOHC	CID	Value of topological constraint CID for specified iteration. <i>n</i> = iteration counter

**Table 220 \*GET Optimization and Probabilistic Design Items, Entity = PDS (pre)**

<i>Entity = PDS, ENTNUM = 0 (or blank)</i>		
<b>*GET, Par, PDS, 0, Item1, IT1NUM, Item2, IT2NUM.</b>		
<b>Item1</b>	<b>IT1NUM</b>	<b>Description</b>
ANLN		<i>Item2 = START; IT2NUM = N</i> ; Name of the analysis file containing the deterministic model. A character parameter of up to 8 characters, starting at position <i>N</i> , is returned. Use <b>*DIM</b> and <b>*DO</b> to get all 32 characters.
ANLX		Extension of the analysis file containing the deterministic model. A character parameter of up to 8 characters is returned.
ANLD		<i>Item2 = START; IT2NUM = N</i> ; Name of the directory of the analysis file containing the deterministic model. A character parameter of up to 8 characters, starting at position <i>N</i> , is returned. Use <b>*DIM</b> and <b>*DO</b> to get all 64 characters.
ASTP	1	Current setting for Autostop option (0 = disabled, 1 = enabled).
"	2	Current Autostop mean value accuracy.
"	3	Current Autostop standard deviation accuracy.
"	4	Current Autostop convergence checking frequency.
CORR	<i>i</i>	<i>Item2 = 0 or blank; IT2NUM = j</i> ; Correlation coefficient specified by the user between the <i>i</i> -th and the <i>j</i> -th random input variable. If no correlation has been specified between these two random variables a value of 0.0 is returned.
METH		Name of the current analysis method as specified in the <b>PDMETH</b> command (MCS or RSM). A character parameter of up to 8 characters is returned.

<i>Entity</i> = PDS, ENTNUM = 0 (or blank)		
<b>*GET, Par, PDS, 0, Item1, IT1NUM, Item2, IT2NUM.</b>		
Item1	IT1NUM	Description
NSIM		Number of simulation loops requested, as specified in the <b>PDMETH</b> and <b>PDDMCS, PDLHS, PDDOEL, or PDUSER</b> commands.
NTRP		Current total number of defined random output parameters.
NTRV		Current total number of defined random input variables
PAR1	<i>i</i>	First distribution parameter of the <i>i</i> -th defined random input variable.
PAR2	<i>i</i>	Second distribution parameter of the <i>i</i> -th defined random input variable.
PAR3	<i>i</i>	Third distribution parameter of the <i>i</i> -th defined random input variable.
PAR4	<i>i</i>	Fourth distribution parameter of the <i>i</i> -th defined random input variable.
RNAM	<i>i</i>	<i>Item2</i> = START; <i>IT2NUM</i> = <i>N</i> ; Name of the <i>i</i> -th defined random output parameter. A character parameter of up to 8 characters, starting at position <i>N</i> , is returned. Use <b>*DIM</b> and <b>*DO</b> to get all characters.
SAMP		Name of the current sampling technique as specified in the <b>PDMETH</b> command (LHS, DIR, USER for MCS, or CCD, BBM, USER for RSM). A character parameter of up to 8 characters is returned.
VDIS	<i>i</i>	Label of the distribution type of the <i>i</i> -th defined random input variable (BETA, EXPO, ..., WEIB). A character parameter of up to 8 characters is returned.
VNAM	<i>i</i>	<i>Item2</i> = START; <i>IT2NUM</i> = <i>N</i> ; Name of the <i>i</i> -th defined random input variable. A character parameter of up to 8 characters, starting at position <i>N</i> , is returned. Use <b>*DIM</b> and <b>*DO</b> to get all characters.
CCDL		<i>Item2</i> = DEFA; <i>IT2NUM</i> = <i>j</i> ; Default value for the probabilities of the <i>j</i> -th design-of-experiment level for a central composite design as used in the by the <b>PDDOEL</b> command.
CCDL	<i>i</i>	<i>Item2</i> = VTYP; <i>IT2NUM</i> =0 (or blank); Type of the level values of the design-of-experiment for a central composite design of the <i>i</i> -th defined random input variable as specified by the <b>PDDOEL</b> command (PROB, PHYS). A character parameter of up to 8 characters is returned.
CCDL	<i>i</i>	<i>Item2</i> = LOPT; <i>IT2NUM</i> =0 (or blank); Type of the level definition of the design-of-experiment for a central composite design of the <i>i</i> -th defined random input variable as specified by the <b>PDDOEL</b> command (BND, ALL). A character parameter of up to 8 characters is returned.
CCDL	<i>i</i>	<i>Item2</i> = LDEF; <i>IT2NUM</i> = <i>j</i> ; Flag indicating if the <i>j</i> -th design-of-experiment level for a central composite design of the <i>i</i> -th defined random input variable has been defined with the <b>PDDOEL</b> command. (0=NO, 1=YES).

<i>Entity</i> = PDS, ENTNUM = 0 (or blank)		
<b>*GET, Par, PDS, 0, Item1, IT1NUM, Item2, IT2NUM.</b>		
Item1	IT1NUM	Description
CCDL	<i>i</i>	<i>Item2</i> = LVAL; <i>IT2NUM</i> = <i>j</i> ; Level value for the <i>j</i> -th design-of-experiment level for a central composite design of the <i>i</i> -th defined random input variable as specified by the <b>PDDOEL</b> command. If the <b>PDDOEL</b> command has not been used for the <i>i</i> -th defined random input variable or if the user has not specified the <i>j</i> -th design-of-experiment level, then the default probability level will be returned.
BBML		<i>Item2</i> = DEFA; <i>IT2NUM</i> = <i>j</i> ; Default value for the probabilities of the <i>j</i> -th design-of-experiment level for a Box-Behnken Matrix design as used in the by the <b>PDDOEL</b> command.
BBML	<i>i</i>	<i>Item2</i> = VTYP; <i>IT2NUM</i> =0 (or blank); Type of the level values of the design-of-experiment level for a Box-Behnken Matrix design of the <i>i</i> -th defined random input variable as specified by the <b>PDDOEL</b> command. (PROB, PHYS). A character parameter of up to 8 characters is returned.
BBML	<i>i</i>	<i>Item2</i> = LOPT; <i>IT2NUM</i> =0 (or blank); Type of the level definition of the design-of-experiment level for a Box-Behnken Matrix design of the <i>i</i> -th defined random input variable as specified by the <b>PDDOEL</b> command (BND, ALL). A character parameter of up to 8 characters is returned.
BBML	<i>i</i>	<i>Item2</i> = LDEF; <i>IT2NUM</i> = <i>j</i> ; Flag indicating if the <i>j</i> -th design-of-experiment level for a Box-Behnken Matrix design of the <i>i</i> -th defined random input variable has been defined with the <b>PDDOEL</b> command. (0=NO, 1=YES).
BBML	<i>i</i>	<i>Item2</i> = LVAL; <i>IT2NUM</i> = <i>j</i> ; Level value for the <i>j</i> -th design-of-experiment level for a Box-Behnken Matrix design of the <i>i</i> -th defined random input variable as specified by the <b>PDDOEL</b> command. If the <b>PDDOEL</b> command has not been used for the <i>i</i> -th defined random input variable or if the user has not specified the <i>j</i> -th design-of-experiment level, then the default probability level will be returned.

**Table 221 \*GET Optimization and Probabilistic Design Items, Entity = PDS (post)**

<i>Entity</i> = PDS, ENTNUM = 0 (or blank)		
<b>*GET, Par, PDS, 0, Item1, IT1NUM, Item2, IT2NUM</b>		
Item1	IT1NUM	Description
NRSS		Number of response surface sets that are currently available.
NSOL		Number of probabilistic solution sets that are currently available. This coincides with the number of probabilistic analyses that are currently done.

*Entity* = PDS, ENTNUM = *n* (*n*-th result set).

The numbering or ordering of the result sets is used as follows: If NSOL is the number of solution sets and NRSS is the number response surface sets then the solution sets are indexed from 1 to NSOL and the response surface sets are indexed from NSOL+1 to NSOL+NRSS.

Note that some options listed below apply only to solution sets (i.e., where  $1 \leq n \leq \text{NSOL}$ ) and some apply only to response surface sets (i.e., where  $\text{NSOL}+1 \leq n \leq \text{NSOL}+\text{NRSS}$ ).

\*GET,Par, PDS, *n*, Item1, IT1NUM, Item2, IT2NUM

Item1	IT1NUM	Description
METH		Name of the analysis method used in the <i>n</i> -th solution set (MCS, RSM). A character parameter of up to 8 characters is returned. This applies only to solution sets (i.e., " <i>n</i> " ranges from 1 to NSOL).
NSIM		Number of simulation samples that are available for postprocessing in the <i>n</i> -th result set. If the <i>n</i> -th result set is a solution set (i.e., $1 \leq n \leq \text{NSOL}$ ) then this is equal to the number of successful (no error occurred) finite element analysis loops regardless of the probabilistic method used for the solution set. If <i>n</i> points to a response surface set, then this is equal to the number of simulation samples that were performed on the response surfaces included in this response surface set.
SAMP		Name of the sampling technique used in the <i>n</i> -th solution set (LHS, DIR, USER for MCS or CCD, BBM, USER for RSM). A character parameter of up to 8 characters is returned. This applies only to solution sets (i.e., <i>n</i> ranges from 1 to NSOL).
RLAB		<i>Item2</i> = START; <i>IT2NUM</i> = <i>N</i> ; Name of the <i>n</i> -th defined result set. A character parameter of up to 8 characters, starting at position <i>N</i> , is returned. Use *DIM and *DO to get all characters.
MEAN		<i>Item2</i> = RV; <i>IT2NUM</i> = <i>j</i> ; The mean value of the <i>j</i> -th defined random input variable in the <i>n</i> -th result set.
MEAN		<i>Item2</i> = RP; <i>IT2NUM</i> = <i>j</i> ; The mean value of the <i>j</i> -th defined random output parameter in the <i>n</i> -th result set.
STDV		<i>Item2</i> = RV; <i>IT2NUM</i> = <i>j</i> ; The standard deviation of the <i>j</i> -th defined random input variable in the <i>n</i> -th result set.
STDV		<i>Item2</i> = RP; <i>IT2NUM</i> = <i>j</i> ; The standard deviation of the <i>j</i> -th defined random output parameter in the <i>n</i> -th result set.
KURT		<i>Item2</i> = RV; <i>IT2NUM</i> = <i>j</i> ; The coefficient of kurtosis of the <i>j</i> -th defined random input variable in the <i>n</i> -th result set.
KURT		<i>Item2</i> = RP; <i>IT2NUM</i> = <i>j</i> ; The coefficient of kurtosis of the <i>j</i> -th defined random output parameter in the <i>n</i> -th result set.
SKEW		<i>Item2</i> = RV; <i>IT2NUM</i> = <i>j</i> ; The coefficient of skewness of the <i>j</i> -th defined random input variable in the <i>n</i> -th result set.
SKEW		<i>Item2</i> = RP; <i>IT2NUM</i> = <i>j</i> ; The coefficient of skewness of the <i>j</i> -th defined random output parameter in the <i>n</i> -th result set.
MIN		<i>Item2</i> = RV; <i>IT2NUM</i> = <i>j</i> ; The minimum sampled value of the <i>j</i> -th defined random input variable in the <i>n</i> -th result set.

*Entity* = PDS, ENTNUM = *n* (*n*-th result set).

The numbering or ordering of the result sets is used as follows: If NSOL is the number of solution sets and NRSS is the number response surface sets then the solution sets are indexed from 1 to NSOL and the response surface sets are indexed from NSOL+1 to NSOL+NRSS.

Note that some options listed below apply only to solution sets (i.e., where  $1 \leq n \leq \text{NSOL}$ ) and some apply only to response surface sets (i.e., where  $\text{NSOL}+1 \leq n \leq \text{NSOL}+\text{NRSS}$ ).

\*GET, *Par*, PDS, *n*, *Item1*, IT1NUM, *Item2*, IT2NUM

Item1	IT1NUM	Description
MIN		<i>Item2</i> = RP; <i>IT2NUM</i> = <i>j</i> ; The minimum sampled value of the <i>j</i> -th defined random output parameter in the <i>n</i> -th result set.
MAX		<i>Item2</i> = RV; <i>IT2NUM</i> = <i>j</i> ; The minimum sampled value of the <i>j</i> -th defined random input variable in the <i>n</i> -th result set.
MAX		<i>Item2</i> = RP; <i>IT2NUM</i> = <i>j</i> ; The minimum sampled value of the <i>j</i> -th defined random output parameter in the <i>n</i> -th result set.
CCDL	<i>i</i>	<i>Item2</i> = VTYP; Type of the level values of the design-of-experiment for a central composite design of the <i>i</i> -th defined random input variable that has been used during the execution of the <i>n</i> -th solution set (PROB, PHYS). A character parameter of up to 8 characters is returned. This applies only to solution sets (i.e., <i>n</i> ranges from 1 to NSOL).
CCDL	<i>i</i>	<i>Item2</i> = LOPT; Type of the level definition of the design-of-experiment for a central composite design of the <i>i</i> -th defined random input variable that has been used during the execution of the <i>n</i> -th solution set (BND, ALL). A character parameter of up to 8 characters is returned. This applies only to solution sets (i.e., <i>n</i> ranges from 1 to NSOL).
CCDL	<i>i</i>	<i>Item2</i> = LDEF; <i>IT2NUM</i> = <i>j</i> ; Flag indicating if the <i>j</i> -th design-of-experiment level for a central composite design of the <i>i</i> -th defined random input variable has been specified for the <i>n</i> -th solution set. (0=NO, 1=YES). This applies only to solution sets (i.e., <i>n</i> ranges from 1 to NSOL).
CCDL	<i>i</i>	<i>Item2</i> = LVAL; <i>IT2NUM</i> = <i>j</i> ; Level value for the <i>j</i> -th design-of-experiment level for a central composite design of the <i>i</i> -th defined random input variable that has been used during the execution of the <i>n</i> -th solution set. This applies only to solution sets (i.e., <i>n</i> ranges from 1 to NSOL).
BBML	<i>i</i>	<i>Item2</i> = VTYP; Type of the level values of the design-of-experiment level for a Box-Behnken Matrix design of the <i>i</i> -th defined random input variable that has been used during the execution of the <i>n</i> -th solution set. (PROB, PHYS). A character parameter of up to 8 characters is returned. This applies only to solution sets (i.e., <i>n</i> ranges from 1 to NSOL).
BBML	<i>i</i>	<i>Item2</i> = LOPT; Type of the level definition of the design-of-experiment level for a Box-Behnken Matrix design of the <i>i</i> -th defined random input variable that has been used during the execution of the <i>n</i> -th solution set (BND, ALL). A character parameter of up to 8

*Entity* = PDS, ENTNUM = *n* (*n*-th result set).

The numbering or ordering of the result sets is used as follows: If NSOL is the number of solution sets and NRSS is the number response surface sets then the solution sets are indexed from 1 to NSOL and the response surface sets are indexed from NSOL+1 to NSOL+NRSS.

Note that some options listed below apply only to solution sets (i.e., where  $1 \leq n \leq \text{NSOL}$ ) and some apply only to response surface sets (i.e., where  $\text{NSOL}+1 \leq n \leq \text{NSOL}+\text{NRSS}$ ).

\*GET,*Par*, PDS, *n*, Item1, IT1NUM, Item2, IT2NUM

Item1	IT1NUM	Description
		characters is returned. This applies only to solution sets (i.e., <i>n</i> ranges from 1 to NSOL).
BBML	<i>i</i>	<i>Item2</i> = LDEF; <i>IT2NUM</i> = <i>j</i> ; Flag indicating if the <i>j</i> -th design-of-experiment level for a Box-Behnken Matrix design of the <i>i</i> -th defined random input variable specified for the <i>n</i> -th solution set. (0=NO, 1=YES).
BBML	<i>i</i>	<i>Item2</i> = LVAL; <i>IT2NUM</i> = <i>j</i> ; Level value for the <i>j</i> -th design-of-experiment level for a Box-Behnken Matrix design of the <i>i</i> -th defined random input variable that has been used during the execution of the <i>n</i> -th solution set. This applies only to solution sets (i.e., <i>n</i> ranges from 1 to NSOL).
RSST		<i>Item2</i> = XSOL; Index of the solution set the response surface set identified by the <i>n</i> -th result set is associated with. This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSST		<i>Item2</i> = NFRP; Number of fitted random output parameters in the response surface set identified by the <i>n</i> -th result set is associated with. This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSUR	<i>i</i>	<i>Item2</i> = XFRP; Index of the random output parameter that has been fitted with the <b>RSFIT</b> command to fit the <i>i</i> -th response surface in the response surface set identified by the <i>n</i> -th result set. <i>IT1NUM</i> = <i>i</i> ranges from 1 to NFRP (see <i>Item1</i> =RSST, <i>Item2</i> =NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSUR	<i>i</i>	<i>Item2</i> = RMOD; Label for the response surface model that has been used in the <b>RSFIT</b> command to fit the <i>i</i> -th response surface in the response surface set identified by the <i>n</i> -th result set. <i>IT1NUM</i> = <i>i</i> ranges from 1 to NFRP (see <i>Item1</i> =RSST, <i>Item2</i> =NFRP). A character parameter of up to 8 characters is returned. This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSUR	<i>i</i>	<i>Item2</i> = YTRT; Label of the type of transformation that has been used in the <b>RSFIT</b> command to fit the random output parameter of <i>i</i> -th response surface in the response surface set identified by the <i>n</i> -th result set. <i>IT1NUM</i> = <i>i</i> ranges from 1 to NFRP (see <i>Item1</i> =RSST, <i>Item2</i> =NFRP). A character parameter of up to 8 characters is returned. This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).



*Entity* = PDS, ENTNUM = *n* (*n*-th result set).

The numbering or ordering of the result sets is used as follows: If NSOL is the number of solution sets and NRSS is the number response surface sets then the solution sets are indexed from 1 to NSOL and the response surface sets are indexed from NSOL+1 to NSOL+NRSS.

Note that some options listed below apply only to solution sets (i.e., where  $1 \leq n \leq \text{NSOL}$ ) and some apply only to response surface sets (i.e., where  $\text{NSOL}+1 \leq n \leq \text{NSOL}+\text{NRSS}$ ).

\*GET, *Par*, PDS, *n*, Item1, IT1NUM, Item2, IT2NUM

Item1	IT1NUM	Description
RSUR	<i>i</i>	<i>Item2</i> = YTRV; Transformation value of the transformation that has been used in the <b>RSFIT</b> command to fit the random output parameter of <i>i</i> -th response surface in the response surface set identified by the <i>n</i> -th result set. IT1NUM= <i>i</i> ranges from 1 to NFRP (see Item1=RSST, Item2=NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSUR	<i>i</i>	<i>Item2</i> = FILT; Label of the filtering type of the regression terms that has been used in the <b>RSFIT</b> command to fit the random output parameter of <i>i</i> -th response surface in the response surface set identified by the <i>n</i> -th result set. IT1NUM= <i>i</i> ranges from 1 to NFRP (see Item1=RSST, Item2=NFRP). A character parameter of up to 8 characters is returned. This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSUR	<i>i</i>	<i>Item2</i> = CONF; Confidence level value of the regression term filtering that has been used in the <b>RSFIT</b> command to fit the random output parameter of <i>i</i> -th response surface in the response surface set identified by the <i>n</i> -th result set. IT1NUM= <i>i</i> ranges from 1 to NFRP (see Item1=RSST, Item2=NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSEQ	<i>i</i>	<i>Item2</i> = YBOX; Box-Cox transformation value "lambda" of the response surface equation for the <i>i</i> -th fitted random output parameter in the <i>n</i> -th result set. IT1NUM= <i>i</i> ranges from 1 to NFRP (see Item1=RSST, Item2=NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSEQ	<i>i</i>	<i>Item2</i> = NTRM; Number of regression terms of the response surface equation for the <i>i</i> -th fitted random output parameter in the <i>n</i> -th result set. IT1NUM= <i>i</i> ranges from 1 to NFRP (see Item1=RSST, Item2=NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSEQ	<i>i</i>	<i>Item2</i> = TTP; <i>IT2NUM=j</i> ; Type of the <i>j</i> -th regression term of the response surface equation for the <i>i</i> -th fitted random output parameter in the <i>n</i> -th result set. IT1NUM= <i>i</i> ranges from 1 to NFRP (see Item1=RSST, Item2=NFRP). <i>IT2NUM=j</i> ranges from 1 to NTRM (see Item1=RSEQ, Item2=NTRM). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS). Possible return values are:  1 = term is a constant (this term does not involve any random input variables)



*Entity* = PDS, ENTNUM = *n* (*n*-th result set).

The numbering or ordering of the result sets is used as follows: If NSOL is the number of solution sets and NRSS is the number response surface sets then the solution sets are indexed from 1 to NSOL and the response surface sets are indexed from NSOL+1 to NSOL+NRSS.

Note that some options listed below apply only to solution sets (i.e., where  $1 \leq n \leq \text{NSOL}$ ) and some apply only to response surface sets (i.e., where  $\text{NSOL}+1 \leq n \leq \text{NSOL}+\text{NRSS}$ ).

\*GET,Par, PDS, *n*, Item1, IT1NUM, Item2, IT2NUM

Item1	IT1NUM	Description
		2 = term is a linear term (this term involves only one random input variable)  3 = term is a purely quadratic term involving only one random input variable (this term involves only one random input variable)  4 = term is a mixed quadratic term involving two random input variables (this term involves two random input variables)
RSEQ	<i>i</i>	<i>Item2</i> = XRV1; <i>IT2NUM</i> = <i>j</i> ; Index of the first random input variable involved in the <i>j</i> -th regression term of the response surface equation for the <i>i</i> -th fitted random output parameter in the <i>n</i> -th result set. <i>IT1NUM</i> = <i>i</i> ranges from 1 to NFRP (see <i>Item1</i> =RSST, <i>Item2</i> =NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS). An error appears if the term does not involve a random input variable, i.e. if the term is a constant.
RSEQ	<i>i</i>	<i>Item2</i> = XRV2; <i>IT2NUM</i> = <i>j</i> ; Index of the second random input variable involved in the <i>j</i> -th regression term of the response surface equation for the <i>i</i> -th fitted random output parameter in the <i>n</i> -th result set. <i>IT1NUM</i> = <i>i</i> ranges from 1 to NFRP (see <i>Item1</i> =RSST, <i>Item2</i> =NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS). An error appears if the term does not involve a second random input variable, i.e. if the term is not a mixed quadratic term.
RSEQ	<i>i</i>	<i>Item2</i> = COEF; <i>IT2NUM</i> = <i>j</i> ; Regression coefficient of the <i>j</i> -th regression term of the response surface equation for the <i>i</i> -th fitted random output parameter in the <i>n</i> -th result set. <i>IT1NUM</i> = <i>i</i> ranges from 1 to NFRP (see <i>Item1</i> =RSST, <i>Item2</i> =NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSEQ	<i>i</i>	<i>Item2</i> = SLOP; <i>IT2NUM</i> = <i>j</i> ; Scaling slope of the <i>j</i> -th random input variable of the response surface equation for the <i>i</i> -th fitted random output parameter in the <i>n</i> -th result set. <i>IT1NUM</i> = <i>i</i> ranges from 1 to NFRP (see <i>Item1</i> =RSST, <i>Item2</i> =NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).
RSEQ	<i>i</i>	<i>Item2</i> = ICPT; <i>IT2NUM</i> = <i>j</i> ; Scaling intercept of the <i>j</i> -th random input variable of the response surface equation for the <i>i</i> -th fitted random output parameter in the <i>n</i> -th result set. <i>IT1NUM</i> = <i>i</i> ranges from 1 to NFRP (see <i>Item1</i> =RSST, <i>Item2</i> =NFRP). This applies only to response surface sets (i.e., <i>n</i> ranges from NSOL+1 to NSOL+NRSS).

## Menu Paths

Main Menu>Prob Design>Prob Method>Response Surface  
Utility Menu>Parameters>Get Scalar Data

---

**/GFILE**, *SIZE*

Specifies the pixel resolution on Z-buffered graphics files.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*SIZE*

Pixel resolution. Defaults to a pixel resolution of 800. Valid values are from 256 to 2400.

## Command Default

800 pixels

## Notes

Defines the pixel resolution on subsequently written graphics files (`Jobname.GRPH`) for software Z-buffered displays [**/TYPE**]. Lowering the pixel resolution produces a "fuzzier" image; increasing the resolution produces a "sharper" image but takes a little longer.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Redirect Plots>To GRPH File  
Utility Menu>PlotCtrls>Redirect Plots>To PSCR File

---

**/GFORMAT**, *Ftype*, *NWIDTH*, *DSIGNF*

Specifies the format for the graphical display of numbers.

[GRAPHICS: Labeling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Ftype*

FORTTRAN format types (G is the default if this field is left blank.)

**G**

G $_{xx.yy}$ .  $xx$  and  $yy$  are described below.

**F**

F $_{xx.yy}$

**E**

E $_{xx.yy}$

*NWIDTH*

Total width (12 maximum) of the field (the  $xx$  in *Ftype*). Defaults to 12.

**DSIGNF**

Number of digits after the decimal point (*yy* in F or E format) or number of significant digits in G format. Range is 1 to *xx-6* for *Ftype* = G or E; and 0 to *xx-3* for *Ftype* = F. The default is a function of *Ftype* and *NWIDTH*.

**Notes**

Lets you control the format of the graphical display of floating point numbers. Issue **/GFORMAT,STAT** to display the current settings; issue **/GFORMAT,DEFA** to let ANSYS choose the format for the graphical display of floating numbers.

This command is valid in any processor.

**Menu Paths**

Utility Menu>PlotCtrls>Style>Floating Point Format

**/GLINE, WN, STYLE**

**Specifies the element outline style.**

GRAPHICS:Style

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WN**

Window number (or ALL) to which command applies (defaults to 1).

**STYLE**

Outline key:

**0**

Solid element outlines (default)

**1**

Dashed element outlines

**-1**

No element outlines

**Command Default**

Solid element outlines

**Notes**

Determines the element outline style. Often used when node numbers are displayed to prevent element lines from overwriting node numbers.

Unless you are using an OpenGL or Starbase driver, the dashed element outline option (**/GLINE,WN,1**) is not available in the following situations:

- Z-buffered displays (**/TYPE,WN,6**).
- Capped Z-buffered displays (**/TYPE,WN,7**).

- Qslice Z-buffered displays (**/TYPE**,*WN*,8).

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Edge Options

---

**/GMARKER**, *CURVE*, *KEY*, *INCR*

Specifies the curve marking style.

[GRAPHICS:Style](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *CURVE*

Curve number markers will be applied on (integer value between 1 and 10).

### *KEY*

Marker key:

**0**

No markers will be applied (default).

**1**

TRIANGLES will be applied.

**2**

SQUARES will be applied.

**3**

DIAMONDS will be applied.

**4**

CROSSES will be applied.

### *INCR*

Determines the curve marking frequency. (a whole number value between 1 and 255). If *INCR* = 1, markers are displayed at every data point on the curve. If *INCR* = 2 then markers are displayed at every second data point. If *INCR* = 3 then they are displayed at every third data point.

## Command Default

No markers will be applied.

## Notes

The user-specified markers will not be drawn when the area under the curve is color-filled (**/GROPT**, FILL).

## Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Curve

---

**GMATRIX**, *SYMFAC*, *Condname*, *NUMCOND*, *--*, *Matrixname*

**Performs electric field solutions and calculates the self and mutual conductance between multiple conductors.**

SOLUTION: Analysis Options

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

***SYMFAC***

Geometric symmetry factor. Conductance values are scaled by this factor which represents the fraction of the total device modeled. Defaults to 1.

***Condname***

Alphanumeric prefix identifier used in defining named conductor components.

***NUMCOND***

Total number of components. If a ground is modeled, it is to be included as a component.

***--***

Unused field.

***Matrixname***

Array name for computed conductance matrix. Defaults to GMATRIX.

## Notes

To invoke the **GMATRIX** macro, the exterior nodes of each conductor must be grouped into individual components using the **CM** command. Each set of independent components is assigned a component name with a common prefix followed by the conductor number. A conductor system with a ground must also include the ground nodes as a component. The ground component is numbered last in the component name sequence.

A ground conductance matrix is a matrix relating current to a voltage vector. A ground matrix cannot be applied to a circuit modeler such as SPICE. The lumped conductance matrix is a matrix formed by a combination of lumped "arrangements" of voltage differences between conductors. You can use the lumped conductance terms in a circuit modeler such as SPICE to represent conductances between conductors.

You must enclose all name-strings in single quotes in the **GMATRIX** command line.

**GMATRIX** works with the following elements:

- [SOLID5](#) (KEYOPT(1) = 9)
- [SOLID98](#) (KEYOPT(1) = 9)
- [LINK68](#)
- [PLANE230](#)
- [SOLID231](#)
- [SOLID232](#)

This command is available from the menu path shown below only if existing results are available.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Component Based>Self/Mutual conductance

---

### GMFACE, Lab, N

Specifies the facet representation used to form solid models.

[GRAPHICS:Style](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### Lab

Valid Labels:

#### FINE

Value that determines how coarse the facets will be.

#### N

An integer value between one (small) and ten (large) that determines the tolerances that will be applied to the creation of arcs and surfaces. Ten will create many facets, which may in turn cause ANSYS to run very slowly. One will create fewer facets, which may in turn cause larger tolerance errors.

## Menu Paths

Utility Menu>PlotCtrls>Style>Solid Model Facets

---

### \*GO, Base

Causes a specified line on the input file to be read next.

[APDL:Process Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

#### Base

"Go to" action:

**:label --**

A user-defined label (beginning with a colon (:), 8 characters maximum). The command reader will skip (and wrap to the beginning of the file, if necessary) to the first line that begins with the matching *label*.

---

### Caution

This label option may not be mixed with do-loop or if-then-else constructs.

**STOP --**

This action will cause an exit from the ANSYS program at this line.

## Command Default

Read lines sequentially.

## Notes

Causes the next read to be from a specified line on the input file. Lines may be skipped or reread. The **\*GO** command will *not* be executed unless it is part of a macro, user file (processed by **\*USE**), an alternate input file (processed by **/INPUT**), or unless it is used in a batch-mode input stream. Jumping into, out of, or within a do-loop or an if-then-else construct to a *:label* line is not allowed.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

## /GO

Reactivates suppressed printout.

SESSION: List Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Reactivates printout suppressed with the **/NOPR** command without producing any output. The **/GOPR** command has the same function except that it also produces a command response from the program.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Coupling / Ceqn>Rigid Region**

**Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts**

## /GOLIST

Reactivates the suppressed data input listing.

SESSION: List Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Reactivates printout of the data input listing suppressed with **/NOLIST**.

This command is valid in any processor, but only within a batch run [**/BATCH**].

## Menu Paths

This command cannot be accessed from a menu.

---

### /GOPR

Reactivates suppressed printout.

SESSION: [List Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Reactivates printout suppressed with the **/NOPR** command. The **/GO** command has the same function except that it does not produce a command response from the program.

This command is valid in any processor.

## Menu Paths

**Main Menu>Solution>Time Controls>Time Step Prediction**

---

**GP**, *NODE1*, *NODE2*, *Lab*, *STIF*, *GAP*, *DAMP*

Defines a gap condition for transient analyses.

SOLUTION: [Gap Conditions](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### **NODE1**

Node I of gap. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### **NODE2**

Node J of gap (must be different from *NODE1*). Non-grounded gap nodes must be defined as master degrees of freedom or be unconstrained, active DOF in a full analysis type. Grounded gap nodes (those not defined as MDOF) need not appear elsewhere in the model.

### **Lab**

Direction of gap action in the nodal coordinate system (implied from the following force labels): FX, FY, FZ, MX, MY, MZ.

### **STIF**

Stiffness (Force/Length) of closed gap (may be positive or negative).

---

### Note

High stiffness requires a small integration time step for numerical stability.



**GAP**

Initial size of gap. A zero (or positive) value assumes an initially open gap. A negative value defines an interference condition. For a rotational gap, *GAP* should be in radians.

**DAMP**

Damping coefficient (Force\*Time/Length) of closed gap using pseudo velocity (Newmark finite difference expansion scheme).

**Notes**

Defines a gap condition for the reduced or mode superposition transient (**ANTYPE,TRANS**) analysis. If used in SOLUTION, this command is valid only within the first load step.

Repeat **GP** command for additional gap conditions. Gaps are numbered sequentially as input.

**Note**

Gaps may be renumbered by the program during the solution (see output listing)

The reduced or mode superposition transient analysis (**ANTYPE,TRANS** and **TRNOPT,REDUC** or **MSUP**) does not allow gap action with the standard ANSYS gap elements. Gap conditions, however, producing the same effect, may be defined. The gap condition simulates the basic gap action of the **COMBIN40** element. The gap condition is treated as an explicit force (equal to the interference times contact stiffness) and affects only the load vector calculation and not the reduced stiffness matrix. The interference is calculated from the displacement extrapolated from the previous time points. A gap condition may be defined between a master degree of freedom and ground or another master degree of freedom. When a non-reduced mode extraction method is used, a master degree of freedom implies an unconstrained, active degree of freedom. Gap nodes not defined as master degrees of freedom or attached to an element are assumed to be grounded. Grounded gap nodes need not be given a spatial location nor do they need to be located on an element. Gap conditions may be defined in parallel (across the same nodes), with varying gap and stiffness values, to simulate a nonlinear (piecewise) force-deflection curve.

The gap direction is determined from the force label input on the **GP** command, i.e., **FX** defines a translational gap acting in the **UX** nodal degree of freedom direction, and **MZ** defines a rotational gap acting in the nodal **ROTZ** degree of freedom direction. The actual degree of freedom directions available for a particular node depends upon the degrees of freedom associated with the element types [**ET**] at that node.

If the coordinate systems of the nodes connecting the gap are rotated relative to each other, the same degree of freedom may be in different directions. The gap, however, assumes only a one-dimensional action. Nodes **I** and **J** may be anywhere in space (preferably coincident). No moment effects are included due to noncoincident nodes. That is, if the nodes are offset from the line of action, moment equilibrium may not be satisfied.

The contact stiffness value represents the stiffness of the closed gap. Stiffness values are related to the integration time step size and should be physically reasonable. High stiffness will require a small integration time step, otherwise, due to the displacement extrapolation, the solution may go unstable. Negative stiffness values may be used with gaps in parallel to produce a decreasing force-deflection curve.

The gap conditions, if any, should be defined in the first load step. Appearances in succeeding load steps are ignored. The order of specifying the gap nodes is important, i.e., a gap condition connecting two nodes will act differently depending upon which node is specified first on the **GP** command. For example, for Node 1 at  $x = 0.0$ , Node 2 at  $x = 0.1$ , and the gap defined from Node 1 to 2, a displacement of Node 1 greater than Node 2 will cause the gap to close. For the gap defined from Node 2 to 1, a displacement of Node 2

greater than Node 1 will cause the gap to close (like a hook action). In general, the gap closes whenever the separation (defined as  $U_J - U_I + GAP$ ) is negative.  $U_J$  is the displacement of node J,  $U_I$  is the displacement of node I, and  $GAP$  is the input gap value. The gap force output appears in the printout only for the time steps for which the gap is closed. A negative spring force is always associated with a closed gap (even with the hook option).

The nonlinear gap damping provided through the DAMP field runs faster than a full transient analysis using a gap element (COMBIN40). Only **ANTYPE** = TRANS and **TRNOPT** = MSUP allow the nonlinear gap damping action. Damping conditions are ignored for the reduced transient analysis method.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Dynamic Gap Cond>Define**

**Main Menu>Solution>Dynamic Gap Cond>Define**

**GPDELE**, *GAP1*, *GAP2*, *GINC*

**Deletes gap conditions.**

[SOLUTION: Gap Conditions](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

*GAP1*, *GAP2*, *GINC*

Delete gap conditions from *GAP1* to *GAP2* (defaults to *GAP1*) in steps of *GINC* (defaults to 1).

## Notes

Deletes gap conditions defined with the **GP** command. Gap conditions following those deleted are automatically compressed and renumbered. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Dynamic Gap Cond>Delete**

**Main Menu>Solution>Dynamic Gap Cond>Delete**

**GPLIST**, *GAP1*, *GAP2*, *GINC*

**Lists the gap conditions.**

[SOLUTION: Gap Conditions](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

*GAP1*, *GAP2*, *GINC*

List gap conditions from *GAP1* to *GAP2* (*GAP2* defaults to *GAP1*) in steps of *GINC* (defaults to 1). If *GAP1* = ALL (default), *GAP2* and *GINC* are ignored and all gap conditions are listed.

## Notes

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Loads>Dynamic Gap Cond>List All**  
**Main Menu>Preprocessor>Loads>Dynamic Gap Cond>List Specified**  
**Main Menu>Solution>Dynamic Gap Cond>List All**  
**Main Menu>Solution>Dynamic Gap Cond>List Specified**  
**Utility Menu>List>Other>Gap Conditions**

## GPLOT

Controls general plotting.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This command displays all entity types as specified via the **/GTYPE** command. Only selected entities (**NSSEL**, **ESEL**, **KSEL**, **LSEL**, **ASEL**, **VSEL**) will be displayed. See the descriptions of the **/GTYPE** and **/GCMD** commands for methods of setting the entity types displayed.

This command is valid in any processor.

## Menu Paths

**Utility Menu>Plot>Multi-Plots**

## **/GRAPHICS**, *Key*

Defines the type of graphics display.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **Key**

Graphics key:

#### **FULL**

Display all model geometry and results.

#### **POWER**

Activate PowerGraphics (default when GUI is on).

## Command Default

PowerGraphics ON (*key* = POWER).

## Notes

The **/GRAPHICS** command specifies the type of graphics display.  $KEY = POWER$  activates the PowerGraphics capability. PowerGraphics offers faster plotting than the  $KEY = FULL$  option, and speeds up element, results, area, line, and volume displays. PowerGraphics mode (the default) is automatically invoked when the GUI is accessed. This action supersedes all prior macros or start up routines (start.ans, config.ans, etc.). Full graphics mode can be accessed only by issuing **/GRAPHICS, FULL** after the GUI is active.

Results values (both printed and plotted) may differ between the  $KEY = FULL$  and  $KEY = POWER$  options because each option specifies a different set of data for averaging and display. For  $KEY = FULL$ , all element and results values (interior and surface) are included. For  $KEY = POWER$ , only element and results values along the model exterior surface are processed.

---

### Caution

*If you have specified one facet per element edge for PowerGraphics displays (via the **/EFACET** command or via choices from the General Postproc or Utility Menus), PowerGraphics does not plot midside nodes.*

The **/EFACET** command is only applicable to element type displays. (See the descriptions of these commands for more information.)

Maximum values shown in plots can differ from printed maximum values. This is due to different averaging schemes used for plotted and printed maximum values.

PowerGraphics displays do not average at geometric discontinuities. The printouts in PowerGraphics will, however, provide averaging information at geometric discontinuities if the models do not contain shell elements. Carefully inspect the data you obtain at geometric discontinuities.

---

### Note

In Full Graphics mode, it is possible to deselect an individual node, select all elements (including the element that contains that node), and then perform postprocessing calculations on those elements and have that unselected node not be considered in those calculations. However, if PowerGraphics is active, postprocessing always displays based on selected elements.

PowerGraphics does not support membrane shell elements such as **SHELL41**, and these elements using the membrane-stiffness-only option ( $KEYOPT1 = 1$ ): **SHELL181**, **SHELL208**, and **SHELL209**.

Commands that are not supported by PowerGraphics are listed below. These commands are executed using the  $KEY = FULL$  option, regardless of whether PowerGraphics is activated. Only certain options for **/CTYPE**, **/EDGE**, **/ESHAPE**, **\*GET**, **/PNUM**, **/PSYMB**, **SHELL**, and **\*VGET** are not supported by PowerGraphics. (See the descriptions of these commands for more information.)

<b>/CTYPE</b>	<b>ESYS</b>	<b>/PBF</b>	<b>PRETAB</b>	<b>/SSCALE</b>
<b>DNSOL</b>	<b>*GET</b>	<b>PLCRACK</b>	<b>PRSECT</b>	<b>/SHRINK</b>
<b>/EDGE</b>	<b>LAYER</b>	<b>PLETAB</b>	<b>PRVECT</b>	<b>TALLOW</b>
<b>ERNORM</b>	<b>/NORMAL</b>	<b>PLLS</b>	<b>/PSYMB</b>	<b>*VGET</b>
<b>ESORT</b>	<b>NSEL</b>	<b>PLSECT</b>	<b>SALLOW</b>	<b>*VPUT</b>
	<b>NSORT</b>	<b>/PNUM</b>	<b>SHELL</b>	

## Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

---

**/GRESUME**, *Fname*, *Ext*, --

Sets graphics settings to the settings on a file.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to `GSAV` if *Fname* is blank.

--

Unused field.

## Notes

Causes a file to be read to reset the graphics slash (/) commands as they were at the last **/GSAVE** command.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Restore Plot Ctrls

---

**/GRID**, *KEY*

Selects the type of grid on graph displays.

[GRAPHICS: Graphs](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *KEY*

Grid key:

**0 (OFF)**

No grid.

**1 (ON)**

Full grid (X and Y grid lines).

**2 (X)**

Partial grid (X grid lines only).

**3 (Y)**

Partial grid (Y grid lines only)

## Command Default

No grid.

## Notes

Selects the type of grid on graph displays. Graphs with multiple Y-axes can have multiple grids [/GRTYP]. The grid of the first curve is also used as the background grid (above and behind the curve). Grids for other curves are limited to be under the curves. See also /GTHK and /GROPT for other grid options.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Grid

---

**/GROPT**, *Lab*, *KEY*

Sets various line graph display options.

GRAPHICS: Graphs

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lab*

Apply display style as selected from the following labels:

**AXDV**

Axis division (tick) marks (defaults to *KEY* = ON).

**AXNM**

Axis scale numbers (defaults to *KEY* = ON, which puts numbers at the back plane of the graph). If *KEY* = FRONT, numbers are on the front plane of the graph.

**AXNSC**

Axis number size scale factor. Input the scale value for *KEY* (defaults to 1.0).

**ASCAL**

Automatic scaling of additional Y-axes for multi-curve [/GRTYP, 2 or 3] graphs (defaults to *KEY* = ON). If *KEY* = OFF, use base Y-axis scaling (see the /YRANGE command).

**LOGX**

Log X scale (defaults to *KEY* = OFF (linear)).

**LOGY**

Log Y scale (applies only to the base Y axis) (defaults to *KEY* = OFF (linear)).

**FILL**

Color fill areas under curves (defaults to *KEY* = OFF).

**CGRID**

Superimpose background grid [/G**GRID**] over areas under filled curves (defaults to *KEY* = OFF).

**DIG1**

Number of significant digits before decimal point for axis values. Input the value for *KEY* (defaults to 4).

**DIG2**

Number of significant digits after decimal point for axis values. Input the value for *KEY* (defaults to 3).

**VIEW**

View key for graph displays (defaults to *KEY* = OFF, in which case the view is (0,0,1) for 2-D graph displays or (1,2,3) for 3-D graph displays). If *KEY* = ON, the view settings for graph displays are the same as the view settings for the model.

**REVX**

Plots the values on the X-axis in reverse order.

**REVY**

Plots the values on the Y-axis in reverse order.

**DIVX**

Determines the number of divisions (grid markers) that will be plotted on the X axis.

**DIVY**

Determines the number of divisions (grid markers) that will be plotted on the Y axis.

**LTYP**

Determines whether ANSYS generated (*KEY* = 1) or system derived (*KEY* = 0) fonts will be used for the axis labels.

**CURL**

Determines the position of the curve labels. If (*KEY* = 1), the curve label will be plotted in the legend column, and the label will be displayed in the same color as the curve. If (*KEY* = 0) the curve labels will be plotted near the curve. (default).

**XAXO**

When you use this label, the subsequent *KEY* value will determine an offset amount from the default (along the bottom) location for the X axis. If *KEY* = 1.0, a full offset occurs (the X axis is moved to the top of the graph). If *KEY* = 0.5, the axis is offset to the midpoint of the graph, and if *KEY* = 0 the axis remains in the original position, along the bottom of the graph. For any offset, a grey copy of the original axis (containing the axis numbering) remains at the original location.

**YAXO**

When you use this label, the subsequent *KEY* value will determine an offset amount from the default (along the left side of the graph) location for the Y axis. If *KEY* = 1.0, a full offset occurs (the Y axis is moved to the right side of the graph). If *KEY* = 0.5, the axis is offset to the midpoint of the graph, and if *KEY* = 0 the axis remains in the original position, along the left side of the graph. For any offset, a gray copy of the original axis (containing the axis numbering) remains at the original location.

**KEY**

Option values:

**OFF (0)**

Do not apply selected style.

**ON (1)**

Apply selected style.

***nnnn***

If *Lab* is DIG1 or DIG2, input the number of digits.

***nn***

If *Lab* is AXNSC, input the scale factor.

**FRONT**

If *Lab* is AXNM, FRONT may also be input.

**Ndiv**

If *Lab* is DIVX or DIVY, determines the number of divisions (1-99) that will be applied to the axis.

**Kfont**

If *Lab* is LTYP, Kfont is ON (1) or OFF(0). ON will use ANSYS generated fonts for the axis labels, while OFF will use SYSTEM (Windows, X-system, etc.) fonts. The default value is ON (ANSYS fonts).

## Notes

Sets various line graph display options. Issue **/GROPT,STAT** to display the current settings. Issue **/GROPT,DEFA** to reset the default specifications. ANSYS informs you that graph view manipulation is inactive unless you have issued the **/GROPT,VIEW,ON** command. See the **/AXLAB**, **/GRTYP**, **/GRID**, and **/GTHK** commands for other graph control options.

Automatic scaling using the **/XRANGE** and **/YRANGE** commands will often yield inappropriate range values for logarithmic scales (**/GROPT, LOGX** or **/GROPT, LOGY**).

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

Utility Menu>PlotCtrls>Style>Graphs>Modify Curve

Utility Menu>PlotCtrls>Style>Graphs>Modify Grid

## GRP, *SIGNIF*, *Label*

**Specifies the grouping mode combination method.**

**SOLUTION: Spectrum Options**

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***SIGNIF***

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multipoint, or DDAM response (**SPOPT**,SPRS, MPRS or DDAM), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**,PSD.)

***Label***

Label identifying the combined mode solution output.



**DISP**

Displacement solution (default). Displacements, stresses, forces, etc., are available.

**VELO**

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

**ACEL**

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

**Notes**

The *SIGNIF* value set with this command (including the default value of 0.001) overrides the *SIGNIF* value set with the **MXPAND** command.

This command is also valid for PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine**

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine**

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine**

**Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine**

**Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine**

**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine**

**/GRTYP, KAXIS**

**Selects single or multiple Y-axes graph displays.**

[GRAPHICS: Graphs](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KAXIS**

Axis selection key:

**0 or 1**

Single Y-axis. Up to 10 curves scaled to a single Y-axis.

**2**

Additional Y-axes (one for each curve) (3 curves maximum). Allows better scaling of curves with widely differing numbering ranges.

**3**

Same as 2 but with additional Y-axis and curves projected out of the plane (6 curves maximum). Allows clearer display with an isometric view. The default view when *KAXIS* = 3 is View,1,1,2,3.

**Command Default**

Single Y-axis graph (except as noted for selection key 3).

**Notes**

The basic line graph has one or more curves plotted against the same Y and X axes. Multiple curve graphs can also be plotted with individual Y axes and the same X axis. The Y axis of the first curve is referred to as

the base Y-axis and the Y axes of the other curves as additional Y axes. Curves are numbered sequentially from 1 (the base curve) in the order in which they are displayed. See the [/AXLAB](#), [/GROPT](#), [/GRID](#), and [/GTHK](#) commands for other display options.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

---

**/GSAVE**, *Fname*, *Ext*, --

Saves graphics settings to a file for later use.

GRAPHICS: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to `GSAV` if *Fname* is blank.

--

Unused field.

## Notes

This command does not save all graphics settings, but only those that may be reset by the [/RESET](#) command. The database remains untouched. Use the [/GRESUME](#) command to read the file. Repeated use of the [/GSAVE](#) command overwrites the previous data on the file. The following commands are saved by [/GSAVE](#):

<a href="#">/ANGLE</a>	<a href="#">/DSCALE</a>	<a href="#">/GRTYP</a>	<a href="#">/PSF</a>	<a href="#">/VIEW</a>
<a href="#">/AXLAB</a>	<a href="#">/EDGE</a>	<a href="#">/GTHK</a>	<a href="#">/PSYMB</a>	<a href="#">/VSCALE</a>
<a href="#">/CLABEL</a>	<a href="#">/EFACET</a>	<a href="#">/GTYPE</a>	<a href="#">/RATIO</a>	<a href="#">/VUP</a>
<a href="#">/COLOR</a>	<a href="#">/ESHAPE</a>	<a href="#">/LIGHT</a>	<a href="#">/SHRINK</a>	<a href="#">/WINDOW</a>
<a href="#">/CONTOUR</a>	<a href="#">/FOCUS</a>	<a href="#">/NORMAL</a>	<a href="#">/SSCALE</a>	<a href="#">/XRANGE</a>
<a href="#">/CPLANE</a>	<a href="#">/GCMD</a>	<a href="#">/NUMBER</a>	<a href="#">/TRIAD</a>	<a href="#">/YRANGE</a>
<a href="#">/CTYPE</a>	<a href="#">/GLINE</a>	<a href="#">/PBC</a>	<a href="#">/TRLCY</a>	
<a href="#">/CVAL</a>	<a href="#">/GRID</a>	<a href="#">/PLOPTS</a>	<a href="#">/TYPE</a>	
<a href="#">/DIST</a>	<a href="#">/GROPT</a>	<a href="#">/PNUM</a>	<a href="#">/VCONE</a>	

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Save Plot Ctrls

**GSBDATA**, *LabZ*, *VALUEZ*, *LabX*, *VALUEX*, *LabY*, *VALUEY*

**Specifies the constraints or applies the load at the ending point for generalized plane strain option.**

SOLUTION: FE Constraints

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### ***LabZ***

Constraint or load at the ending point in the fiber Z direction.

#### **F**

Apply a force in the fiber direction (default).

#### **LFIBER**

Define a length change in the fiber direction.

### ***VALUEZ***

Value for *LabZ*. The default is zero.

### ***LabX***

Constraint or load on rotation about X.

#### **MX**

Supply a moment to cause the rotation of the ending plane about X (default).

#### **ROTX**

Define a rotation angle (in radians) of the ending plane about X.

### ***VALUEX***

Value for *LabX*. The default is zero.

### ***LabY***

Constraint or load on rotation about Y

#### **MY**

Supply a moment to cause the rotation of the ending plane about Y (default).

#### **ROTY**

Define a rotation angle (in radians) of the ending plane about Y.

### ***VALUEY***

Value for *LabY*. The default is zero.

## Notes

All inputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see [Generalized Plane Strain Option of Current-Technology Solid Elements](#) in the *Element Reference*.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Gen Plane Strain  
Main Menu>Solution>Define Loads>Apply>Structural>Gen Plane Strain

---

**GSGDATA**, *LFIBER*, *XREF*, *YREF*, *ROTX0*, *ROTY0*

Specifies the reference point and defines the geometry in the fiber direction for the generalized plane strain element option.

[PREP7: Meshing](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### ***LFIBER***

Fiber length from the reference point. Defaults to 1.

### ***XREF***

X coordinate of the reference point. Defaults to zero.

### ***YREF***

Y coordinate of the reference point. Defaults to zero.

### ***ROTX0***

Rotation of the ending plane about X in radians Defaults to zero.

### ***ROTY0***

Rotation of the ending plane about Y in radians Defaults to zero.

## Notes

The ending point is automatically determined from the starting (reference) point and the geometry inputs. All inputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see [Generalized Plane Strain Option of Current-Technology Solid Elements](#) in the *Element Reference*.

## Menu Paths

Main Menu>Preprocessor>Modeling>Genl plane strn

---

**GSLIST**, *Lab*

When using generalized plane strain, lists the input data or solutions.

[SOLUTION: FE Constraints](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### ***Lab***

Specify the content to be listed.

#### **GEOMETRY**

List the data input using GSGDATA

#### **BC**

List the data input using GSBDATA.

**REACTIONS**

When the command is issued in POST1, list the reaction force at the ending point, and the moment about X and Y if the corresponding constraints were applied.

**RESULTS**

When the command is issued in POST1, list the change of fiber length at the ending point during deformation and the rotation of the ending plane about X and Y during deformation.

**ALL**

List all of the above (default).

**Notes**

This command can be used to list the initial position of the ending plane, the applied load or displacements in the fiber direction, the resulting position of the ending plane after deformation, and the available reaction forces and moments at the ending point.

All inputs and outputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see [Generalized Plane Strain Option of Current-Technology Solid Elements](#) in the *Element Reference*.

This command is valid in any processor.

**Menu Paths**

Utility Menu>List>Other>Genl Plane Strn

**GSSOL**, *NVAR*, *Item*, *Comp*, *Name*

**Specifies which results to store from the results file when using generalized plane strain.**

POST26: Set Up

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***NVAR***

Arbitrary reference number or name assigned to this variable. Variable numbers can be 2 to *NV* (**NUMVAR**) while the name can be an eight byte character string. Overwrites any existing results for this variable.

***Item***

Label identifying item to be stored.

**LENGTH**

Change of fiber length at the ending point.

**ROT**

Rotation of the ending plane during deformation.

**F**

Reaction force at the ending point in the fiber direction.

**M**

Reaction moment applied on the ending plane.

***Comp***

Component of the item, if Item = ROT or M.

**X**

The rotation angle or reaction moment of the ending plane about X.

**Y**

The rotation angle or reaction moment of the ending plane about Y.

**Name**

Thirty-two character name identifying the item on the printout and display. Defaults to the label formed by concatenating the first four characters of the *Item* and *Comp* labels.

## Notes

This command stores the results (new position of the ending plane after deformation) for generalized plane strain. All outputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see [Generalized Plane Strain Option of Current-Technology Solid Elements](#) in the *Element Reference*.

## Menu Paths

**Main Menu>TimeHist Postpro>Variable Viewer**

---

**/GST**, *Lab*, *Lab2*

**Turns Graphical Solution Tracking (GST) on or off.**

[SOLUTION: Nonlinear Options](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

**Lab**

Determines whether the Graphical Solution Tracking feature is active. Specify ON to activate GST, or OFF to deactivate the feature.

**Lab2**

Activates generation of interface and field convergence files (ANSYS MFX analyses only).

## Notes

For interactive runs using GUI [**/MENU**,ON] or graphics [**/MENU**,GRPH] mode, ANSYS directs GST graphics to the screen. For interactive sessions not using GUI or graphics mode, or for batch sessions, GST graphics are saved in the ANSYS graphics file *Jobname.GST* when *Lab2* is unspecified. The file *Jobname.GST* can be viewed with the DISPLAY program in this case. You must select **All File Types** to access it. For more information on the DISPLAY program see [Getting Started with the DISPLAY Program](#) in the *Basic Analysis Guide*. For MFX runs (when *Lab2*=ON), the *Jobname.GST* file is in XML format, and it can be viewed with the **Results Tracker Utility**, accessed from within the **Tools** menu of the Mechanical APDL Product Launcher.

The GST feature is available only for nonlinear structural, thermal, electric, magnetic, fluid, or CFD simulations. For more information about this feature and illustrations of the GST graphics for each analysis type, see the *ANSYS Analysis Guide* for the appropriate discipline. See also the **CNVTOL** command description.

The GST feature is also available for a p-method electrostatic analysis. For interactive sessions with the p-method electrostatic preference set, ANSYS directs the GST graphics to the screen.

When running an **ANSYS MFX analysis**, specify **/GST,ON,ON** to generate both the interface (**Jobname.NLH**) and field convergence (**Fieldname.GST**) files for monitoring the analysis. This field is not available on the GUI.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Grph Solu Track**  
**Main Menu>Solution>Load Step Opts>Output Ctrls>Grph Solu Track**

## GSUM

Calculates and prints geometry items.

PREP7: Keypoints  
 PREP7: Lines  
 PREP7: Areas  
 PREP7: Volumes

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Calculates and prints geometry items (centroid location, moments of inertia, length, area, volume etc.) associated with the selected keypoints, lines, areas, and volumes. Geometry items are reported in the global Cartesian coordinate system. For volumes, a unit density is assumed unless the volumes have a material association via the **VATT** command. For areas, a unit density (and thickness) is assumed unless the areas have a material (and real constant) association via the **AATT** command. For lines and keypoints, a unit density is assumed, irrespective of any material associations [**LATT**, **KATT**, **MAT**]. Items calculated by **GSUM** and later retrieved by a **\*GET** or **\*VGET** commands are valid only if the model is not modified after the **GSUM** command is issued. This command combines the functions of the **KSUM**, **LSUM**, **ASUM**, and **VSUM** commands.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Geometry**

**/GTHK**, *Label*, *THICK*

Sets line thicknesses for graph lines.

GRAPHICS: Graphs

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Label*

Apply thicknesses as selected from the following labels:

#### **AXIS**

Modify thickness of ordinate and abscissa axes on graph displays.

#### **GRID**

Modify thickness of grid lines on graph displays.

#### **CURVE**

Modify thickness of curve lines (when no area fill [**/GROPT**]).

**THICK**

Thickness ratio (whole numbers only, from -1 to 10):

**-1**

Do not draw the curve, but show only the markers specified by **/GMARKER**.

**0 or 1**

Thin lines.

**2**

The default thickness.

**3**

1.5 times the default thickness.

**etc.**

(up to 10)

## Notes

Sets line thicknesses for graph lines (in raster mode only). Use **/GTHK,STAT** to show settings.

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Style>Graphs>Modify Axes**

**Utility Menu>PlotCtrls>Style>Graphs>Modify Curve**

**Utility Menu>PlotCtrls>Style>Graphs>Modify Grid**

---

**/GTYPE**, *WN*, *LABEL*, *KEY*

**Controls the entities that the **GPLOT** command displays.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WN**

Window number (or ALL) to which this command applies (defaults to 1)

**LABEL**

This represents the type of entity to display:

**NODE**

Nodes

**ELEM**

Elements

**KEYP**

Keypoints

**LINE**

Lines

**AREA**

Areas



**VOLU**

Volumes

**GRPH**

Graph displays

**KEY**

Switch:

**0**

Turns the entity type off.

**1**

Turns the entity type on.

**Notes**

The **/GTYPE** command controls which entities the **GPLOT** command displays. NODE, ELEM, KEYP, LINE, AREA, and VOLU are on by default. When ELEM is activated, you can control the type of element displayed via the **/GCMD** command (which also controls the type of graph display). When the GRPH entity type is activated, all other entity types are deactivated. Conversely, when any of the NODE, ELEM, KEYP, LINE, AREA, and VOLU entity types are active, the GRPH entity type is deactivated.

The **/GTYPE** command gives you several options for multi-window layout:

- One window
- Two windows (left and right or top and bottom of the screen)
- Three windows (two at the top and one at the bottom of the screen, or one top and two bottom windows)
- Four windows (two at the top and two at the bottom)

Once you choose a window layout, you can choose one of the following: multiple plots, replotting, or no redisplay.

This command is valid in any processor.

**Menu Paths****Utility Menu>PlotCtrls>Multi-Plot Contrls**



---

## H Commands

### **HARFRQ**, *FREQB*, *FREQE*

Defines the frequency range in the harmonic response analysis.

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### ***FREQB***

Frequency (Hz) at the beginning of the *FREQB* to *FREQE* range (if *FREQE* > *FREQB*). If *FREQE* is blank, the solution is done only at frequency *FREQB*.

#### ***FREQE***

Frequency at end of this range. Solutions are done at an interval of  $(FREQE-FREQB)/NSBSTP$ , ending at *FREQE*. No solution is done at the beginning of the frequency range. *NSBSTP* is input on the **NSUBST** command. See **EXPSOL** for expansion pass solutions.

## Notes

Defines the frequency range for loads in the harmonic response analysis (**ANTYPE**,HARMIC).

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substps**  
**Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substps**

---

### **/HBC**, *WN*, *Key*

Determines how boundary condition symbols are displayed in a display window.

[GRAPHICS: Labeling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ***WN***

Window reference number. This number can be any window numbered 1 to 5, or ALL (for all active windows). Defaults to 1

#### ***Key***

Key to enable/disable hidden surface boundary condition symbol display for 2-D graphics devices and to request improved pressure contour display for 2-D and 3-D devices: *Key* = ON, YES or 1 will show your BC symbols on the hidden surfaces and use an improved pressure contour display. *Key* = OFF, NO or 0 (default) will hide the symbols .

## Command Default

**/HBC**,WN,Off

For 2-D graphics devices (X11, win32, PNG, et al.), boundary condition symbols are NOT drawn to the hidden surface display. For both 2-D and 3-D graphics devices, no enhanced rendering of pressure contours occurs.

## Menu Paths

**Utility Menu>PlotCtrls>Style>Hidden-Line Options**

---

**HBMAT**, *Fname*, *Ext*, *--*, *Form*, *Matrx*, *Rhs*, *Mapping*

**Writes an assembled global matrix in Harwell-Boeing format.**

[AUX2: Binary Files](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Defaults to the current Jobname if left blank.

### ***Ext***

Filename extension (8 character maximum).

Defaults to `.MATRIX` if left blank.

**--**

Unused field.

### ***Form***

Specifies format of output matrix file:

#### **ASCII**

Write output matrix file in ASCII form.

#### **BIN**

Write output matrix file in binary form.

### ***Matrx***

Specify which matrix to write to the output matrix file:

#### **STIFF**

Write stiffness matrix to output matrix file. Valid for all types of analyses that write a `.FULL` file.

#### **MASS**

Write mass matrix to output matrix file. Valid for buckling, substructure, and modal analyses. If `.FULL` file was generated in a buckling analysis, then this label will write stress stiffening matrix to output matrix file.

#### **DAMP**

Write damping matrix to output matrix file. Only valid for damped modal analyses.

**Rhs**

Specifies whether to write the right-hand side vector to output matrix file:

**YES**

Write right-hand side vector to output matrix file

**NO**

Do not write right-hand side vector to output matrix file

**Mapping**

Specifies whether to write the mapping file. This file is always named *Fname*.MAPPING.

**YES**

Write the mapping file

**NO**

Do not write the mapping file (default)

## Command Default

By default, assuming a proper filename and extension have been entered, writes the stiffness matrix and right-hand side vector to `jobname.MATRIX` in the current working directory in ASCII format.

## Notes

This command is used to copy a matrix from the assembled global matrix file (`.FULL` file) or from the super-element matrix file (`.SUB` file) as specified on the **FILEAUX2** command and write it in Harwell-Boeing format to a new file, named `jobname.MATRIX`. The Harwell-Boeing format is widely used by other applications that deal with matrices.

The assembled global matrix file is created during solution depending on the analysis type, equation solver, and other solution options. By default, the assembled global matrix file is never deleted at the end of solution. For most analysis types, the Sparse direct solver, the ICCG solver, and the AMG solver (when available) will write a `.FULL` file. With the exception of the reduced (**MODOPT,REDUC**) and Variational Technology (**MODOPT,VT**) options, all mode extraction methods used for buckling and modal analyses will write a properly formatted `.FULL` file to be used with the **HBMAT** command.

When dumping the stiffness matrix for transient and harmonic analyses, be aware that the element mass matrix values (and possibly element damping matrix values) are incorporated into the globally assembled stiffness matrix. Thus, the globally assembled stiffness matrix represents more than the stiffness of the model for these analysis types. Please refer to the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for more details.

When dumping a `.FULL` file, the rows and columns corresponding to specified constraints (e.g., **D** commands) are eliminated from the system of equations and therefore are not written to the `.MATRIX` file. Also, rows and columns corresponding to eliminated (slave) degrees of freedom from coupling and/or constraint equations (e.g., **CE**, **CP** commands) are also eliminated from the system of equations and are not written to the `.MATRIX` file. The DOFs that are eliminated from any coupling and/or constraint equations are determined internally by the solution code and may not match what you specified via the **CE/CP** (or similar) commands.

When dumping a `.SUB` file, the full  $n \times n$  matrix will be written to the `.MATRIX` file for either symmetric or unsymmetric matrices, regardless of whether any of the matrix coefficients are zero-valued. When dumping a `.FULL` file, only the lower triangular part of the matrix will be written to the `.MATRIX` file if the matrix is symmetric; the full matrix is written if the matrix is unsymmetric. Only matrix coefficients that are greater than zero will be written.

The Harwell-Boeing format is column-oriented. That is, non-zero matrix values are stored with their corresponding row indices in a sequence of columns. However, since the ANSYS matrix files are stored by row and not column, when the **HBMAT** command is used with a non-symmetric matrix, the transpose of the matrix is, in fact, written.

The **WRFULL** command, in conjunction with the **SOLVE** command, can be used to generate the assembled global matrix file and eliminate the equation solution process and results output process.

The mapping file can be used to map the matrix equation numbers found in the `.MATRIX` file directly to the corresponding node numbers and degrees of freedom.

When dumping a CMS `.SUB` file, the last rows/columns of the matrix are non-physical degrees of freedom added internally by the **CMS process** and cannot be mapped directly to a node number or particular degree of freedom.

## Menu Paths

Utility Menu>File>List>Binary Files  
Utility Menu>List>Files>Binary Files

---

**/HEADER**, *Header, Stitle, Idstmp, Notes, Colhed, Minmax*

**Sets page and table heading print controls.**

[POST1: Listing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **Header**

ANSYS page header (system, date, time, version, copyright, title, etc.):

#### **ON**

Turns this item on (default for batch mode; not available for interactive mode).

#### **OFF**

Turns this item off.

#### **(blank)**

Retains the previous setting.

### **Stitle**

Subtitles (see **/STITLE** command): ON, OFF, or (blank) (see above).

### **Idstmp**

Load step information (step number, substep number, time value): ON, OFF, or (blank) (see above).

### **Notes**

Information relative to particular table listings: ON, OFF, or (blank) (see above).

### **Colhed**

Column header labels of table listings (currently only for single column tables): ON, OFF, or (blank) (see above).

### **Minmax**

Minimum/maximum information or totals after table listings: ON, OFF, or (blank) (see above).

## Command Default

All specifications are on (batch mode); *Header* = OFF, all other specifications are on (interactive mode)

## Notes

Sets specifications on or off for page and table heading print controls associated with the POST1 **PRNSOL**, **PRESOL**, **PRETAB**, **PRRSOL**, and **PRPATH** commands.

---

### Note

If the printout caused a top-of-form (page eject to top of next page), the top-of-form is also suppressed with the printout. Issue **/HEADER,STAT** to display the current settings. Issue **/HEADER,DEFA** to reset the default specifications.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## HELP, *Name*

Displays help information on ANSYS commands and element types.

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Name*

Command name or element type. Examples are: **HELP,MP** or **HELP,SOLID185** (or **HELP,185**). For a list of elements of a particular type, enter **HELP,BEAM**, **HELP,SOLID**, **HELP,HYPER**, etc.

## Notes

If *Name* uniquely matches a command or element name, the description for that command or element will be displayed in the Help Window. For command help, you must type the complete command name (including the \* or /). The help system cannot find partial matches. If \* is used at the beginning of the string, it will be interpreted as an ANSYS \* command.

For help on topics that are not ANSYS commands or element types (for example, help for the word "material"), use the index or full text search features of the ANSYS online help system.

The **HELP** command is valid only in GUI mode. To obtain help when not in GUI mode, you can either activate the GUI by typing **/MENU,ON**, or you can activate the help system directly by issuing **/UI,HELP**.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

**HELPDISP**, *Commandname*

Displays help information on DISPLAY program commands.

[DISPLAY: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Commandname**

Any DISPLAY command. If blank, a list of DISPLAY commands is produced.

**Menu Paths**

It is part of the DISPLAY command.

---

**HEMIOPT**, *HRES*

Specifies options for Hemicube view factor calculation.

[SOLUTION: Radiosity](#)

[AUX12: Radiosity Solver](#)

MP ME <> PR PRN <> <> <> <> <> <> PP <> EME MFS

**HRES**

Hemicube resolution. Increase value to increase the accuracy of the view factor calculation. Defaults to 10.

**Menu Paths**

**Main Menu>Preprocessor>Radiation Opts>View Factor**

**Main Menu>Radiation Opt>Radiosity Meth>View Factor**

**Main Menu>Solution>Radiation Opts>View Factor**

---

**HFADP**, *Lab*

Turns a high-frequency adaptive error calculation on or off.

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**Lab**

Adaptive error label:

**ON**

Turns on adaptive error calculation.

**OFF**

Turns off adaptive error calculation.

**Notes**

Adaptive error calculation must be turned on to automatically refine the mesh with the **HFEREFINE** macro.



This command is also valid in PREP7.

## Menu Paths

This command cannot be accessed from a menu.

### HFANG, *Lab*, *PHI1*, *PHI2*, *THETA1*, *THETA2*

Defines or displays spatial angles of a spherical radiation surface for antenna parameter calculations.

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

#### *Lab*

Spatial angle label.

#### ANGLE

Define spatial angles (default).

#### STATE

Display spatial angles. PHI1, PHI2, THETA1, and THETA2 are ignored.

#### *PHI1*, *PHI2*

Starting and ending  $\phi$  angles (degrees) in the spherical coordinate system. Defaults to 0.

#### *THETA1*, *THETA2*

Starting and ending  $\theta$  angles (degrees) in the spherical coordinate system. Defaults to 0.

## Notes

Defines or displays spatial angles of a spherical radiation surface. See [Spherical Coordinates](#) in the *High-Frequency Electromagnetic Analysis Guide*. Use this command only with **PLFAR**,*Lab* = ANT or **PRFAR**,*Lab* = ANT.

## Menu Paths

Main Menu>General Postproc>List Results>Field Extension>Direct Gain

Main Menu>General Postproc>List Results>Field Extension>Efficiency

Main Menu>General Postproc>List Results>Field Extension>Power Gain

Main Menu>General Postproc>List Results>Field Extension>Rad Power

Main Menu>General Postproc>Plot Results>Field Extension>Direct Gain

### HFARRAY, *NUMX*, *NUMY*, *PX*, *PY*, *SKEW*, *PHASEX*, *PHASEY*

Defines phased array antenna characteristics.

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

#### *NUMX*

Number of array units in X-direction (defaults to 1).

**NUMY**

Number of array units in Y-direction (defaults to 1).

**PX**

Spatial periodicity in X-direction in meters.

**PY**

Spatial periodicity in Y-direction in meters.

**SKEW**

Skew angle for triangular periodicity from X-axis to Y-axis in degrees (defaults to 90).

**PHASEX**

Initial phase angle difference between array units in X-direction in degrees (defaults to 0).

**PHASEY**

Initial phase angle difference between array units in Y-direction in degrees (defaults to 0).

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>General Postproc>List Results>Field Extension>Direct Gain**

**Main Menu>General Postproc>List Results>Field Extension>Far Field**

**Main Menu>General Postproc>List Results>Field Extension>Pattern**

**Main Menu>General Postproc>Plot Results>Field Extension>Direct Gain**

**Main Menu>General Postproc>Plot Results>Field Extension>Far Field**

**Main Menu>General Postproc>Plot Results>Field Extension>Pattern**

---

**HFDEEM**, *Filename*, *Snp*, *PORTNUM1*, *L1*, *Filename1*, *Ext1*, *PORTNUM2*, *L2*, *Filename2*, *Ext2*

**Calibrates S-parameter phase shift.**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**Filename**

A Touchstone file containing the S-parameters of a n-port system.

**Snp**

Extension of the n-port Touchstone file.

**PORTNUM1**

Port number of excitation port. Defaults to 1.

**L1**

Distance from the S-parameter extraction plane to the reference plane at PORTNUM1 in meters. Defaults to 0.

**Filename1**

A two-column Touchstone format file containing frequencies and propagating constants.

**Ext1**

Extension of the Filename1 file.

**PORTNUM2**

Port number of output port. Defaults to 1.

**L2**

Distance from the S-parameter extraction plane to the reference plane at PORTNUM2 in meters. Defaults to 0.

**Filename2**

A two-column Touchstone format file containing frequencies and propagating constants.

**Ext2**

Extension of the Filename2 file.

## Notes

The first line of the Touchstone format files Filename1 and Filename2 is a general description of the output data and it starts with an exclamation point (!). The second line starts with a # and it is followed by a frequency unit of GHz, MHz, KHz, or Hz (for example, #GHz). The first column of the two-column format file is the frequency. The second column is the propagating constant corresponding to the working frequency.

## Menu Paths

**This command cannot be accessed from a menu.**

## HFEIGOPT, Lab, Val1

**Specifies high frequency electromagnetic modal analysis options.**

**SOLUTION: Analysis Options**

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**Lab**

High frequency modal analysis type:

**CAVITY**

Perform a 3-D eigenvalue analysis (default for HF119 and HF120).

**CUTOFF**

Perform a 2-D cutoff frequency analysis (default for HF118).

**GAMMA**

Perform a 2-D propagating constant analysis.

**Val1**

Valid for the GAMMA option only. If Lab = GAMMA, Val1 is the frequency in Hz.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>Analysis Options**

**HFEREFINE**, *FACTOR*, *NUMLIST*

Automatically refines high-frequency tetrahedral elements (**HF119**) or lists high-frequency brick elements (**HF120**) with the largest error.

[PREP7: Special Purpose](#)

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**FACTOR**

Percentage applied in adaptive mesh refinement scheme. This percentage of the elements with local error greater than the average error will be refined. *FACTOR* should be in the range of 0 to 100 percent and it defaults to 10 percent. Used only with **HF119** elements.

**NUMLIST**

Number of brick elements to be listed. Defaults to 25. Used only with **HF120** elements.

**Notes**

For **HF119** elements, **HFEREFINE** automatically refines the mesh so that the measured discretization error will decrease. Execution deletes all boundary conditions and excitation sources on the finite element model.

For **HF120** elements, **HFEREFINE** provides a list of elements with the largest error.

**Menu Paths**

Main Menu>Preprocessor>Meshing>Modify Mesh>HF Refine

---

**HFMODPRT**, *FREQ*

Calculates electromagnetic field distribution for a modal port.

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**FREQ**

Working frequency (Hz). No default.

**Notes**

This command automatically generates a port electromagnetic field by solving the 2-D eigenvalue problem and stores the solution as 3-D harmonic excitation and matching condition.

This command must be issued in **SOLUTION** when a modal port is defined and the **SPSWP** command is not used.

**Menu Paths**

This command cannot be accessed from a menu.

---

**HFPA**, *Lab*, *Local*, *VAL1*, *VAL2*

**Specifies a radiation scan angle for a phased array antenna analysis.**

SOLUTION: Analysis Options

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**Lab**

Enter SCAN in this field.

**Local**

Local coordinate system number (defaults to 0).

**VAL1**

Angle from x-axis towards y-axis,  $\phi$ , in degrees (defaults to 0).

**VAL2**

Angle from +z-axis towards -z-axis,  $\theta$ , in degrees (defaults to 0).

## Notes

See Figure 4.23: "Spherical Coordinates" in the *High-Frequency Electromagnetic Analysis Guide* for an illustration of the coordinate system.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

**HFPCSWP**, *FREQB*, *FREQE*, *FREQINC*, *NUMMODE*, *IMPDMULT*, *ZOPT*

**Calculates propagating constants and characteristic impedance of a transmission line or waveguide over a frequency range.**

SOLUTION: Analysis Options

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**FREQB**

Beginning frequency of the analysis (in Hertz).

**FREQE**

Ending frequency of the analysis (in Hertz). If blank, *FREQE* is set to *FREQB* and one solution is performed. (This is recommended for initially verifying the model setup.)

**FREQINC**

Frequency increment between solutions. If *FREQINC* is blank, solutions are performed at *FREQB* and *FREQE* only (default).

**NUMMODE**

Number of required modes. Defaults to 1.

**IMPDMULT**

Impedance multiplier (defaults to 1). For a half symmetry model, use 2.

**ZOPT**

Impedance calculation inputs:

- 0** Power and current (PI) definition (default).
- 1** Power and voltage (PV) definition.
- 2** Voltage and current (VI) definition.

**Notes**

**HFPCSWP** calculates propagating constants and characteristic impedance for a transmission line or waveguide over a frequency range from *FREQB* to *FREQE*, in increments of *FREQINC*. The output data is written to the file *HFPCSWP.OUT*. Use the **PLTLINE** command to plot the results.

**HFPCSWP** can only be used with **HF118** elements.

**Menu Paths**

**Main Menu>Solution>Solve>Electromagnet>HF Emag>2D Freq Sweep**

---

**HFPOWER**, *ARG1*, *ARG2*

**Calculates power terms of a multi-port network.**

POST1:Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**ARG1**

Excitation source port number.

**ARG2**

Matched output port number.

**Notes**

**HFPOWER** calculates the input power, reflected power, return loss, and power reflection coefficient for a driven port. If a matching output port is defined, it also calculates transmitted power, insertion loss and the power transmission coefficient. For lossy materials and conducting surfaces, **HFPOWER** also calculates the time-averaged dissipated power. The elements must be selected to calculate the dissipated power.

If *ARG1* and *ARG2* are both blank, no ports are specified and only the dissipated power is calculated.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>General Postproc>Elec&Mag Calc>Element Based>Power**

---

**HFPORT**, *Portnum*, *Porttype*, *Local*, *Opt1*, *Opt2*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*, *VAL10*, *VAL11*

**Specifies input data for waveguide, plane wave, lumped gap, modal, and modal lumped gap ports.**

SOLUTION: Misc Loads

MP <> <> <> <> <> <> <> <> EH <> PP <> <>

***Portnum***

Port number. You specify the integer number for exterior and interior ports by the **SF** and **BF** family of commands, respectively. The number must be between 1 and 50. If *Portnum* = STATUS, provides the status of port option settings.

***Porttype***

Port type:

**COAX**

Cylindrical coaxial waveguide.

**RECT**

Rectangular waveguide.

**CIRC**

Circular waveguide.

**PARA**

Parallel plate waveguide.

**PLAN**

Incident plane wave.

**LGAP**

Lumped gap port.

**MODAL**

Modal port.

**MGAP**

Modal lumped gap port.

***Local***

A previously defined local Cartesian coordinate system number (>10) or 0 (global Cartesian) used to specify the geometric properties of the waveguide. Defaults to 0. The local Z-direction must be the direction of wave propagation. The origin of the local coordinate system must be centered about the face of the waveguide port without considering symmetry. A local Cartesian coordinate system number is not necessary for *Porttype* = MODAL and MGAP.

***Opt1***

For *Porttype* = COAX, RECT, CIRC, PARA, and LGAP, *Opt1* defines the mode type:

**TEM**

Transverse electromagnetic wave. See note below.

**TE<sub>mn</sub>**

Transverse electric wave. See note below.

**TM<sub>mn</sub>**

Transverse magnetic wave. See note below.

**Note**

For *Porttype* = COAX, only the TEM mode is available. For *Porttype* = LGAP, *Opt1* is not used.

For *Porttype* = PLAN, *Opt1* defines the extracted harmonic component:

**ALL**

No specified extraction (default).

**TE**

Transverse electric wave.

**TM**

Transverse magnetic wave.

For *Porttype* = MODAL, *Opt1* is the required number of modes. Defaults to 1.

For *Porttype* = MGAP, *Opt1* is equal to 1 and defaults to 1.

**Opt2****EXT**

Exterior port.

**INT**

Interior port.

For *Porttype* = PLAN, LGAP, and MGAP, only *Opt2* = INT is available.

**VAL1, VAL2, VAL3, . . . , VAL11**

If *Porttype* = COAX:

**VAL1**

Inner radius of the coaxial waveguide.

**VAL2**

Outer radius of the coaxial waveguide.

**VAL3**

Zero-to-peak amplitude of the voltage between the inner and the outer conductors. If blank, the port will create a matching impedance.

**VAL4**

Phase angle of the applied voltage (in degrees). Defaults to 0 degrees.

**VAL5**

Input power (time-average). If power is input, it overrides the applied voltage input.

**VAL6**

Distance from S-parameter extraction plane to reference plane. Defaults to 0.

**VAL7-VAL11**

Not used.

If *Porttype* = RECT:

**VAL1**

Width of the rectangular waveguide.



**VAL2**

Height of the rectangular waveguide.

**VAL3**

Zero-to-peak amplitude of the electric field component  $E_z$  for a TM wave or the magnetic field component  $H_z$  for a TE wave. If blank, the port will appear as a matching impedance.

**VAL4**

Phase angle of the applied field (in degrees). Defaults to 0 degrees.

**VAL5**

Input power (time-average). If power is input, it overrides the field component input.

**VAL6**

Distance from S-parameter extraction plane to reference plane. Defaults to 0.

**VAL7-VAL11**

Not used.

If *Porttype* = CIRC

**VAL1**

Radius of the circular waveguide.

**VAL2**

Not used.

**VAL3**

Zero-to-peak amplitude of the electric field component  $E_z$  for a TM wave or the magnetic field component  $H_z$  for a TE wave. If blank, the port will appear as a matching impedance.

**VAL4**

Phase angle of the applied field (in degrees). Defaults to 0 degrees.

**VAL5**

Input power (time-average). If power is input, it overrides the field component input.

**VAL6**

Distance from S-parameter extraction plane to reference plane. Defaults to 0.

**VAL7-VAL11**

Not used.

If *Porttype* = PARA:

**VAL1**

Width of the parallel plate waveguide (defaults to 1).

**VAL2**

Separation between the two plates.

**VAL3**

Zero-to-peak amplitude of the electric field component  $E_y$  for a TEM wave, electric field component  $E_z$  for a TM wave, or the magnetic field component  $H_z$  for a TE wave. If blank, the port will appear as a matching impedance.

**VAL4**

Phase angle of the applied field (in degrees). Defaults to 0 degrees.

**VAL5**

Input power (time-average). If power is input, it overrides the field component input.

**VAL6**

Distance from S-parameter extraction plane to reference plane. Defaults to 0.

**VAL7-VAL11**

Not used.

If *Porttype* = PLAN:

**VAL1**

X Component of incident planar wave (V/m) in local coordinate system. Defaults to 0.

**VAL2**

Y Component of incident planar wave (V/m) in local coordinate system. Defaults to 0.

**VAL3**

Z Component of incident planar wave (V/m) in local coordinate system. Defaults to 0.

**VAL4**

Angle between incident wave vector and X-axis ( $\phi$ ).

**VAL5**

Angle between incident wave vector and Z-axis ( $\theta$ ).

**VAL6**

Distance from S-parameter extraction plane to reference plane. Defaults to 0.

**VAL7**

Extracted wave number in X direction (m in TEMn or TMmn). Defaults to 0.

**VAL8**

Extracted wave number in Y direction (n in TEMn or TMmn). Defaults to 0.

**VAL9**

Spatial periodicity in X direction in meters. It must be defined if the m value specified by *VAL7* is not equal to zero. Defaults to 0.

**VAL10**

Spatial periodicity in Y direction in meters. It must be defined if the n value specified by *VAL8* is not equal to zero. Defaults to 0.

**VAL11**

Skew angle for triangular periodicity from X-axis to Y-axis in degrees. Defaults to 90.

See [Figure 4.19: "Spherical Coordinates"](#) in the *High-Frequency Electromagnetic Analysis Guide* for an illustration of the coordinate system.

---

**Note**

VAL1, VAL2, and VAL3 = 0 for *Porttype* = PLAN and *Opt2* = INT.

If *Porttype* = LGAP:

**VAL1**

Width of the lumped gap port.

**VAL2**

Separation between the central conductor and ground.

**VAL3**

Zero-to-peak amplitude of the voltage between the central conductor and ground. Defaults to 0.

**VAL4**

Phase angle of the applied voltage (in degrees). Defaults to 0 degrees.

**VAL5**

Input power (time-average). If power is input, it overrides the field component input.

**VAL6**

Distance from S-parameter extraction plane to reference plane. Defaults to 0.

**VAL7**

Not used.

**VAL8**

Characteristic impedance of lumped gap port. Defaults to 50 ohm.

**VAL9-VAL11**

Not used.

If *Porttype* = MODAL:

**VAL1**

Impedance multiplier (defaults to 1). For a half symmetry model, use 2.

**VAL2**

Impedance calculation inputs:

**PI**

Power and current (PI) definition (default).

**PV**

Power and voltage (PV) definition.

**VI**

Voltage and current (VC) definition.

**VAL3**

Zero-to-peak amplitude of the electric field. Defaults to 0.

**VAL4**

Phase angle of the applied electric field (in degrees). Defaults to 0 degrees.

**VAL5**

Input power (time-average). If power is input, it overrides the field component input.

**VAL6**

Distance from S-parameter extraction plane to reference plane. Defaults to 0.

**VAL7-VAL11**

Not used.

If *Porttype* = MGAP:

**VAL1**

Not used.

**VAL2**

Not used.

**VAL3**

Zero-to-peak amplitude of the voltage between the central conductor and ground. Defaults to 0.

**VAL4**

Phase angle of the applied voltage (in degrees). Defaults to 0 degrees.

**VAL5**

Input power (time-average). If power is input, it overrides the field component input.

**VAL6**

Distance from S-parameter extraction plane to reference plane. Defaults to 0.

**VAL7**

Not used.

**VAL8**

Characteristic impedance of lumped gap port. Defaults to 50 ohm.

**VAL9-VAL11**

Not used.

## Notes

The origin of the local coordinate system must be at the center of the waveguide structure. For a rectangular waveguide, the X and Y axes of the local coordinate system must be parallel to the width and height of the waveguide, respectively. For a parallel plate waveguide or a lumped gap port, the X and Y axes of the local coordinate system must be parallel to the width and separation, respectively.

The following apply to the TEM<sub>n</sub> and TM<sub>m</sub>n mode types:

- For a rectangular waveguide, the suffix m and n mean the variation of the field along the wide side and narrow side of the waveguide, respectively.
- For a circular waveguide, the suffix m and n mean the variation of the field along the angular and radial directions, respectively.
- For a parallel plate waveguide, the suffix m is 0 and the suffix n means the variation of the field between the plates.
- For a coaxial waveguide, only the TEM mode is available.
- For a lumped gap port, only the TEM mode or blank is available.

For a lumped gap port, a uniformly distributed electromagnetic field is assumed (that is, E<sub>y</sub> and H<sub>x</sub> components exist in the local coordinate system).

For a modal lumped gap port, a voltage path connecting the central conductor and the ground must be defined using the **LPRT** command.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas**

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Modify Port

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Port Status

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Modify Port

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Port Status

## HFSCAT, Lab

Specifies a high-frequency scattering analysis.

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> <> EH <> <> <> <>

### Lab

Label identifying scattering analysis options:

#### OFF

Do not perform a scattering analysis.

#### SCAT

Perform a scattering analysis and store the scattering field (default).

#### TOTAL

Perform a scattering analysis and store the total field.

## Notes

Specifies a high frequency scattering analysis and the type of electromagnetic field output. **HFSCAT**, **SCAT** provides a scattering field output,  $E^{sc}$ , which is required for the calculation of Radar Cross Section (RCS).

**HFSCAT**, **TOTAL** provides a sum of the incident and scattering fields,  $E^{total} = E^{inc} + E^{sc}$ .

Use the **PLWAVE** command to specify the incident field,  $E^{inc}$ .

## Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

**HFSYM**, *KCN*, *Xkey*, *Ykey*, *Zkey*

**Indicates the presence of symmetry planes for the computation of high-frequency electromagnetic fields in the near and far field domains (beyond the finite element region).**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

***KCN***

Coordinate system reference number. *KCN* may be 0 (Cartesian), or any previously defined local Cartesian coordinate system number (>10). Defaults to 0.

***Xkey***

Key for electromagnetic field boundary condition, as prescribed for the solution, corresponding to the  $x = \text{constant}$  plane:

**None**

No electric wall or magnetic wall boundary conditions (default).

**PEC**

Perfect electric conductor (electric wall boundary conditions).

**PMC**

Perfect magnetic conductor (magnetic wall boundary conditions).

***Ykey***

Key for electromagnetic field boundary condition, as prescribed for the solution, corresponding to the  $y = \text{constant}$  plane:

**None**

No electric wall or magnetic wall boundary conditions (default).

**PEC**

Perfect electric conductor (electric wall boundary conditions).

**PMC**

Perfect magnetic conductor (magnetic wall boundary conditions).

***Zkey***

Key for electromagnetic field boundary condition, as prescribed for the solution, corresponding to the  $z = \text{constant}$  plane:

**None**

No electric wall or magnetic wall boundary conditions (default).

**PEC**

Perfect electric conductor (electric wall boundary conditions).

**PMC**

Perfect magnetic conductor (magnetic wall boundary conditions).

**Notes**

**HFSYM** uses the image principle to indicate symmetry planes ( $x$ ,  $y$ , or  $z = \text{constant}$  plane) for high-frequency electromagnetic field computations outside the modeled domain. A perfect magnetic conductor (PMC) boundary condition must be indicated even though it occurs as a natural boundary condition.

## Menu Paths

Main Menu>General Postproc>List Results>Field Extension>Direct Gain  
 Main Menu>General Postproc>List Results>Field Extension>Efficiency  
 Main Menu>General Postproc>List Results>Field Extension>Far Field  
 Main Menu>General Postproc>List Results>Field Extension>Near Field  
 Main Menu>General Postproc>List Results>Field Extension>Pattern  
 Main Menu>General Postproc>List Results>Field Extension>Power Gain  
 Main Menu>General Postproc>List Results>Field Extension>Rad Power  
 Main Menu>General Postproc>List Results>Field Extension>RCS  
 Main Menu>General Postproc>List Results>Field Extension>RCS Normalized  
 Main Menu>General Postproc>Path Operations>Map onto Path>HF Near Field  
 Main Menu>General Postproc>Plot Results>Field Extension>Direct Gain  
 Main Menu>General Postproc>Plot Results>Field Extension>Far Field  
 Main Menu>General Postproc>Plot Results>Field Extension>Pattern  
 Main Menu>General Postproc>Plot Results>Field Extension>RCS  
 Main Menu>General Postproc>Plot Results>Field Extension>RCS Normalized

## HMAGSOLV, *FREQ*, *NRAMP*, *CNVA*, *CNVV*, *CNVC*, *CNVE*, *NEQIT*

Specifies 2-D or axisymmetric harmonic magnetic solution options and initiates the solution.

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> <> EM <> <> PP <> EME <>

### *FREQ*

Analysis frequency (Hz).

### *NRAMP*

Number of ramped substeps for the first load step of a nonlinear 2-D harmonic electromagnetic solution. Defaults to 3. If *NRAMP* = -1, ignore the ramped load step entirely.

### *CNVA*

Convergence tolerance on the program calculated reference value for the magnetic vector potential degree of freedom. Defaults to 0.001.

### *CNVV*

Convergence tolerance on the program calculated reference value for the time-integrated electric potential VOLT. Defaults to 0.001.

### *CNVC*

Convergence tolerance on the program calculated reference value for the current degree of freedom CURR. Defaults to 0.001.

### *CNVE*

Convergence tolerance on the program calculated reference value for the voltage drop degree of freedom EMF. Defaults to 0.001.

### *NEQIT*

Maximum number of equilibrium iterations per load step. Defaults to 50.

## Notes

**HMAGSOLV** invokes an ANSYS macro which specifies harmonic electromagnetic solution options and initiates the solution. The macro is applicable to any ANSYS 2-D or axisymmetric linear or nonlinear harmonic analysis. Results are only stored for the final converged solution. (In POST1, issue **\*SET,LIST** to identify the load step of solution results.) The macro internally determines if a nonlinear analysis is required based on magnetic material properties defined in the database.

The macro performs a two-load-step solution sequence. The first load step ramps the applied loads over a prescribed number of substeps (*NRAMP*), and the second load step calculates the converged solution. For linear problems, only a single load step solution is performed. The ramped load step can be bypassed by setting *NRAMP* to -1.

A 3-D harmonic electromagnetic analysis is available for linear solutions only and does not require this solution macro.

The following analysis options and nonlinear options are controlled by this macro: **KBC, NEQIT, NSUBST, CNVTOL, OUTRES**.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Solve>Electromagnet>Harmonic Analys>Opt&Solv**

---

**HPGL**, *Kywrđ*, *Opt1*, *Opt2*

**Specifies various HP options.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

If *Kywrđ* = **MODEL**, command format is **HPGL,MODEL,*Pmod***.

***Pmod***

Valid plotter model: 7475A (default), 7550A, 7580B, 7585B, 7586B, COLORPRO, DRAFTPRO, or DRAFTMASTER.

If *Kywrđ* = **PAPER**, command format is **HPGL,PAPER,*Size*,*Orien***.

***Size***

Valid paper size: A (default), B, C, D, E, A4, A3, A2, A1, A0, CARCH, DARCH, or EARCH.

***Orien***

Orientation: HORIZONTAL (default) or VERTICAL.

If *Kywrđ* = **COLOR**, command format is **HPGL,COLOR,*KEY***.

***KEY***

Pen choice:

**0**

Single pen



**1****Multiple pens**

If *Kywrđ* = **DIRECT**, command format is **HPGL,DIRECT,Port**.

Used to direct plotter commands to a port. Use **HPGL,FILE** to redirect output from a port back to a file. This option for *Kywrđ* is valid in the DISPLAY program only.

**Port**

Port name for direct connection.

**Notes**

This command is available in both the ANSYS and DISPLAY programs. It is valid for Hewlett Packard Graphics Language (HPGL) format files selected in the ANSYS program with **/SHOW,HPGL** (or HPGL2), or with **/SHOWDISP,HPGL** (or HPGL2) in the DISPLAY program.

An output file is generated for each plot. The ANSYS file is named *JobnameNN.HPGL*. In the DISPLAY program, this file is named *HPGLnn*. This file remains open for a subsequent **/NOERASE** plot, and will be incomplete until the program is closed (**/EXIT**), or until the next file is opened by the next **/ERASE** plot request.

**Menu Paths**

**Utility Menu>PlotCtrls>Redirect Plots>To HPGL File**  
**Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File**

**HPTCREATE, TYPE, ENTITY, NHP, LABEL, VAL1, VAL2, VAL3**

**Defines a hard point.**

[PREP7: Hard Points](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**TYPE**

Type of entity on which the hard point will be created.

**LINE**

Hard point will be created on a line.

**AREA**

Hard point will be created within an area (not on the boundaries).

**ENTITY**

Number of the line or area on which the hard point will be created.

**NHP**

Number assigned to the hard point. Defaults to the lowest available hard point number.

**LABEL**

If *LABEL* = COORD, *VAL1*, *VAL2*, and *VAL3* are the respective global X, Y, and Z coordinates. If *LABEL* = RATIO, *VAL1* is the parameter value (this is available only for lines). Valid parameter values are between 0 and 1. *VAL2* and *VAL3* are ignored.

**VAL1**

If *LABEL* = RATIO, ratio value for line. If *LABEL* = COORD, global X coordinate value.

**VAL2**

If *LABEL* = COORD, global Y coordinate value.

**VAL3**

If *LABEL* = COORD, global Z coordinate value.

## Notes

The ability to enter a parameter value provides a simple way of positioning hard points on lines. For example, to place a hard point halfway along a line, one can simply specify a *VAL1* value of 0.5.

For models imported through the DEFAULT IGES filter, you can place hard points on models only by specifying coordinates (you can't place a hard point using interactive picking).

If you issue any commands that update the geometry of an entity, such as Boolean or simplification commands, any hard points associated with that entity are deleted. Therefore, you should add any hard points after completing the solid model. If you delete an entity that has associated hard points, those hard points are either

- Deleted along with the entity (if the hard point is not associated with any other entities).
- Detached from the deleted entity (if the hard point is associated with additional entities).

When archiving your model (**CDWRITE**), hardpoint information cannot be written to the IGES file. The `Job-name.cdb` file can be written with the **CDWRITE,DB** option.

Hard points are only applicable for area and volume meshing, not for beams.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Hard Points>Hard PT on area>Hard PT by coordinates**

**Main Menu>Preprocessor>Modeling>Create>Hard Points>Hard PT on line>Hard PT by coordinates**

**Main Menu>Preprocessor>Modeling>Create>Hard Points>Hard PT on line>Hard PT by ratio**

**Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on area>Hard PT by coordinates**

**Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on area>Hard PT by picking**

**Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on line>Hard PT by coordinates**

**Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on line>Hard PT by picking**

**Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on line>Hard PT by ratio**

---

## HPTDELETE, *NP1*, *NP2*, *NINC*

Deletes selected hardpoints.

[PREP7: Hard Points](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NP1*, *NP2*, *NINC*

Delete the pattern of hard points beginning with *NP1* to *NP2* in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and the pattern is all selected hard points [**KSEL**]. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored.

## Notes

Deletes all attributes attached to the designated hard points as well as the hard points themselves. If any entity is attached to a designated hard point, the command detaches the hard point from that entity (the program will alert you that this will occur).

## Menu Paths

**Main Menu>Preprocessor>Modeling>Delete>Hard Points**

**HRCPLX**, *LOADSTEP*, *SUBSTEP*, *OMEGAT*, *1STLCASE*, *2NDLCASE*

**Computes and stores in the database the time-harmonic solution at a prescribed phase angle.**

[POST1: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *LOADSTEP*

Load step number of the data set to be read (defaults to 1).

### *SUBSTEP*

Substep number within *LOADSTEP*.

### *OMEGAT*

Angle in degrees ( $\Omega$  (angle) times T (time)).

- If  $\geq 360^\circ$ , the amplitude is supplied.
- All others supply results at that angle. For example, if the angle is set to  $0.0^\circ$ , the real part of the solution is supplied. If the angle is set to  $-90^\circ$  the imaginary part of the solution is supplied.

### *1STLCASE*

First load case number (defaults to 1).

### *2NDLCASE*

Second load case number (defaults to 2).

## Notes

**HRCPLX** invokes a macro which combines the real and imaginary parts of the solution. If the angle is specified, it produces the following:

$$\{R\} = \{R_R\} \cos \alpha - \{R_I\} \sin \alpha$$

Where:

$R_R$  and  $R_I$  are, respectively, the real and imaginary parts of the results quantity (e.g. the nodal displacements, the reaction forces, ...).

$\alpha$  is the angle (*OMEGAT*).

*1STLCASE* and *2NDLCASE* are internally generated load cases. You may want to specify these to avoid overwriting an existing load case number 1 or 2.

Not all results computed by this command are valid. See [Summable, Non-Summable and Constant Data](#) in the *Basic Analysis Guide* for more information.

For postprocessing amplitudes, the only appropriate coordinate system is the solution coordinate system (**RSYS**, SOLU). When displaying the displacement amplitudes, use a contour display (**PLNSOL** command). Because a deformed shape display (**PLDISP** command) could lead to a non-physical shape, the displacement scaling is off by default (**/DSCALE,,OFF**).

For postprocessing cylindrical geometry, it is suggested that you rotate the element coordinate systems into the appropriate cylindrical system (**EMODIF**, , **ESYS**) before running the solution and then view the results in this system (**RSYS**, SOLU) in POST1.

Since **HRCPLX** performs load case combinations, it alters most of the data in the database. In particular, it alters applied loads such as forces and imposed displacements. To restore the original loads in the database for a subsequent analysis, reissue the **SET** command in POST1 to retrieve the real and imaginary set data.

To animate the solution over one period, use the **ANHARM** command.

See the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for more information on harmonic analysis equations and their relationship to real and imaginary data sets.

## Menu Paths

**This command cannot be accessed from a menu.**

## HREXP, ANGLE

**Specifies the phase angle for the harmonic analysis expansion pass.**

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

### ANGLE

Phase angle (degrees) for expansion pass. If ALL (default), use both 0.0° (real) and 90.0° (imaginary) phase angles.

## Notes

Specifies the phase angle where the expansion pass will be done for a harmonic reduced or harmonic mode superposition expansion pass.

For a specific angle, the following real solution is stored in the results (\*.rst) file:

$$\{u\} = \{u_{\max}^i \cos(\phi^i - \varphi)\}$$

Where:

$i$  is the degree of freedom number.

$u_{\max}^i$  is the amplitude of the  $i$  th degree of freedom solution

$\Phi^i$  is the phase shift angle of the  $i$  th degree of freedom solution

$\phi$  is the supplied phase shift angle (*ANGLE*)

If *ANGLE* is ALL, both the real and imaginary parts of the solution are stored in the results file.

For more details about the solution equations, see [Harmonic Response Analyses](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

This command is ignored if the **HROPT** command has been issued with *Method* = VT or *Method* = VTRU.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Load Step**  
**Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq**  
**Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's**  
**Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Load Step**  
**Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq**  
**Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's**

**HROPT**, *Method*, *MAXMODE*, *MINMODE*, *MCOut*, *Damp*

**Specifies harmonic analysis options.**

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS  
[Product Restrictions](#)

### *Method*

Solution method for the harmonic analysis:

#### **AUTO**

Automatically select the most efficient method. Either the FULL method or the Variational Technology method is selected depending on the model. (default method).

#### **FULL**

Full method.

#### **REDUC**

Reduced method.

#### **MSUP**

Mode superposition method.

#### **VT**

Variational Technology method (based on FULL harmonic algorithm).

#### **VTPA**

Variational Technology perfect absorber method (based on FULL harmonic algorithm)..

#### **VTRU**

Variational Technology reuse method (based on FULL harmonic algorithm)..

**Note**

If the solution method is not specified by the user, ANSYS automatically selects either the Full method or the Variational Technology, depending on which method is most efficient for the model.

The Variational Technology perfect absorber method (applicable to elements [HF119](#) and [HF120](#) only) (*Method* = VTPA) is about 20% faster but slightly less accurate than the Variational Technology method (*Method* = VT).

The Variational Technology reuse method (*Method* = VTRU) simply does the stress pass of a previous run using the Variational Technology method (*Method* = VT) using the .RSX file. It can be used to refine the frequency range and sample density without needing to redo the entire analysis.

**MAXMODE**

Largest mode number to be used to calculate the response (for *Method* = MSUP only). Defaults to the highest mode calculated in the preceding modal analysis.

**MINMODE**

Smallest mode number to be used (for *Method* = MSUP only). Defaults to 1.

**MCout**

Modal coordinates output key (valid only for the mode superposition method MSUP):

**NO**

No output of modal coordinates (default).

**YES**

Output modal coordinates to the text file `jobname.MCF`.

---

**Note**

*MCout* provides a singular (one-time) output of the modal coordinates. It will not provide the information for [LSWRITE/LSSOLVE](#) operations.

**Damp**

Damping mode for frequency-dependent material properties (valid only for the Variational Technology Method VT).

**Hysteretic**

Not proportional to the frequency.

**Viscous**

Proportional to the frequency (default).

**Notes**

Specifies the method of solution for a harmonic analysis ([ANTYPE,HARMIC](#)). If used in SOLUTION, this command is valid only within the first load step. See the product restrictions indicated below.

To include [residual vectors](#) in your [mode-superposition harmonic analysis](#), specify [RESVEC,ON](#).

This command is also valid in PREP7.

If *Method* = FULL, this command is valid in Distributed ANSYS.

## Product Restrictions

The VT, VTPA, and VTRU options are only available with a HPC license.

Command Option Method	Available Products
AUTO	MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS
FULL	MP ME ST <> <> <> <> <> EM EH <> PP <> EME MFS
REDUC	MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS
MSUP	MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS
VT / VTPA / VTRU	MP ME ST <> <> <> <> <> EM EH <> PP <> EME MFS

## Menu Paths

**Main Menu>DesignXplorer>Solution>Solve**  
**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>Analysis Options**

## HROUT, *Reimky, Clust, Mcont*

Specifies the harmonic analysis output options.

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *Reimky*

Real/Imaginary print key:

#### **ON**

Print complex displacements as real and imaginary components (default).

#### **OFF**

Print complex displacements as amplitude and phase angle (degrees).

### *Clust*

Cluster option (for [HROPT](#),MSUP):

#### **OFF**

Uniform spacing of frequency solutions (default).

#### **ON**

Cluster frequency solutions about natural frequencies.

### *Mcont*

Mode contributions key (for [HROPT](#),MSUP):

#### **OFF**

No print of mode contributions at each frequency (default).

**ON**

Print mode contributions at each frequency.

**Notes**

Specifies the harmonic analysis (**ANTYPE**,HARMIC) output options. If used in SOLUTION, this command is valid only within the first load step. **OUTPR**,NSOL must be specified to print mode contributions at each frequency.

This command is ignored if the **HROPT** command has been issued with *Method* = VT, VTPA, or VTRU. Displacements are not available at expanded frequencies with these solution methods.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>Analysis Options**

---



---

# I Commands

**IC**, *NODE*, *Lab*, *VALUE*, *VALUE2*, *NEND*, *NINC*

**Specifies initial conditions at nodes.**

**SOLUTION:** Misc Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

## *NODE*

Node at which initial condition is to be specified. If ALL, apply to all selected nodes (**NSEL**). If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NODE*.

## *Lab*

Degree-of-freedom label for which the initial condition is to be specified. If ALL, use all appropriate labels.

**Structural labels:** UX, UY, or UZ (displacements or linear velocities); ROTX, ROTY, or ROTZ (rotations or angular velocities); HDSP (hydrostatic pressure).

**Thermal labels:** TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature).

**FLOTRAN fluid labels:** PRES (pressure); VX, VY, or VZ (velocities); ENKE or ENDS (turbulent kinetic energy or turbulent energy dissipation); SP01 through SP06 (multiple species mass fractions) or their user-defined names (**MSSPEC**). Electric label: VOLT (voltage).

**Magnetic labels:** MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials).

## *VALUE*

Initial value of the degree of freedom (first-order value). Defaults to the program default for that degree of freedom (0.0 for structural analysis, **TUNIF** for thermal analysis, etc.). Values are in the nodal coordinate system and in radians for rotational degrees of freedom.

## *VALUE2*

Second-order degree of freedom value, mainly used to specify initial structural velocity. Defaults to the program default for that degree of freedom (0.0 for structural analysis). Values are in the nodal coordinate system and in radians/time for rotational degrees of freedom.

## *NEND*, *NINC*

Specifies the same initial condition values at the range of nodes from *NODE* to *NEND* (defaults to *NODE*), in steps of *NINC* (defaults to 1).

## Notes

The **IC** command specifies initial conditions, which are the initial values of the specified degrees of freedom. It is valid only for a static analysis and full method transient analysis (**TIMINT,ON** and **TRNOPT,FULL**). For the transient, the initial value is specified at the beginning of the first load step, that is, at time = 0.0.

Initial conditions should always be step applied (**KBC,1**) and not ramped. (In a transient analysis when **SOLCONTROL** is ON, **KBC,1** is the default as long as **TIMINT** is also on.)

If constraints (**D**, **DSYM**, etc.) and initial conditions are applied at the same node, the constraint specification overrides. Exercise caution when specifying constraints. The degree-of-freedom values start from zero, or

the first value given in the table when table name is specified. To match the nonzero initial condition value with the initial value for degree-of-freedom constraint, use a table for the degree-of-freedom constraint.

For thermal analyses, any **TUNIF** specification should be applied before the **IC** command; otherwise, the **TUNIF** specification is ignored. If the **IC** command is input before any **TUNIF** specification, use the **ICDELE** command and then reissue any **TUNIF** specification and then follow with the **IC** command.

Be sure to define consistent initial conditions. For example, if you define an initial velocity at a single degree of freedom, the initial velocity at every other degree of freedom will be 0.0, potentially leading to conflicting initial conditions. In most cases, you should define initial conditions at every unconstrained degree of freedom in your model. If you define an initial condition for any degree of freedom at the pilot node of a rigid body (see [Modeling Rigid Bodies](#) in the *Contact Technology Guide* for the definition of rigid body), then the same initial condition must also be defined for the same degree of freedom on all other nodes of the rigid body.

After a solution has been performed, the specified initial conditions are overwritten by the actual solution and are no longer available. You must respecify them if you want to perform a subsequent analysis. You may want to keep a database file saved prior to the first solution for subsequent reuse.

If you use the **CDWRITE** command to archive your model, first-order values (initial displacements, temperatures, etc.) specified via the **IC** command are not written to the archive file; however, second-order (structural velocity) terms *are* written.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>Define**  
**Main Menu>Solution>Define Loads>Apply>Initial Condit'n>Define**

---

## ICDELE

Deletes initial conditions at nodes.

**SOLUTION: Misc Loads**

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

## Notes

Deletes all initial conditions previously specified with the **IC** command at all nodes.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Initial Condit'n**  
**Main Menu>Solution>Define Loads>Delete>Initial Condit'n**

---

**ICE**, *ELEM*, *Lab*, *VALUE*

**Specifies initial conditions on elements.**

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

***ELEM***

Element to which initial condition applies. If All, apply initial condition to all selected elements [ **ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

***Lab***

Valid initial conditions label. FLOTRAN fluid labels: VFRC (volume fraction).

***VALUE***

Initial value for the volume fraction.

## Notes

The **ICE** command specifies initial conditions on selected elements.

Once a solution has been performed, the specified initial conditions will be overwritten by the actual solution, and are no longer available. You must respecify them if you want to perform a re-analysis.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>On Elements**

**Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>On Elements**

**ICEDELE**, *ELEM*, *Lab*

**Deletes initial conditions on elements.**

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

***ELEM***

Element at which initial conditions are to be deleted. If ALL, initial conditions at all selected elements [ **ESEL** ] are deleted. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

***Lab***

Valid initial conditions label. FLOTRAN fluid labels: VFRC (volume fraction).

## Notes

Deletes all initial conditions previously specified with the **ICE** command at all elements.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Volume Fract>Init Loads>On Elements

Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Volume Fract>Init Loads>On Elements

## ICELIST, *ELEM*, *Lab*

**Lists initial conditions on elements.**

SOLUTION: Misc Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### ***ELEM***

List initial condition for elements for *ELEM* (defaults to ALL). If *ELEM* = All, initial conditions for all selected elements [ESEL] are listed. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in GUI). A component name may be substituted for *ELEM*.

### ***Lab***

Valid initial conditions label. FLOTRAN fluid labels: VFRC (volume fraction).

## Notes

Lists the initial on elements specified by the **ICE** command. Listing applies to all the selected elements [ESEL].

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>Elem Init Condit'n>On Picked Elemts

## ICLIST, *NODE1*, *NODE2*, *NINC*, *Lab*

**Lists the initial conditions.**

SOLUTION: Misc Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### ***NODE1*, *NODE2*, *NINC***

List initial conditions for nodes *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL (default), *NODE2* and *NINC* are ignored and initial conditions for all selected nodes [NSEL] are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

### ***Lab***

Velocity key:

#### **DISP**

Specification is for first order degree of freedom value (displacements, temperature, etc.) (default).

#### **VELO**

Specification is for second order degree of freedom value (velocities).

## Notes

Lists the initial conditions specified by the **IC** command. Listing applies to all the selected nodes [**NSEL**] and DOF labels. **ICLIST** is not the same as the **DLIST** command. All the initial conditions including the default conditions are listed for the selected nodes.

Initial conditions for a FLOTRAN analysis are primary (first order) degrees of freedom and are thus listed with the DISP key.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>List Picked**  
**Main Menu>Solution>Define Loads>Apply>Initial Condit'n>List Picked**  
**Utility Menu>List>Loads>Initial Conditions>On Picked Nodes**

---

## **/ICLWID**, *FACTOR*

**Scales the line width of circuit builder icons.**

[GRAPHICS:Scaling](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

### **FACTOR**

Multiplication factor applied to the default line width (defaults to 1). The minimum is 1 and the maximum is 6.

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon**

---

## **/ICSCALE**, *WN*, *FACTOR*

**Scales the icon size for elements supported in the circuit builder.**

[GRAPHICS:Scaling](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

### **WN**

Window number (or ALL) to which command applies (defaults to 1).

### **FACTOR**

Factor applied to the default icon size (defaults to 1).

## Notes

Scaling the icon size can provide better visualization of the circuit components when using the Circuit Builder (an interactive builder available in the ANSYS GUI).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon**

---

**ICVFRC**, *Geom*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

**Sets the initial volume fraction field for a geometry.**

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

*Geom*

Geometry:

**CIRC**

Circle.

**ELPT**

Ellipse.

*VAL1*, *VAL2*

Location of the center. *VAL1* and *VAL2* are the x and y coordinates, respectively.

*VAL3*, *VAL4*

If *Geom* = CIRC, *VAL3* is the radius of the circle and *VAL4* is not used. If *Geom* = ELPT, *VAL3* and *VAL4* are the x and y semiaxes of the ellipse, respectively.

## Notes

The **ICVFRC** command sets the initial volume fraction field for a geometry. The initial VFRC is set to one for elements completely within the geometry. The initial VFRC is set equal to the fraction of element area within the geometry for elements intersected by the boundary of the geometry.

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>By Geom>Circle**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>By Geom>Elliptic**

**Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>By Geom>Circle**

**Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>By Geom>Elliptic**

---

**\*IF**, *VAL1*, *Oper1*, *VAL2*, *Base1*, *VAL3*, *Oper2*, *VAL4*, *Base2*

**Conditionally causes commands to be read.**

APDL: Process Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *VAL1*

First numerical value (or parameter which evaluates to a numerical value) in the conditional comparison operation. *VAL1*, *VAL2*, *VAL3*, and *VAL4* can also be character strings (enclosed in quotes) or parameters for *Oper* = EQ and NE only.

### *Oper1*

Operation label. A tolerance of 1.0E-10 is used for comparisons between real numbers:

#### **EQ** --

Equal (for  $VAL1 = VAL2$ ).

#### **NE** --

Not equal (for  $VAL1 \neq VAL2$ ).

#### **LT** --

Less than (for  $VAL1 < VAL2$ ).

#### **GT** --

Greater than (for  $VAL1 > VAL2$ ).

#### **LE** --

Less than or equal (for  $VAL1 \leq VAL2$ ).

#### **GE** --

Greater than or equal (for  $VAL1 \geq VAL2$ ).

#### **ABLT** --

Absolute values of *VAL1* and *VAL2* before < operation.

#### **ABGT** --

Absolute values of *VAL1* and *VAL2* before > operation.

### *VAL2*

Second numerical value (or parameter which evaluates to a numerical value) in the conditional comparison operation.

### *Base1*

Action based on the logical expression (*Oper1*) being true. If false, continue reading at the next line. This is conditional, except for the IF-THEN-ELSE constructs described below; any of the following constructs (through *Base1* = THEN) cause all subsequent fields to be ignored:

#### **:label** --

A user-defined label (beginning with a colon (:), 8 characters maximum). The command reader will skip (and wrap to the beginning of the file, if necessary) to the first line that begins with the matching *:label*.

## Caution

This label option may not be mixed with do-loop or if-then-else constructs.

### STOP --

This action will cause an exit from the ANSYS program at this line, unless running in interactive mode. In interactive mode, the program will not stop.

### EXIT --

Exit the current do-loop [**\*EXIT**].

### CYCLE --

Skip to the end of the current do-loop [**\*CYCLE**].

### THEN --

Make this **\*IF** an if-then-else construct (see below).

The following optional values determine the connection between the two logical clauses *Oper1* and *Oper2*

### AND --

True if both clauses (*Oper1* and *Oper2*) are true.

### OR --

True if either clause is true.

### XOR --

True if either (but not both) clause is true.

### VAL3

Third numerical value (or parameter which evaluates to a numerical value).

### Oper2

Operation label. This will have the same labels as *Oper1*, except it uses *Val3* and *Val4*. A tolerance of 1.0E-10 is used for comparisons between real numbers.

### VAL4

Fourth numerical value (or parameter value which evaluates to a numerical value).

### Base2

Action based on the logical expression (*Oper1* and *Oper2*) being true. They will be the same values as *Base1*, except as noted.

## Command Default

Read commands sequentially.

## Notes

Conditionally causes commands to be read from a specific block or at a specific location. Twenty levels of nested **\*IF** blocks are allowed. Jumping to a *:label* line is not allowed with keyboard entry. Jumping into, out of, or within a do-loop or an if-then-else construct to a *:label* line is not allowed. Using **\*IF** interactively or from the command line prevents rereading the file to find a *label*. To do so, use batch mode or **/INPUT**.

The following is an example of an if-then-else construct:

```
*IF,VAL1,Oper,VAL2,THEN
```



```

----
*ELSEIF,VAL1,Oper,VAL2
----
*ELSEIF,VAL1,Oper,VAL2
----
*ELSE
----
*ENDIF

```

where "----" represents a block of any number of commands. Any number of **\*ELSEIF** clauses (or none) may be included (in the location shown). One **\*ELSE** clause (at most) may be included (in the location shown). The **\*IF** command is executed by evaluating its logical expression. If it is true, the block of commands following it is executed. The construct is considered to be complete and the command following the **\*ENDIF** is executed next. If the logical expression is false, the next **\*ELSEIF** command (if any) following the block is executed. The execution logic is the same as for **\*IF**. The effect is that the logical expressions in the **\*IF** and the **\*ELSEIF** commands are sequentially tested until one is found to be true. Then the block of commands immediately following the expression is executed, which completes the execution of the if-then-else construct. If all **\*IF** and **\*ELSEIF** expressions are false, the block following the **\*ELSE** command is executed, if there is one. Only one block of commands (at most) is executed within the if-then-else construct. If a batch input stream hits an end-of-file during a false **\*IF** condition, the ANSYS run will not terminate normally. You will need to terminate it externally (use either the UNIX "kill" function or the Windows task manager). The **\*IF**, **\*ELSEIF**, **\*ELSE**, and **\*ENDIF** commands for each if-then-else construct must all be read from the same file (or keyboard).

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**IGESIN**, *Fname*, *Ext*, --

**Transfers IGES data from a file into ANSYS.**

AUX15:IGES

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

### ***Ext***

Filename extension (8 character maximum).

The extension defaults to CAD if *Fname* is blank.

--

Unused field.

## Notes

Reads a file containing IGES data and transfers it into the ANSYS database. The file transferred is the IGES Version 5.1, ASCII format file. IGES (Initial Graphics Exchange Specification) is a neutral format developed by the U.S. Dept. of Commerce, National Institute of Standards and Technology. There is no output transfer file written since the transferred data is read directly into the ANSYS database.

You can import multiple files into a single database, but you must use the same import option (set with the **IOPTN** command) for each file.

The **IOPTN** command sets the parameters for reading the file. Files read via the SMOOTH method (the only available method) use the standard database.

## Menu Paths

Utility Menu>File>Import

---

**IGESOUT**, *Fname*, *Ext*, --, *ATT*

**Writes solid model data to a file in IGES Version 5.1 format.**

[PREP7: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to *Jobname*.

### ***Ext***

Filename extension (8 character maximum).

The extension defaults to IGES if *Fname* is blank.

--

Unused field.

### ***ATT***

Attribute key:

**0**

Do not write assigned numbers and attributes of the solid model entities to the IGES file (default).

**1**

Write assigned numbers and attributes of solid model entities (keypoints, lines, areas, volumes) to the IGES file. Attributes include MAT, TYPE, REAL, and ESYS specifications as well as associated solid model loads and meshing (keypoint element size, number of line divisions and spacing ratio) specifications.

## Notes

Causes the selected solid model data to be written to a coded file in the IGES Version 5.1 format. Previous data on this file, if any, are overwritten. Keypoints that are not attached to any line are written to the output file as IGES entity 116 (Point). Lines that are not attached to any area are written to the output file as either IGES Entity 100 (Circular Arc), 110 (Line), or 126 (Rational B-Spline Curve) depending upon whether the ANSYS entity was defined as an arc, straight line, or spline. Areas are written to the output file as IGES Entity 144 (Trimmed Parametric Surface). Volumes are written to the output file as IGES entity 186 (Manifold Solid B-Rep Object). Solid model entities to be written must have all corresponding lower level entities selected (use **ALLSEL, BELOW, ALL**) before issuing command. Concatenated lines and areas are not written to the IGES file; however, the entities that make up these concatenated entities are written.

### Caution

Section properties assigned to areas, lines and other solid model entities will not be maintained when the model is exported using **IGESOUT**.

If you issue the **IGESOUT** command after generating a beam mesh with orientation nodes, the orientation keypoints that were specified for the line (**LATT**) are no longer associated with the line and are not written out to the IGES file. The line does not recognize that orientation keypoints were ever assigned to it, and the orientation keypoints do not "know" that they are orientation keypoints. Thus the **IGESOUT** command does not support (for beam meshing) any line operation that relies on solid model associativity. For example, meshing the areas adjacent to the meshed line, plotting the line that contains the orientation nodes, or clearing the mesh from the line that contains orientation nodes may not work as expected. See [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide* for more information about beam meshing.

## Menu Paths

Utility Menu>File>Export

**/IMAGE**, *Label, Fname, Ext, --*

**Allows graphics data to be captured and saved.**

GRAPHICS: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Label*

Label specifying the operation to be performed:

#### **CAPTURE**

Capture the image from the graphics window to a new window.

#### **RESTORE**

Restore the image from a file to a new window.

#### **SAVE**

Save the contents of the graphic window to a file.

#### **DELETE**

Delete the window that contains the file.

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

***Ext***

Filename extension (8 character maximum).

If no extension is specified, `bmp` will be used on Windows systems, and `img` will be used on UNIX systems.

--

Unused field.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**IMAGIN**, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*

**Forms an imaginary variable from a complex variable.**

[POST26: Operations](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

***IA***

Reference number of the variable to be operated on.

--, --

Unused fields.

***Name***

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --

Unused fields.

***FACTA***

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

## Notes

This command forms a new variable from a complex variable by storing the imaginary part as the real part. The imaginary part can then be used in other operations. Used only with harmonic analyses (**ANTYPE**,HARMIC).

Complex variables are stored in two-column arrays with the real component stored in the first column and the imaginary component stored in the second column. This command extracts the value stored in the second column (i.e., imaginary component). However, with harmonic analyses, all variables are stored in two-column arrays as complex variables. If the variable is not complex, then the same value is stored in both

columns. This command will extract the variable in the second column of the array, even if this variable is not the imaginary component of a complex variable.

## Menu Paths

**Main Menu>TimeHist Postpro>Math Operations>Imaginary Part**

**IMESH**, *LAKY*, *NSLA*, *NTLA*, *KCN*, *DX*, *DY*, *DZ*, *TOL*

**Generates nodes and interface elements along lines or areas.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **LAKY**

Copies mesh according to the following:

#### **LINE or 1**

Copies line mesh (default).

#### **AREA or 2**

Copies area mesh.

### **NSLA**

Number that identifies the source line or area. This is the line or area whose mesh will provide the pattern for the interface elements. ANSYS copies the pattern of the line or area elements through the area or volume to create the mesh of area or volume interface elements.

### **NTLA**

Number that identifies the target line or area. This is the line or area that is opposite the source line or area specified by *NSLA*. Add *NTLA* to obtain the copied mesh from the source line or area.

### **KCN**

Number that identifies the particular ANSYS coordinate system.

### **DX, DY, DZ**

Incremental translation of node coordinates in the active coordinate system (*DR, D , DZ* for cylindrical, and *DR, D , D* for spherical or toroidal). The source line or area coordinates + *DX, DY, DZ* = the target line or area coordinates. If left blank, ANSYS automatically estimates the incremental translation.

### **TOL**

Tolerance for verifying topology and geometry. By default, ANSYS automatically calculates the tolerance based on associated geometries.

## Notes

Generates nodes and interface elements along lines or areas. The **IMESH** command requires that the target line or area exactly match the source line or area. Also, both target and source lines or areas must be in the same area or volume. The area or volume containing the source line or area must be meshed before executing **IMESH**, while the area or volume containing the target line or area must be meshed after executing **IMESH**.

For three dimensional problems where *LAKY* = AREA, ANSYS fills the interface layer according to the following table:

If source mesh consists of:	ANSYS fills the interface layer with:
Quadrilateral elements	Hexahedral elements
Triangle elements	Degenerated wedge elements
Combination quadrilateral and triangle elements	Combination hexahedral and degenerated wedge elements

## Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Interface Mesh>2D Interface  
Main Menu>Preprocessor>Meshing>Mesh>Interface Mesh>3D Interface

## IMMED, KEY

Allows immediate display of a model as it is generated.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### KEY

Immediate mode key:

- 0 Display only upon request, i.e., no immediate display (default with the GUI *off*).
- 1 Display immediately as model is generated (default with the GUI *on*).

## Command Default

As described above.

## Notes

Allows immediate display of a model (as it is generated) without a screen erase or a display request. Available only during an interactive session at a graphics display terminal. A valid graphics device name must first be specified on the **/SHOW** command.

The **IMMED** command allows you to control whether or not the model is displayed immediately as it is generated in an interactive session. By default in the GUI, your model will immediately be displayed in the Graphics Window as you create new entities (such as areas, keypoints, nodes, elements, local coordinate systems, boundary conditions, etc.). This is called *immediate mode* graphics. Also note that symbols (such as boundary conditions, local coordinate system triads, etc.) are shown immediately and will be present on subsequent displays unless you "turn off" the appropriate symbol using the GUI plot controls function or the appropriate graphics specification command.

An immediate image will also be automatically scaled to fit nicely within the Graphics Window -- a feature called *automatic scaling*. The new scaling is usually apparent on the automatic replot associated with immediate mode. To suppress automatic replot, issue **/UIS,REPLOTT,0**. (With automatic replot suppressed, the immediate image may not always be automatically scaled correctly.)

## Note

An immediate display in progress should *not* be aborted with the usual system "break" feature (or else the ANSYS session itself will be aborted). When you run the ANSYS program interactively *without* using the GUI, immediate mode is off by default.

This command is valid only in PREP7.

## Menu Paths

Utility Menu>PlotCtrls>Erase Options>Immediate Display

**IMPD**, *Vpath*, *Ipath*, *Vsymm*, *Isymm*

**Calculates the impedance of a conductor at a reference plane.**

POST1: Magnetics Calculations

MP ME <> <> <> <> <> <> EH <> PP <> EME MFS

### *Vpath*

Path name for a predefined path [**PATH** command] for calculating the EMF (voltage drop) from the conductor to a reference point. The path should start at the outer conductor wall and end at a reference voltage point.

### *Ipath*

Path name for a predefined path [**PATH** command] for calculating the MMF (current) in a conductor. The path should traverse a closed contour surrounding the conductor, and you should define the path in a counterclockwise direction.

### *Vsymm*

Symmetry factor applied to the calculated EMF (voltage drop). The EMF (voltage drop) from the conductor to the reference point is multiplied by *Vsymm*.

### *Isymm*

Symmetry factor applied to the calculated current. The calculated current is multiplied by *Isymm*.

## Notes

Used in a harmonic high-frequency electromagnetic analysis, **IMPD** calculates the impedance of a conductor at a reference plane from the EMF (voltage) and MMF (current) at the reference plane. The EMF (voltage drop) is calculated by a line integral from the input path name (specified by the *Vpath* argument) that extends from the conductor to a reference point. The MMF (current) is calculated by a closed path around the conductor from the input path name (specified with *Ipath*). In cases having modeled symmetry, you can multiply the voltage drop or current by symmetry factors (*Vsymm* and *Isymm* respectively).

This command macro returns the scalar parameters Zre and Zim, representing the real and imaginary components of the impedance.

See magnetic macros for further details.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>Impedance

**INISTATE**, *Action*, *Val1*, *Val2*, *Val3*, *Val4*, *Val5*, *Val6*, *Val7*, *Val8*, *Val9*

Defines initial state data and parameters.

PREP7: Data Tables  
SOLUTION: Analysis Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Action*

Specifies action for defining or manipulating **initial state** data:

#### SET

Use *Action* = SET to designate initial state coordinate system, data type and material type parameters. See "*Command Specification for Action = SET*" (p. 844).

#### DEFINE

Use *Action* = DEFINE to specify the actual state values, and the corresponding element, integration point, or layer information. See "*Command Specifications for Action = DEFINE*" (p. 845).

#### WRITE

Use *Action* = WRITE to write the initial state values to a file when the **SOLVE** command is issued. See "*Command Specifications for Action = WRITE*" (p. 846).

#### READ

Use *Action* = READ to read the initial state values from a file. See "*Command Specifications for Action = READ*" (p. 846).

#### LIST

Use *Action* = LIST to read out the initial state data. See "*Command Specifications for Action = LIST*" (p. 847).

#### DELETE

Use *Action* = DELE to delete initial state data from a selected set of elements. See "*Command Specifications for Action = DELETE*" (p. 847)

## Notes

The **INISTATE** command is available for **current-technology** elements only. The command is not for use with kinematic hardening material properties.

For detailed information about using the initial state capability, see "Initial State" in the *Basic Analysis Guide*.

## Command Specification for *Action* = SET

**INISTATE, SET, *Val1*, *Val2***

<i>Val1</i> =	<i>Val2</i> =
CSYS	Coordinate system. <i>Val2</i> will be an integer corresponding to the coordinate system:



<i>Val1</i> =	<i>Val2</i> =
	-2 = Element Coordinate System -1 = Material Coordinate System 0 = Global Cartesian Coordinate System 0 - 10 = Any ANSYS defined coordinate system. >= 11 = Any user defined coordinate system ID.
DTYP	Data type. <i>Val2</i> is the type of data that will be set on the subsequent <b>INISTATE,DEFINE</b> command:  STRE = Stress data (default) EPEL = Strain data EPPL = Plastic strain data PLEQ = Accumulated equivalent plastic strain PLWK = Plastic strain energy density
MAT	Material type. <i>Val2</i> is the material ID. Using <i>Val2</i> = -1 disables material-based initial state and enables integration-point-based initial state data.

## Notes

*Action* = SET specifies and modifies the environment into which you will define the initial state data (via a subsequent **INISTATE,DEFINE** command). Otherwise, subsequent **INISTATE,DEFINE** data is input as initial state data in the global Cartesian coordinate system.

## Command Specifications for *Action* = DEFINE

***INISTATE,DEFINE,ELID, Eint, Klayer, Parmint, Cxx, Cyy, Czz, Cxy, Cyz, Cxz***

***ELID*** --

Element ID number. If (blank), defaults to current element selection.

***Eint*** --

Gauss integration point (defaults to ALL). Not valid for material-based initial state data.

***Klayer*** --

Layer number (for layered solid/shell elements) or cell number for beam elements. Blank for nonlayered, non-beam, elements, and not valid (ignored) for material-based initial state data.

***ParmInt*** --

Section integration point within a layer, or cell-integration point for beams (typically four integration points). The default value is ALL. Not valid for material-based initial state data.

***Cxx, Cyy, Czz, Cxy, Cyz, Cxz*** --

Stress (S), strain (EPEL), or plastic strain (EPPL) values.

## Notes

You can issue the **INISTATE** command repeatedly to define multiple sets of initial state values. Initial state data can be specified according to elements, layers or integration points.

When the initial state parameters are being defined based on the material, (**INISTATE,SET,MAT,MATID**), the *ELID* value designates the element number and all subsequent values are ignored.

## Command Specifications for *Action = WRITE*

***INISTATE, WRITE, FLAG, , , , CSID, Dtype***

***FLAG*** --

Set this value to 1 to generate the initial state file, or 0 to disable initial state file generation.

***CSID*** --

Determines the coordinate system for the initial state:

**0 (default)**

Write in global Cartesian coordinate system for solid elements.

**-1 (or MAT)**

Write in material coordinate system

**-2 (or ELEM)**

Write in element coordinate system for link, beam, and layered elements.

***Dtype*** --

Sets the data type to be written in the IST file:

**S**

Output stresses.

**EPEL**

Output elastic strain.

**EPPL**

Output plastic strain.

**PLEQ**

Output equivalent plastic strain.

**PLWK**

Output plastic strain energy density.

### Notes

Default is 0 for solid elements and -2 for link, beam, and shell elements.

## Command Specifications for *Action = READ*

***INISTATE, READ, Fname, Ext, Path***

Read initial state data from a standalone initial state file of the specified name (*Fname*) and file name extension (*Ext*), located in the specified path (*Path*). The initial state file must be in a comma-delimited ASCII file format, consisting of individual rows for each stress/strain item, with each row consisting of columns separated by commas.

### Notes

Use the READ option to apply complex sets of initial state data to various elements, cells, layers, sections and integration points. See "Initial State" in the *Basic Analysis Guide* for additional information.

## Command Specifications for *Action = LIST*

### *INISTATE, LIST, ELID*

Lists initial state data for elements with ID = *ELID*. If *ELID* is blank, all initial state data for all selected elements are listed.

## Command Specifications for *Action = DELETE*

### *INISTATE, DELE, ELID*

Deletes initial state data for elements with ID = *ELID*. If *ELID* is blank, all initial state data for all selected elements are deleted.

## Menu Paths

This command cannot be accessed from a menu.

---

## **/INPUT**, *Fname, Ext, --, LINE, LOG*

Switches the input file for the commands that follow.

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to the current *Jobname* if *Ext* is specified.

### ***Ext***

Filename extension (8 character maximum).

--

Unused field.

### ***LINE***

A value indicating either a line number in the file or a user-defined label in the file from which to begin reading the input file.

#### **(blank), 0, or 1**

Begins reading from the top of the file (default).

#### ***LINE\_NUMBER***

Begins reading from the specified line number in the file.

#### ***:label***

Begins reading from the first line beginning with the matching user-defined label *:label* (beginning with a colon (:), 8 characters maximum).

**LOG**

Indicates whether secondary input from this command should be recorded in the command log (File.LOG) and the database log:

- 0**  
Record only the **/INPUT** command on the log (default).
- 1**  
Record commands in the specified secondary file as they are executed.

**Notes**

Switches the input file for the next commands. Commands are read from this file until an end-of-file or another file switching directive is read. An end-of-file occurs after the last record of the file or when a **/EOF** command is read. An automatic switch back one level (to the previous file) occurs when an end-of-file is encountered. Twenty levels of nested file switching are allowed. Note that files including **\*DO**, **\*USE**, **\*ULIB**, and the "Unknown Command" Macro have less nesting available because each of these operations also uses a level of file switching. For an interactive run, a **/INPUT,TERM** switches to the terminal for the next input. A **/EOF** read from the terminal then switches back to the previous file. A **/INPUT** (with a blank second field) switches back to the primary input file.

Setting **LOG = 1** on **/INPUT** causes all commands read from the specified file to be recorded in the command log (File.LOG) and the internal database command log [**LGWRITE**]. This option is recommended if the log file will be used later (e.g., as batch input or as an analysis file for design optimization). The **LOG = 1** option is only valid when the **/INPUT** occurs in the primary input file. Using **LOG = 1** on a nested **/INPUT** or on a **/INPUT** within a do-loop will have no effect (i.e., commands in the secondary input file are not written to the command log).

This command is valid in any processor.

**Menu Paths**

Utility Menu>File>Read Input from

---

**/INQUIRE**, *StrArray*, *FUNC*

Returns system information to a parameter.

[APDL: Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Argument Descriptions**

**StrArray**

Name of the "string array" parameter that will hold the returned values. String array parameters are similar to character arrays, but each array element can be as long as 128 characters. If the string parameter does not exist, it will be created.

**FUNC**

Specifies the type of system information returned:

**LOGIN --**

Returns the pathname of the login directory on UNIX systems or the pathname of the default directory (including drive letter) on Windows systems.

**DOCU --**

Returns the pathname of the ANSYS docu directory.

**APDL --**

Returns the pathname of the ANSYS APDL directory.

**PROG --**

Returns the pathname of the ANSYS executable directory.

**AUTH --**

Returns the pathname of the directory in which the license file resides.

**USER --**

Returns the name of the user currently logged-in.

**DIRECTORY --**

Returns the pathname of the current directory.

**JOBNAME --**

Returns the current *Jobname*. The value of *Jobname* can be up to 250 characters in length.

**RSTDIR --**

Returns rst directory (**FILE** command).

**RSTFILE --**

Returns rst file name (**FILE** command).

**RSTEXT --**

Returns rst file extension (**FILE** command).

**PSEARCH --**

Returns path used for "unknown command" macro (**/PSEARCH** command).

### ***Returning the Value of an Environment Variable to a Parameter***

If *FUNC*=ENV, the command format is **/INQUIRE**,*StrArray*,ENV,*ENVNAME*,*Substring*. In this instance, ENV specifies that the command should return the value of an environment variable. The following defines the remaining fields:

***ENVNAME***

Specifies the name of the environment variable.

***Substring***

If *Substring* = 1, the first substring (up to the first colon (:)) is returned. If *Substring* = 2, the second substring is returned, etc. For Windows platforms, the separating character is semicolon (;). If this argument is either blank or 0, the entire value of the environment variable is returned.

### ***Returning the Value of a Title to a Parameter***

If *FUNC* = TITLE, the command format is **/INQUIRE**,*StrArray*,TITLE,*Title\_num*. In this context, the value of *Title\_num* can be blank or 1 through 5. If the value is 1 or blank, the title is returned. If the value is 2 through 5, a corresponding subtitle is returned (2 denoting the first subtitle, and so on).

## Returning Information About a File to a Parameter

The **/INQUIRE** command can also return information about specified files within the file system. For these capabilities, the format is **/INQUIRE,Parameter,FUNC,Fname,Ext,--**. The following defines the fields:

### **Parameter**

Name of the parameter that will hold the returned values.

### **FUNC**

Specifies the type of file information returned:

#### **EXIST --**

Returns a 1 if the specified file exists, and 0 if it does not.

#### **DATE --**

Returns the date stamp of the specified file in the format *yyyymmdd.hhmmss*.

#### **SIZE --**

Returns the size of the specified file in MB.

#### **WRITE --**

Returns the status of the write attribute. A 0 denotes no write permission while a 1 denotes write permission.

#### **READ --**

Returns the status of the read attribute. A 0 denotes no read permission while a 1 denotes read permission.

#### **EXEC --**

Returns the status of the execute attribute (this has meaning only in UNIX). A 0 denotes no execute permission while a 1 denotes execute permission.

#### **LINES --**

Returns the number of lines in an ASCII file.

### **Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

### **Ext**

Filename extension (8 character maximum).

--

Unused field.

## Notes

The **/INQUIRE** command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**INRES**, *Item1*, *Item2*, *Item3*, *Item4*, *Item5*, *Item6*, *Item7*, *Item8*

**Identifies the data to be retrieved from the results file.**

POST1:Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Item1*, *Item2*, *Item3*, ..., *Item8*

Data to be read into the database from the results file. May consist of any of the following labels:

**ALL**

All solution items (default).

**BASIC**

NSOL, RSOL, NLOAD, STRS, FGRAD, and FFLUX items.

**NSOL**

Nodal DOF solution.

**RSOL**

Nodal reaction loads.

**ESOL**

Element solution items (includes all of the following):

**NLOAD**

Element nodal loads.

**STRS**

Element nodal stresses.

**EPEL**

Element elastic strains.

**EPTH**

Element thermal, initial, and swelling strains.

**EPPL**

Element plastic strains.

**EPCR**

Element creep strains.

**FGRAD**

Element nodal gradients.

**FFLUX**

Element nodal fluxes.

**MISC**

Element miscellaneous data (SMISC and NMISC).

## Notes

Identifies the type of data to be retrieved from the results file for placement into the database through commands such as **SET**, **SUBSET**, and **APPEND**. **INRES** is a companion command to the **OUTRES** command controlling data written to the database and the results file. Since the **INRES** command can only flag data that has already been written to the results file, care should be taken when using the **OUTRES** command to include all data you wish to retrieve for postprocessing later on.

## Menu Paths

Main Menu>General Postproc>Data & File Opts

---

## INRTIA

Specifies "Inertial loads" as the subsequent status topic.

SOLUTION: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Loads>Inertia Loads

Utility Menu>List>Status>Solution>Inertia Loads

---

**INT1**, *IR, IY, IX, --, Name, --, --, FACTA, FACTB, CONST*

Integrates a variable.

POST26: Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*IR*

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result. Table values represent integrated sum of *IY* to current table position of *IX*.

*IY, IX*

Integrate variable *IY* with respect to *IX*.

--

Unused field.

*Name*

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --

Unused fields.

*FACTA, FACTB*

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).



**CONST**

Initial value.

**Notes**

Integrates variables according to the operation:

$$IR = \int (FACTA \times IY) d(FACTB \times IX) + CONST$$

**Menu Paths**

**Main Menu>TimeHist Postpro>Math Operations>Integrate**

**INTSRF, Lab**

**Integrates nodal results on an exterior surface.**

POST1: Special Purpose

MP ME <> <> <> <> <> FL <> <> DY PP <> EME MFS

**Lab**

Label indicating degree of freedom to be integrated:

**PRES**

Pressure.

**TAUW**

Wall shear stress.

**FLOW**

Both pressure and wall shear stress.

**Notes**

Integrates nodal results on a surface. Use node selection (such as the EXT option of the **NSEL** command) to indicate the surface(s) of element faces to be used in the integration. A surface can be "created" by unselecting elements (such as unselecting non-fluid elements that are adjacent to fluid elements for the postprocessing of fluid flow result). Element faces attached to the selected nodes will be automatically determined. All nodes on a face must be selected for the face to be used. The integration results will cancel for nodes on common faces of adjacent selected elements.

Integration results are in the active coordinate system (see the **RSYS** command). The type of results coordinate system must match the type used in the analysis. However, you may translate and rotate forces and moments as needed. Use the **\*GET** command (**Utility Menu> Parameters> Get Scalar Data**) to retrieve the results.

**Menu Paths**

**Main Menu>General Postproc>Nodal Calcs>Surface Integral**

**IOPTN**, *Lab*, *VAL1***Controls options relating to importing a model.**

AUX15:IGES

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***Lab***

Label identifying the import option. The meaning of *VAL1* varies depending on *Lab*.

**STAT**

List overall status of import facilities, including current option values. *VAL1* is ignored.

**DEFA**

Set default values for all import options. *VAL1* is ignored.

**MERG**

Entity merge option. *VAL1* can be:

**YES**

Automatic merging is performed (default).

**NO**

No merging of entities.

**SOLID**

Solid option. *VAL1* can be:

**YES**

Solid is created automatically (default).

**NO**

No solid created.

**GTOLER**

Entity merging tolerance. If IGES = SMOOTH, the GTOLER,*VAL1* can be:

**DEFA**

Use system defaults (default).

**FILE**

Use tolerance from the imported file.

**n**

A user-specified tolerance value.

**IGES**

IGES import option. *VAL1* can be:

**STAT**

List status of IGES related options in the output window.

**SMOOTH (or RV52)**

Use more robust IGES revision 5.2 import function (default).

**SMALL**

Small areas option. *VAL1* can be:

**YES**

Small areas are deleted (default).

**NO**

Small areas are retained.

**VAL1**

Additional input value as described under each *Lab* option.

## Command Default

Merging is performed during the IGES transfer with no global solid model tolerance (GTOLER) used.

## Notes

Controls various options during a model file transfer. A global solid model tolerance (GTOLER) can be specified.

The SMALL,YES option (default) delete small areas and can cause geometrical inconsistencies that could cause the import process to abort. Retaining the small areas increases processor time and memory usage.

The data is stored in the standard ANSYS graphics database.

The IGES,SMOOTH (default) option is capable of reading in any rational B-spline curve entity (type 126), or rational B-spline surface entity (type 128) with a degree less than or equal to 20. Attempts to read in B-spline curve or surface entities of degree higher than 20 may result in error messages.

If you issue the **/CLEAR** command, the **IOPTN** settings return to their defaults.

For MERG,YES, merging of coincident geometry items is performed automatically when the **IGESIN** command is issued (that is, an internal **NUMMRG,KP** command is issued). The model is merged with the *consideration tolerance* (*TOLER* on **NUMMRG**) set equal to  $0.75 * \text{the shortest distance between the endpoints of any active line}$ . See the **NUMMRG** command for more information about the tolerances. In most cases, the default merging is appropriate. Use the **IOPTN** command when you want to:

- Disable merging operations.
- Override the default merging and specify a global solid model tolerance value (GTOLER).
- Disable the automatic creation of solids (SOLID).

The **IOPTN** command should be issued *before* the **IGESIN** command. You cannot change these options after your model has been imported or created. If you must change the options:

1. Clear the database (**/CLEAR**) or exit and restart the program.
2. Set the correct options.
3. Reimport or recreate the model.

This command is valid in any processor.

## Menu Paths

**Utility Menu>File>Import**

**IRLF**, *KEY*

**Specifies that inertia relief calculations are to be performed.**

SOLUTION: Inertia

MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS

**KEY**

Calculation key:

**0**

No inertia relief calculations.

**1**

Counterbalance loads with inertia relief forces.

**-1**

Precalculate masses for summary printout only (no inertia relief).

**Command Default**

No inertia relief calculations.

**Notes**

The **IRLF** command specifies that the program is to calculate accelerations to counterbalance the applied loads (inertia relief). Displacement constraints on the structure should be only those necessary to prevent rigid-body motions (3 are needed for a 2-D structure and 6 for a 3-D structure). The sum of the reaction forces at the constraint points will be zero. Accelerations are calculated from the element mass matrices and the applied forces. Data needed to calculate the mass (such as density) must be input. Both translational and rotational accelerations may be calculated.

This option applies only to the static (**ANTYPE**,STATIC) analysis. Nonlinearities, [elements that operate in the nodal coordinate system](#), and axisymmetric or generalized plane strain elements are not allowed. Models with both 2-D and 3-D element types are not recommended. Loads may be input as usual. Displacements and stresses are calculated as usual. Use **IRLIST** to print inertia relief calculation results. The mass and moment of inertia summary printed before the solution is accurate (because of the additional pre-calculations required for inertia relief). See [Inertia Relief](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for calculation details. See also the *Structural Analysis Guide* for procedural details.

If the inertia relief calculation is to be performed in the second or later load step, you must specify **EMAT-WRITE**,YES in the initial load step for the element matrices needed to perform the calculations to be available.

When a [superelement \(MATRIX50\)](#) is present in the model, any DOF constraints that you need to apply (**D**) on a degree of freedom (DOF) belonging to the superelement must be applied in the [use pass](#) of the **MATRIX50** element (*not* in the [generation pass](#)). The command has no effect in the [generation pass](#) of a substructure. In the [expansion pass](#), precalculation of masses for summary printout (**IRLF**,-1) occurs only on elements that are part of the substructure.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** A *KEY* value of 1 is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Inertia Relief

Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Incl Mass Summry

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Inertia Relief

Main Menu>Solution>Load Step Opts>Output Ctrls>Incl Mass Summry

## IRLIST

Prints inertia relief summary table.

POST1: Listing

MP ME ST PR PRN DS DSS FL <> <> <> PP <> EME MFS

## Notes

Prints the inertia relief summary data, including the mass summary table, the total load summary table, and the inertia relief summary table resulting from the inertia relief calculations. These calculations are performed in the solution phase [**SOLVE** or **PSOLVE**] as specified by the **IRLF** command.

Inertia relief output is stored in the database rather than in the results file (Jobname.RST). When you issue **IRLIST**, ANSYS pulls the information from the database, which contains the inertia relief output from the most recent solution [**SOLVE** or **PSOLVE**].

This command is valid in any processor.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

This command cannot be accessed from a menu.

**\*ITENGINE**, *Type, EngineName, PrecondName, Matrix, RhsVector, SolVector, MaxIter, Toler*

Performs a solution using an iterative solver.

APDL: Matrix Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### Type

Specifies the algorithm to be used:

#### PCG --

Preconditioned conjugate gradient (default).

### EngineName

Name used to identify this iterative solver engine. Must be specified.

### PrecondName

Linear solver engine name (**\*LSENGINE**) identifying the factored matrix to be used as the preconditioner.

**Matrix**

Name of the matrix to solve.

**RhsVector**

Matrix (load vector) name.

**SolVector**

Solution vector name. If non-zero, it will be taken as the initial vector for the iterative process.

**MaxIter**

Maximum number of iterations allowed. Default is 2 times the number of rows in the matrix.

**Toler**

Convergence tolerance. Default is 1.0E-8.

## Notes

This command solves  $Ax = b$  using a preconditioned conjugate gradient algorithm. It uses an existing factored system as the preconditioner. This solution method is useful if an existing matrix has been solved and minor changes have been made to the matrix.

## Menu Paths

**This command cannot be accessed from a menu.**

---

---

## J Commands

**JPEG**, *Kywrđ*, *OPT*

Provides JPEG file export for ANSYS displays.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Kywrđ*

Specifies various JPEG file export options.

### QUAL

If *Kywrđ* = QUAL, then *OPT* is an integer value defining the JPEG quality index on an arbitrary scale ranging from 1 to 100. The default value is 75.

### ORIENT

If *Kywrđ* = ORIENT, then *OPT* will determine the orientation of the entire plot. *OPT* can be either Horizontal (default) or Vertical.

### COLOR

If *Kywrđ* = COLOR, then *OPT* will determine the color depth of the saved file. *OPT* can be 0, 1, or 2, corresponding to Black and White, Grayscale, and Color (default), respectively.

### TMOD

If *Kywrđ* = TMOD, then *OPT* will determine the text method. *OPT* can be either 1 or 0, corresponding to bitmap text (default) or line stroke text, respectively.

### DEFAULT

If *Kywrđ* = DEFAULT, then all of the default values, for all of the *Kywrđ* parameters listed above, are active.

*OPT*

*OPT* can have the following names or values, depending on the value for *Kywrđ* (see above).

### 1 to 100

If *Kywrđ* = QUAL, a value between 1 and 100 will determine the quality index of the JPEG file.

### Horizontal, Vertical

If *Kywrđ* = ORIENT, the terms Horizontal or Vertical determine the orientation of the plot.

### 0,1,2

If *Kywrđ* = COLOR, the numbers 0, 1, and 2 correspond to Black and White, Grayscale and Color, respectively.

### 1,0

If *Kywrđ* = TMOD, the values 1 and 0 determine whether bitmap (1) or stroke text (0) fonts will be used

## Menu Paths

Utility Menu>PlotCtrls>HardCopy>ToFile

---

**JSOL**, *NVAR*, *ELEM*, *ITEM*, *COMP*, *Name*

**Specifies result items to be stored for the joint element.**

[POST26: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NVAR***

Arbitrary reference number or name assigned to this variable. Variable numbers can be 2 to *NV* (**NUMVAR**) while the name can be an eight-byte character string. Overwrites any existing results for this variable.

***ELEM***

Element number for which to store results.

***Item***

Label identifying the item. Valid item labels are shown in [Table 222: JSOL - Valid Item and Component Labels](#) (p. 860) below.

***Comp***

Component of the *Item* (if required). Valid component labels are shown in [Table 222: JSOL - Valid Item and Component Labels](#) (p. 860) below.

***Name***

Thirty-two character name identifying the item on printouts and displays. Defaults to a label formed by concatenating the first four characters of the *Item* and *Comp* labels.

## Notes

This command is valid for the [MPC184](#) joint elements. The values stored are for the free or unconstrained degrees of freedom of a joint element. Relative reaction forces and moments are available only if stiffness, damping, or friction is associated with the joint element.

**Table 222 JSOL - Valid Item and Component Labels**

Item	Comp	Description
U	X, Y, Z	x, y, or z relative displacement.
ROT	X, Y, Z	x, y, or z relative rotation.
VEL	X, Y, Z	x, y, or z relative linear velocity.
OMG	X, Y, Z	x, y, or z relative angular velocity.
ACC	X, Y, Z	x, y, or z relative linear acceleration.
DMG	X, Y, Z	x, y, or z relative angular acceleration.
RF	X, Y, Z	Relative reaction forces in the local x, y, or z direction.
RM	X, Y, Z	Relative reaction moments in the local x, y, or z direction.

## Menu Paths

**This command cannot be accessed from a menu.**



---

## K Commands

**K**, *NPT*, *X*, *Y*, *Z*

**Defines a keypoint.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NPT*

Reference number for keypoint. If zero, the lowest available number is assigned [**NUMSTR**].

*X*, *Y*, *Z*

Keypoint location in the active coordinate system (may be  $R, \theta, Z$  or  $R, \theta, \Phi$ ). If  $X = P$ , graphical picking is enabled and all other fields (including *NPT*) are ignored (valid only in the GUI).

### Notes

Defines a keypoint in the active coordinate system [**CSYS**] for line, area, and volume descriptions. A previously defined keypoint of the same number will be redefined. Keypoints may be redefined only if it is not yet attached to a line or is not yet meshed. Solid modeling in a toroidal system is not recommended.

### Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Keypoints>In Active CS**  
**Main Menu>Preprocessor>Modeling>Create>Keypoints>On Working Plane**

---

**KATT**, *MAT*, *REAL*, *TYPE*, *ESYS*

**Associates attributes with the selected, unmeshed keypoints.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*MAT*, *REAL*, *TYPE*, *ESYS*

Material number, real constant set number, type number, and coordinate system number to be associated with selected, unmeshed keypoints.

### Notes

Keypoints subsequently generated from the keypoints will also have these attributes. These element attributes will be used when the keypoints are meshed. If a keypoint does not have attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT**, **REAL**, **TYPE**, and **ESYS** command settings. Reissue the **KATT** command (before keypoints are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association.

If any of the arguments *MAT*, *REAL*, *TYPE*, or *ESYS* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, ANSYS can proceed with a keypoint meshing operation even when no logical element type has been assigned via **KATT,,,TYPE** or **TYPE**. For more information, see the discussion on setting element attributes in [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide*.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh Attributes>All Keypoints**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked KPs**

## KBC, KEY

Specifies stepped or ramped loading within a load step.

[SOLUTION: Load Step Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### KEY

Ramping key:

**0**

Loads are linearly interpolated (ramped) for each substep from the values of the previous load step to the values of this load step.

**1**

Loads are step changed (stepped) at the first substep of this load step to the values of this load step (i.e., the same values are used for all substeps). Useful for rate-dependent behavior (e.g., creep, viscoplasticity, etc.) or transient load steps only.

## Command Default

When **SOLCONTROL** is ON, ANSYS performs ramped loading if **ANTYPE,STATIC**, or if **ANTYPE,TRANS** and **TIMINT,OFF**. It performs stepped loading if **ANTYPE,TRANS** and **TIMINT,ON**. (In a transient analysis, **TIMINT,ON** is the default.)

When **SOLCONTROL** is OFF, ramped loading for all types of transient or nonlinear analysis.

## Notes

Specifies whether loads applied to intermediate substeps within the load step are to be stepped or ramped. Used only if *DTIME* on the **DELTIM** command is less than the time span or, conversely, if *NSBSTP* on the **NSUBST** command is greater than one. Flags (FSI, MXWF, MVDI, etc.) are always stepped.

For ramped loading (**KBC,0**), when a load is applied for the first time, it is interpolated from zero to the value of the current load step, and not from the initial condition or value of the degree of freedom from the previous load step. Spatially varying tabular loads or boundary conditions do not support direct ramping or stepping options and, instead, apply their full values according to the supplied tabular functions regardless of the **KBC** setting.

For a static or harmonic cyclic symmetry analysis, any load that varies by sector (**CYCOPT**,**LDSECT**) is tabular and is applied as a step change, regardless of the **KBC** setting; however, any non-tabular loads in the same analysis are ramped or stepped according to the **KBC** setting.

Irrespective of the **KBC** setting, loads are usually step-removed. See [Stepping or Ramping Loads](#) in the *Basic Analysis Guide* for more information.

It is sometimes difficult to obtain successful convergence with stepped loading in a nonlinear transient problem. If divergence is encountered, determine if stepped loading was used by default, then determine if it is appropriate for the analysis.

If you run an analysis using optimized nonlinear solution defaults (**SOLCONTROL**,**ON**) but do not issue the **KBC** command, the program chooses whether or not to use stepped or ramped loads. The program-chosen option will be recorded on the load step files as **KBC**, -1.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substps**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Transient**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substps**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substps**

---

**KBETW**, *KP1*, *KP2*, *KPNEW*, *Type*, *VALUE*

**Creates a keypoint between two existing keypoints.**

PREP7: Keypoints

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***KP1***

First keypoint. If *KP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### ***KP2***

Second keypoint.

### ***KPNEW***

Number assigned to the new keypoint. Defaults to the lowest available keypoint number.

### ***Type***

Type of input for *VALUE*.

### **RATIO**

Value is the ratio of the distances between keypoints as follows:  $(KP1-KPNEW)/(KP1-KP2)$ .

### **DIST**

Value is the absolute distance between *KP1* and *KPNEW* (valid only if current coordinate system is Cartesian).

**VALUE**

Location of new keypoint, as defined by *Type* (defaults to 0.5). If *VALUE* is a ratio (*Type* = *RATIO*) and is less than 0 or greater than 1, the keypoint is created on the extended line. Similarly, if *VALUE* is a distance (*Type* = *DIST*) and is less than 0 or greater than the distance between *KP1* and *KP2*, the keypoint is created on the extended line.

**Notes**

Placement of the new keypoint depends on the currently active coordinate system [**CSYS**]. If the coordinate system is Cartesian, the keypoint will lie on a straight line between *KP1* and *KP2*. If the system is not Cartesian (e.g., cylindrical, spherical, etc.), the keypoint will be located as if on a line (which may not be straight) created in the current coordinate system between *KP1* and *KP2*. Note that solid modeling in a toroidal coordinate system is not recommended.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Keypoints>KP between KPs**

**KCALC**, *KPLAN*, *MAT*, *KCSYM*, *KLOCPR*

**Calculates stress intensity factors in fracture mechanics analyses.**

POST1: Special Purpose

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**KPLAN**

Key to indicate stress state for calculation of stress intensity factors:

- 0**  
Plane strain and axisymmetric condition (default).
- 1**  
Plane stress condition.

**MAT**

Material number used in the extrapolation (defaults to 1).

**KCSYM**

Symmetry key:

- 0 or 1**  
Half-crack model with symmetry boundary conditions [**DSYM**] in the crack-tip coordinate system.  $K_{II} = K_{III} = 0$ . Three nodes are required on the path.
- 2**  
Like 1 except with antisymmetric boundary conditions ( $K_I = 0$ ).
- 3**  
Full-crack model (both faces). Five nodes are required on the path (one at the tip and two on each face).

**KLOCPR**

Local displacements print key:

- 0**  
Do not print local crack-tip displacements.
- 1**  
Print local displacements used in the extrapolation technique.

## Notes

Calculates the stress intensity factors ( $K_I$ ,  $K_{II}$ , and  $K_{III}$ ) associated with homogeneous isotropic linear elastic fracture mechanics. A displacement extrapolation method is used in the calculation (see [POST1 - Crack Analysis](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*). This method assumes that the displacement calculations are for the plane strain state. If the displacement calculations are performed using a plane stress formulation, the calculation of the stress intensity factors can be converted to the plane strain state by using  $KPLAN = 1$ . ANSYS Uses minor Poisson's ratio (**MP**,**NUXY**) for the stress intensity factor calculation, therefore the material's Poisson's ratio must be defined using **MP**,**NUXY** command. The **PATH** and **PPATH** commands must be used to define a path with the crack face nodes (*NODE1* at the crack tip, *NODE2* and *NODE3* on one face, *NODE4* and *NODE5* on the other (optional) face). A crack-tip coordinate system, having x parallel to the crack face (and perpendicular to the crack front) and y perpendicular to the crack face, must be the active RSYS and CSYS before **KCALC** is issued.

## Menu Paths

**Main Menu>General Postproc>Nodal Calcs>Stress Int Factr**

**KCENTER**, *Type*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *KPNEW*

**Creates a keypoint at the center of a circular arc defined by three locations.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Type of entity used to define the circular arc. The meaning of *VAL1* through *VAL4* will vary depending on *Type*. If *Type* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### **KP**

Arc is defined by keypoints.

### **LINE**

Arc is defined by locations on a line.

***VAL1*, *VAL2*, *VAL3*, *VAL4***

Values used to specify three locations on the arc (see table below).

***KPNEW***

Number assigned to new keypoint. Defaults to the lowest available keypoint number.

### Definitions:

If *Type* = KP, inputs *VAL1* through *VAL4* are defined as follows:

***VAL1***

First keypoint.

**VAL2**

Second keypoint.

**VAL3**

Third keypoint.

**VAL4**

Arc radius. If  $VAL4 = 0$  or blank (default), the arc is defined by the three keypoints specified as  $VAL1$ ,  $VAL2$ , and  $VAL3$  and arc radius is not used. If  $VAL4$  is nonzero,  $VAL1$ ,  $VAL2$ , and  $VAL4$  are used to calculate the center point, and  $VAL3$  is used to locate the center point as follows:

**VAL4 > 0**

Center point and  $VAL3$  are on the same side of the line between the first two keypoints.

**VAL4 < 0**

Center point and  $VAL3$  are on opposite sides of the line between the first two keypoints.

If  $Type = LINE$ , inputs  $VAL1$  through  $VAL4$  are defined as follows:

**VAL1**

Line number.

**VAL2**

Line ratio (0 to 1) indicating the first location (defaults to 0).

**VAL3**

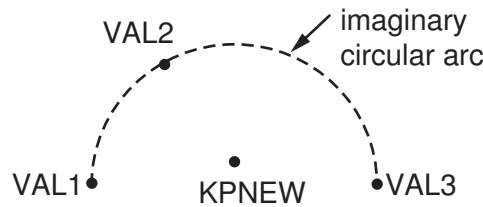
Line ratio (0 to 1) indicating the second location (defaults to 0.5).

**VAL4**

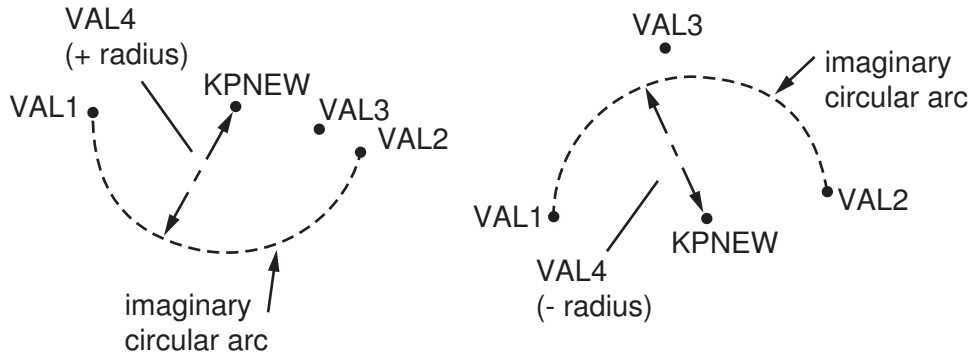
Line ratio (0 to 1) indicating the third location (defaults to 1).

## Notes

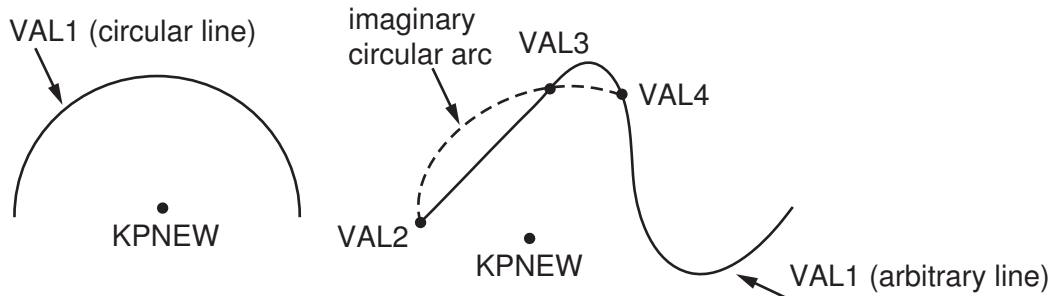
**KCENTER** should be used in the Cartesian coordinate system (**CSYS,0**) only. This command provides three methods to define a keypoint at the center of three locations. As shown below, the center point can be calculated based on a) three keypoints, b) three keypoints and a radius, or c) three locations on a line. Note that for method c, if a circular line is specified by  $VAL1$ ,  $VAL2$  through  $VAL4$  are not needed.



(a) Three keypoints



(b) Three keypoints and a radius



(c) Locations on line

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Keypoints>KP at center>3 keypoints**

**Main Menu>Preprocessor>Modeling>Create>Keypoints>KP at center>3 KPs and radius**

**Main Menu>Preprocessor>Modeling>Create>Keypoints>KP at center>Location on line**

## KCLEAR, NP1, NP2, NINC

**Deletes nodes and point elements associated with selected keypoints.**

PREP7: Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NP1, NP2, NINC

Delete mesh for keypoints *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and the mesh for all selected keypoints [KSEL] is deleted. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1*.

## Notes

Deletes *all* nodes and point elements associated with selected keypoints (regardless of whether the nodes or elements are selected). Nodes associated with non-point elements will not be deleted. Attributes assigned as a result of **KATT** are maintained. In the program's response to the command, if a keypoint is tallied as "cleared," it means either its node or element reference was deleted.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Clear>Keypoints**

---

**KDELE**, *NP1, NP2, NINC*

**Deletes unmeshed keypoints.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NP1, NP2, NINC*

Delete keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and all selected keypoints [**KSEL**] are deleted. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

## Notes

Deletes selected keypoints. A keypoint attached to a line cannot be deleted unless the line is first deleted.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Delete>Keypoints**

---

**KDIST**, *KP1, KP2*

**Calculates and lists the distance between two keypoints.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*KP1*

First keypoint in distance calculation. If *KP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

*KP2*

Second keypoint in distance calculation.

## Notes

**KDIST** lists the distance between keypoints *KP1* and *KP2*, as well as the current coordinate system offsets from *KP1* to *KP2*, where the X, Y, and Z locations of *KP1* are subtracted from the X, Y, and Z locations of *KP2* (respectively) to determine the offsets. **KDIST** is valid in any coordinate system except toroidal [**CSYS**,3].



**KDIST** returns a variable, called "*\_RETURN*," which contains the distance value. You can use this value for various purposes; for example, to set the default number of line divisions to be generated along region boundary lines [**ESIZE**, *\_RETURN*]. In interactive mode, you can access this command by using the Model Query Picker (**Utility Menu > List > Picked Entities**), where you can also access automatic annotation functions, and display the value on your model.

This command is valid in any processor.

## Menu Paths

**Main Menu > Preprocessor > Modeling > Check Geom > KP distances**

## KEEP, *Key*

Stores POST26 definitions and data during active session.

POST26: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Key

State or value

#### On or 1

Allows you to exit and reenter **/POST26** without losing your current time history variable information. Keeps a cache of the **/POST26** variable information including the active file name (**FILE**), variable definitions (**NSOL**, **ESOL**, **GAPF**, **RFORCE**, **SOLU**, and **EDREAD**) and stored variable data in memory for the current ANSYS session.

#### Off or 0

**/POST26** variable information is deleted when you exit **/POST26**.

## Command Default

ON - Hold time history information in memory. You can, for example, move back and forth between **/POST1** and **/POST26** without redefining and storing the time history variables each time you enter **/POST26**.

## Notes

Your variable information is saved in memory only for the current active ANSYS session. It is deleted when you exit ANSYS. This information is also deleted when you issue **/CLEAR**, **RESUME**, **SOLVE**, or **RESET**.

When you reenter **/POST26** all time history variable data is available for use. When you issue **STORE,NEW**, variable definitions created by math operations such as **ADD** or **PROD** will not be restored. However, variables defined with **NSOL**, **ESOL**, **GAPF**, **RFORCE**, **SOLU**, and **EDREAD** will be restored. Only the last active results file name is kept in memory (**FILE**).

Commands such as **LAYERP26**, **SHELL**, and **FORCE** that specify the location or a component of data to be stored will retain the setting at the time of exiting **/POST26**.

## Menu Paths

**Main Menu > TimeHist Postpro > Settings > Data**

**KESIZE**, *NPT*, *SIZE*, *FACT1*, *FACT2*

Specifies the edge lengths of the elements nearest a keypoint.

PREP7: Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NPT***

Number of the keypoint whose lines will be adjusted. If ALL, use all selected keypoints [**KSEL**]. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

***SIZE***

Size of elements along lines nearest keypoint *NPT* (overrides any other specified size). If *SIZE* is zero (or blank), use *FACT1* or *FACT2*.

***FACT1***

Scale factor applied to a previously defined *SIZE*. Not used if *SIZE* is input.

***FACT2***

Scale factor applied to the minimum element division at keypoint *NPT* for any attached line. This feature is useful with adaptive mesh refinement. Not used if *SIZE* or *FACT1* is input.

## Notes

Affects only the line divisions adjacent to the keypoint on lines not previously assigned divisions by other line commands [**LESIZE**, etc.]. The remaining line divisions are determined from the division nearest the keypoint at the other end of the line (specified by another **KESIZE** command or the **ESIZE** command). Divisions are transferred to the lines during the mesh operation. If smart element sizing is being used [**SMRTSIZE**], **KESIZE** specifications may be overridden (i.e., a smaller element size may be used) to accommodate curvature and small features.

This command is valid in any processor. The command is also valid for rezoning.

## Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Keypoints>All KPs

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Keypoints>Clr Size

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Keypoints>Picked KPs

**KEYOPT**, *ITYPE*, *KNUM*, *VALUE*

Sets element key options.

PREP7: Element Type

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***ITYPE***

Element type number as defined on the **ET** command.

***KNUM***

Number of the KEYOPT to be defined (KEYOPT(*KNUM*)).

***VALUE***

Value of this KEYOPT.

## Notes

Alternative to inputting KEYOPT values on **ET** command. Must be used if KEYOPT(7) or greater values are to be input. *ITYPE* must first be defined with the **ET** command.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>FLOTRAN Coor Sys**

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**

**Main Menu>Solution>FLOTRAN Set Up>Flow Environment>FLOTRAN Coor Sys**

**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

## KEYPTS

Specifies "Keypoints" as the subsequent status topic.

**PREP7: Status**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**This command cannot be accessed from a menu.**

**KEYW**, *Keyword*, *KEY*

**Sets a keyword used by the GUI for context filtering (GUI).**

**SESSION: Run Controls**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Keyword*

A keyword which, when set to either true or false, changes the behavior of the GUI.

### *KEY*

Keyword switch:

**0**

Sets the keyword to "false."

**1**

Sets the keyword to "true."

## Notes

Defines a keyword used by the GUI for context filtering. This is a command generated by the GUI and may appear in the log file (`Jobname.LOG`) if the GUI is used. This command is usually *not* typed in directly in an ANSYS session.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preferences**

---

**KFILL**, *NP1*, *NP2*, *NFILL*, *NSTRT*, *NINC*, *SPACE*

**Generates keypoints between two keypoints.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NP1*, *NP2***

Beginning and ending keypoints for fill-in. *NP1* defaults to next to last keypoint specified, *NP2* defaults to last keypoint specified. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

***NFILL***

Fill *NFILL* keypoints between *NP1* and *NP2* (defaults to  $\lfloor \frac{NP2-NP1}{NFILL} \rfloor$ ). *NFILL* must be positive.

***NSTRT***

Keypoint number assigned to first filled-in keypoint (defaults to  $NP1 + NINC$ ).

***NINC***

Add this increment to each of the remaining filled-in keypoint numbers (may be positive or negative). Defaults to  $(NP2-NP1)/(NFILL + 1)$ , i.e., linear interpolation.

***SPACE***

Spacing ratio. Ratio of last division size to first division size. If  $> 1.0$ , divisions increase. If  $< 1.0$ , divisions decrease. Ratio defaults to 1.0 (uniform spacing).

## Notes

Generates keypoints (in the active coordinate system) between two existing keypoints. The two keypoints may have been defined in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Any number of keypoints may be filled in and any keypoint numbering sequence may be assigned.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Keypoints>Fill between KPs**

---

**KGEN**, *ITIME*, *NP1*, *NP2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

**Generates additional keypoints from a pattern of keypoints.**

PREP7: Keypoints

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***ITIME***

Do this generation operation a total of *ITIME* times, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be more than 1 for generation to occur.

### ***NP1*, *NP2*, *NINC***

Generate keypoints from the pattern of keypoints beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and the pattern is all selected keypoints [KSEL]. If *NP1* is negative, *NP2* and *NINC* are ignored and the last  $|NP1|$  keypoints (in sequence from the highest keypoint number) are used as the pattern to be repeated. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

### ***DX*, *DY*, *DZ***

Keypoint location increments in the active coordinate system (DR, D $\theta$ , DZ for cylindrical, DR, D $\theta$ , D $\Phi$  for spherical).

### ***KINC***

Keypoint increment between generated sets. If zero, the lowest available keypoint numbers are assigned [NUMSTR].

### ***NOELEM***

Specifies if elements and nodes are also to be generated:

**0**

Generate nodes and point elements associated with the original keypoints, if they exist.

**1**

Do not generate nodes and elements.

### ***IMOVE***

Specifies whether keypoints will be moved or newly defined:

**0**

Generate additional keypoints as requested with the *ITIME* argument.

**1**

Move original keypoints to new position retaining the same keypoint numbers (*ITIME*, *KINC*, and *NOELEM* are ignored). Valid only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

## **Notes**

Generates additional keypoints (and corresponding mesh) from a given keypoint pattern. The MAT, TYPE, REAL, and ESYS attributes are based upon the keypoints in the pattern and not upon the current settings. Generation is done in the active coordinate system. Keypoints in the pattern may have been defined in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

## **Menu Paths**

**Main Menu>Preprocessor>Modeling>Copy>Keypoints**

**KL**, *NL1*, *RATIO*, *NK1***Generates a keypoint at a specified location on an existing line.**[PREP7: Keypoints](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**NL1**

Number of the line. If negative, the direction of line (as interpreted for *RATIO*) is reversed. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**RATIO**

Ratio of line length to locate keypoint. Must be between 0.0 and 1.0. Defaults to 0.5 (divide the line in half).

**NK1**

Number to be assigned to keypoint generated at division location (defaults to lowest available keypoint number [**NUMSTR**]).

**Menu Paths****Main Menu>Preprocessor>Modeling>Create>Keypoints>On Line****Main Menu>Preprocessor>Modeling>Create>Keypoints>On Line w/Ratio****KLIST**, *NP1*, *NP2*, *NINC*, *Lab***Lists the defined keypoints or hard points.**[PREP7: Keypoints](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**NP1, NP2, NINC**

List keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL (default), *NP2* and *NINC* are ignored and all selected keypoints [**KSEL**] are listed. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

**Lab**

Coordinate listing key:

**(blank)**

List all keypoint information.

**COORD**

Suppress all but the keypoint coordinates (shown to a higher degree of accuracy than when displayed with all information).

**HPT**

List only hard point information.

**Notes**

Lists keypoints in the active display coordinate system [**DSYS**]. An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **KATT** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates

that the attribute was assigned using the attribute pointer [**TYPE**, **MAT**, **REAL**, or **ESYS**] that was active during meshing (and will be reset to zero if the mesh is cleared).

This command is valid in any processor.

## Menu Paths

**Utility Menu>List>Keypoints>Coordinates +Attributes**  
**Utility Menu>List>Keypoints>Coordinates only**  
**Utility Menu>List>Keypoints>Hard Points**

---

## KMESH, NP1, NP2, NINC

**Generates nodes and point elements at keypoints.**

PREP7: Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NP1, NP2, NINC

Mesh keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and all selected keypoints [**KSEL**] are meshed. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1*.

## Notes

Missing nodes required for the generated elements are created and assigned the lowest available numbers.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh>Keypoints**

---

## KMODIF, NPT, X, Y, Z

**Modifies an existing keypoint.**

PREP7: Keypoints

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NPT

Modify coordinates of this keypoint. If *NPT* = ALL, modify coordinates of all selected keypoints [**KSEL**]. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

### X, Y, Z

Replace the previous coordinate values assigned to this keypoint with these corresponding coordinate values. Values are interpreted according to the active coordinate system (R,  $\theta$ , Z for cylindrical, R,  $\theta$ ,  $\Phi$  for spherical). If *X* = P, graphical picking is used to locate keypoint and *Y* and *Z* are ignored. A blank retains the previous value. You cannot specify *Y* = P.

## Notes

Lines, areas, and volumes attached to the modified keypoint (if any) must all be selected and will be redefined using the active coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

---

### Caution

Redefined entities may be removed from any defined components and assemblies. Nodes and elements will be automatically cleared from any redefined keypoints, lines, areas, or volumes.

The **KMODIF** command moves keypoints for geometry modification without validating underlying entities. To merge keypoints and update higher order entities, issue the **NUMMRG** command instead.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Keypoints>Set of KPs**

**Main Menu>Preprocessor>Modeling>Move / Modify>Keypoints>Single KP**

---

**KMOVE**, *NPT*, *KC1*, *X1*, *Y1*, *Z1*, *KC2*, *X2*, *Y2*, *Z2*

**Calculates and moves a keypoint to an intersection.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **NPT**

Move this keypoint. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

### **KC1**

First coordinate system number. Defaults to 0 (global Cartesian).

### **X1, Y1, Z1**

Input one or two values defining the location of the keypoint in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Fields are R1,  $\theta_1$ , Z1 for cylindrical, or R1,  $\theta_1$ ,  $\phi_1$  for spherical.

### **KC2**

Second coordinate system number.

### **X2, Y2, Z2**

Input two or one value(s) defining the location of the keypoint in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Arguments are R2,  $\theta_2$ , Z2 for cylindrical, or R2,  $\theta_2$ ,  $\phi_2$  for spherical.

## Notes

Calculates and moves a keypoint to an intersection location. The keypoint must have been previously defined (at an approximate location) or left undefined (in which case it is internally defined at the **SOURCE** location). The actual location is calculated from the intersection of three surfaces (implied from three coordinate constants in two different coordinate systems). Note that solid modeling in a toroidal coordinate system is



not recommended. See the **MOVE** command for surface and intersection details. The three (of six) constants easiest to define should be used. The program will calculate the remaining three coordinate constants. All arguments, except *KC1*, must be input. Use the repeat command [**\*REPEAT**] after the **KMOVE** command to move a series of keypoints, if desired.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Keypoints>To Intersect**

---

### **KNODE**, *NPT*, *NODE*

**Defines a keypoint at an existing node location.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *NPT*

Arbitrary reference number for keypoint. If zero, the lowest available number is assigned [**NUMSTR**].

#### *NODE*

Node number defining global X, Y, Z keypoint location. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Keypoints>On Node**

---

### **KPLOT**, *NP1*, *NP2*, *NINC*, *Lab*

**Displays the selected keypoints.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *NP1*, *NP2*, *NINC*

Display keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL (default), *NP2* and *NINC* are ignored and all selected keypoints [**KSEL**] are displayed.

#### *Lab*

Determines what keypoints are plotted (one of the following):

##### **(blank)**

Plots all keypoints.

##### **HPT**

Plots only those keypoints that are hard points.

## Notes

This command is valid in any processor.

## Menu Paths

Utility Menu>Plot>Keypoints>Hardpoints

Utility Menu>Plot>Keypoints>Keypoints

Utility Menu>Plot>Specified Entities>Keypoints

**KPSCALE**, *NP1*, *NP2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE*

**Generates a scaled set of (meshed) keypoints from a pattern of keypoints.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NP1*, *NP2*, *NINC***

Set of keypoints (*NP1* to *NP2* in steps of *NINC*) that defines the pattern to be scaled. *NP2* defaults to *NP1*, *NINC* defaults to 1. If *NP1* = ALL, *NP2* and *NINC* are ignored and the pattern is defined by all selected keypoints. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

### ***RX*, *RY*, *RZ***

Scale factors to be applied to the X, Y, Z keypoint coordinates in the active coordinate system (RR, R $\theta$ , RZ for cylindrical; RR, R $\theta$ , R $\Phi$  for spherical). The R $\theta$  and R $\Phi$  scale factors are interpreted as angular *offsets*. For example, if CSYS = 1, an *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints.) Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

### ***KINC***

Increment to be applied to the keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned [**NUMSTR**].

### ***NOELEM***

Specifies whether nodes and elements are also to be generated:

**0**

Nodes and point elements associated with the original keypoints will be generated (scaled) if they exist.

**1**

Nodes and point elements will *not* be generated.

### ***IMOVE***

Specifies whether keypoints will be moved or newly defined:

**0**

Additional keypoints will be generated.

**1**

Original keypoints will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

## Notes

Generates a scaled set of keypoints (and corresponding mesh) from a pattern of keypoints. The MAT, TYPE, REAL, and ESYS attributes are based on the keypoints in the pattern and not the current settings. Scaling is

done in the active coordinate system. Keypoints in the pattern could have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Scale>Keypoints**

**KREFINE**, *NP1*, *NP2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

**Refines the mesh around specified keypoints.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NP1*, *NP2*, *NINC***

Keypoints (*NP1* to *NP2* in increments of *NINC*) around which the mesh is to be refined. *NP2* defaults to *NP1*, and *NINC* defaults to 1. If *NP1* = ALL, *NP2* and *NINC* are ignored and all selected keypoints are used for refinement. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

### ***LEVEL***

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

### ***DEPTH***

Depth of mesh refinement in terms of the number of elements outward from the indicated keypoints (defaults to 1).

### ***POST***

Type of postprocessing to be done after element splitting, in order to improve element quality:

#### **OFF**

No postprocessing will be done.

#### **SMOOTH**

Smoothing will be done. Node locations may change.

#### **CLEAN**

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

### ***RETAIN***

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

#### **ON**

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

#### **OFF**

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

## Notes

**KREFINE** performs local mesh refinement around the specified keypoints. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

**KREFINE** refines all area elements and tetrahedral volume elements that are adjacent to the specified keypoints. Any volume elements that are adjacent to the specified keypoints, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. See [Revising Your Model](#) in the *Modeling and Meshing Guide* for additional restrictions on mesh refinement.

This command is also valid for [rezoning](#).

## Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Keypoints

---

**KSCALE**, *KINC*, *NP1*, *NP2*, *NINC*, *RX*, *RY*, *RZ*

**Generates a scaled pattern of keypoints from a given keypoint pattern.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *KINC*

Do this scaling operation one time, incrementing all keypoints in the given pattern by *KINC*. If *KINC* = 0, keypoints will be redefined at the scaled locations.

### *NP1*, *NP2*, *NINC*

Scale keypoints from pattern beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and pattern is all selected keypoints [**KSEL**]. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

### *RX*, *RY*, *RZ*

Scale factor ratios. Scaling is relative to the origin of the active coordinate system (RR, R $\theta$ , RZ for cylindrical, RR, R $\theta$ , R $\Phi$  for spherical). If > 1.0, pattern is enlarged. If < 1.0, pattern is reduced. Ratios each default to 1.0.

## Notes

Generates a scaled pattern of keypoints from a given keypoint pattern. Scaling is done in the active coordinate system (see analogous node scaling [**NSCALE**]). Solid modeling in a toroidal coordinate system is not recommended.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**KSCON**, *NPT*, *DELR*, *KCTIP*, *NTHET*, *RRAT*

**Specifies a keypoint about which an area mesh will be skewed.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NPT***

Keypoint number at concentration. If *NPT* = ALL, use all selected keypoints. If remaining fields are blank, remove concentration from this keypoint (if unmeshed). If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

***DELR***

Radius of first row of elements about keypoint.

***KCTIP***

Crack tip singularity key:

**0**

Do not skew midside nodes, if any, within the element.

**1**

Skew midside nodes of the first row of elements to the 1/4 point for crack tip singularity.

***NTHET***

Number of elements in circumferential direction (defaults to approximately one per 45° (or one per 30°, if *KCTIP* = 1)).

***RRAT***

Ratio of 2nd row element size to *DELR* (defaults to 0.75, or 0.5 if *KCTIP* = 1).

## Notes

Defines a concentration keypoint about which an area mesh will be skewed. Useful for modeling stress concentrations and crack tips. During meshing, elements are initially generated circumferentially about, and radially away, from the keypoint. Lines attached to the keypoint are given appropriate divisions and spacing ratios. Only one concentration keypoint per unmeshed area is allowed. Use **KSCON,STAT** to list current status of concentration keypoints. The **KSCON** command does not support 3-D modeling.

This command is also valid for [rezoning](#).

## Menu Paths

**Main Menu>Preprocessor>Meshing>Size Cntrls>Concentrat KPs>Create**

**Main Menu>Preprocessor>Meshing>Size Cntrls>Concentrat KPs>List**

**KSEL**, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS***Selects a subset of keypoints or hard points.**

DATABASE: Selecting

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**Type**

Label identifying the type of select:

**S**

Select a new set (default).

**R**

Reselect a set from the current set.

**A**

Additionally select a set and extend the current set.

**U**

Unselect a set from the current set.

**ALL**

Restore the full set.

**NONE**

Unselect the full set.

**INVE**

Invert the current set (selected becomes unselected and vice versa).

**STAT**

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:**Item**Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to KP.**Comp**

Component of the item (if required). Valid component labels are shown in the table below.

**VMIN**Minimum value of item range. Ranges are keypoint numbers, coordinate values, attribute numbers, etc., as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, or ESYS and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **KLIST** command for a discussion of signed attributes.**VMAX**Maximum value of item range. *VMAX* defaults to *VMIN*.**VINC**Value increment within range. Used only with integer ranges (such as for keypoint numbers). Defaults to 1. *VINC* cannot be negative.**KABS**

Absolute value key:

- 0** Check sign of value during selection.
- 1** Use absolute value during selection (sign ignored).

## Command Default

All keypoints are selected.

## Notes

Selects a subset of keypoints or hard points. For example, to select a new set of keypoints based on keypoint numbers 1 through 7, use **KSEL,S,KP,,1,7**. The selected subset is used when the ALL label is entered (or implied) on other commands, such as **KLIST,ALL**. Only data identified by keypoint number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

This command is valid in any processor.

For selections based on non-integer numbers (coordinates, results, etc.), items that are within the range  $VMIN - Toler$  and  $VMAX + Toler$  are selected. The default tolerance  $Toler$  is based on the relative values of  $VMIN$  and  $VMAX$  as follows:

- If  $VMIN = VMAX$ ,  $Toler = 0.005 \times VMIN$ .
- If  $VMIN = VMAX = 0.0$ ,  $Toler = 1.0E-6$ .
- If  $VMAX \neq VMIN$ ,  $Toler = 1.0E-8 \times (VMAX - VMIN)$ .

Use the **SELTOL** command to override this default and specify  $Toler$  explicitly.

**Table 223 KSEL - Valid Item and Component Labels**

**Valid Item and Component Labels KSEL, Type, Item, Comp, VMIN, VMAX, VINC, KABS**

Item	Comp	Description
KP		Keypoint number.
EXT		Keypoint numbers on exterior of selected lines (ignore remaining fields).
HPT		Hard point number.
LOC	X,Y,Z	X, Y, or Z location in the active coordinate system.
MAT		Material number associated with the keypoint.
TYPE		Element type number associated with the keypoint.
REAL		Real constant set number associated with the keypoint.
ESYS		Element coordinate system associated with the keypoint.

## Menu Paths

**Utility Menu>Select>Entities**

**KSLL**, *Type*

Selects those keypoints contained in the selected lines.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of keypoint select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

**Notes**

This command is valid in any processor.

**Menu Paths**

Utility Menu>Select>Entities

---

**KSLN**, *Type*

Selects those keypoints associated with the selected nodes.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of keypoint select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

**Notes**

Valid only if the nodes were generated by a *meshing* operation [**KMESH**, **LMESH**, **AMESH**, **VMESH**] on a solid model that contains the associated keypoints.



This command is valid in any processor.

## Menu Paths

Utility Menu>Select>Entities

### KSUM

Calculates and prints geometry statistics of the selected keypoints.

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

Calculates and prints geometry statistics (centroid location, moments of inertia, etc.) associated with the selected keypoints. Geometry items are reported in the global Cartesian coordinate system. A unit density is assumed, irrespective of any material associations [**KATT**, **MAT**]. Items calculated by **KSUM** and later retrieved by a **\*GET** or **\*VGET** command are valid only if the model is not modified after the **KSUM** command is issued.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Keypoints

**KSYMM**, *Ncomp*, *NP1*, *NP2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Generates a reflected set of keypoints.

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Ncomp*

Symmetry key:

**X**

X (or R) symmetry (default).

**Y**

Y (or  $\theta$ ) symmetry.

**Z**

Z (or  $\Phi$ ) symmetry.

#### *NP1*, *NP2*, *NINC*

Reflect keypoints from pattern beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and pattern is all selected keypoints [**KSEL**]. If *Ncomp* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

#### *KINC*

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

**NOELEM**

Specifies whether nodes and elements are also to be generated:

- 0**  
Generate nodes and point elements associated with the original keypoints, if they exist.
- 1**  
Do not generate nodes and elements.

**IMOVE**

Specifies whether keypoints will be moved or newly defined:

- 0**  
Generate additional keypoints.
- 1**  
Move original keypoints to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

## Notes

Generates a reflected set of keypoints (and corresponding mesh) from a given keypoint pattern by a symmetry reflection (see analogous node symmetry command, **NSYM**). The MAT, TYPE, REAL, and ESYS attributes are based upon the keypoints in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. Keypoints in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Reflect>Keypoints**

---

### **KTRAN**, *KCNTO*, *NP1*, *NP2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

**Transfers a pattern of keypoints to another coordinate system.**

[PREP7: Keypoints](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KCNTO**

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system.

**NP1, NP2, NINC**

Transfer keypoints from pattern beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and pattern is all selected keypoints [**KSEL**]. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

**KINC**

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

**NOELEM**

Specifies whether nodes and elements are also to be generated:

- 0**  
Generate nodes and point elements associated with the original keypoints, if they exist.
- 1**  
Do not generate nodes and elements.

**IMOVE**

Specifies whether keypoints will be moved or newly defined:

- 0**  
Generate additional keypoints.
- 1**  
Move original keypoints to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

**Notes**

Transfers a pattern of keypoints (and corresponding mesh) from one coordinate system to another (see analogous node transfer command, **TRANSFER**). The MAT, TYPE, REAL, and ESYS attributes are based upon the keypoints in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. Coordinate values are interpreted in the active coordinate system and are transferred directly. Solid modeling in a toroidal coordinate system is not recommended.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Keypoints**

**KUSE, KEY**

**Specifies whether or not to reuse the factorized matrix.**

**SOLUTION: Load Step Options**

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

**KEY**

Reuse key:

- 0**  
Program decides whether or not to reuse the previous factorized stiffness matrix.
- 1**  
Force the previous factorized stiffness matrix to be reused. Used mainly in a restart. Forcing reuse of the matrix is a nonstandard use of the program, and should be done with caution. For instance, using this option and changing the number of elements, or the number or type of degrees of freedom, may cause an abort.
- 1**  
All element matrices are reformed and are used to reform a new factorized stiffness matrix.

**Command Default**

Program makes decision.

## Notes

Overrides the program logic to determine whether or not to reuse the previous factorized stiffness matrix for each substep of this load step. Applies only to static or full transient analyses and to full harmonic analyses if the frequency is not changed for continuing loadsteps. For full harmonic analyses, only  $KEY = 1$  or  $KEY = 0$  is valid.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Reuse LN22 Matrix**  
**Main Menu>Solution>Load Step Opts>Other>Reuse LN22 Matrix**

---

**KWPAVE**,  $P1, P2, P3, P4, P5, P6, P7, P8, P9$

**Moves the working plane origin to the average location of keypoints.**

[DATABASE: Working Plane](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

$P1, P2, P3, \dots, P9$

Keypoints used in calculation of the average. At least one must be defined. If  $P1 = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

## Notes

Moves the origin of the working plane to the average of the specified keypoints. Averaging is based on the active coordinate system.

This command is valid in any processor.

## Menu Paths

**Utility Menu>WorkPlane>Offset WP to>Keypoints**

---

**KWPLAN**,  $WN, KORIG, KXAX, KPLAN$

**Defines the working plane using three keypoints.**

[DATABASE: Working Plane](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

$WN$

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If  $WN$  is a negative value, the viewing direction will not be modified. If fewer than three points are used, the viewing direction of window  $WN$  will be used instead to define the normal to the working plane.

**KORIG**

Keypoint number defining the origin of the working plane coordinate system. If *KORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**KXAX**

Keypoint number defining the x-axis orientation (defaults to the x-axis being parallel to the global X-axis; or if the normal to the working plane is parallel to the global X-axis, then defaults to being parallel to the global Y-axis).

**KPLAN**

Keypoint number defining the working plane (the normal defaults to the present display view [[/VIEW](#)] of window *WN*).

## Notes

Defines a working plane to assist in picking operations using three keypoints as an alternate to the [WPLANE](#) command. The three keypoints also define the working plane coordinate system. A minimum of one keypoint (at the working plane origin) is required. Immediate mode may also be active. See [WPSTYL](#) command to set the style of working plane display.

This command is valid in any processor.

## Menu Paths

**Utility Menu>WorkPlane>Align WP with>Keypoints**

---



---

## L Commands

**L**, *P1*, *P2*, *NDIV*, *SPACE*, *XV1*, *YV1*, *ZV1*, *XV2*, *YV2*, *ZV2*

**Defines a line between two keypoints.**

PREP7:Lines

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **P1**

Keypoint at the beginning of line. If  $P1 = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### **P2**

Keypoint at the end of line.

### **NDIV**

Number of element divisions within this line. Normally this field is not used; specifying divisions with **LESIZE**, etc. is recommended.

### **SPACE**

Spacing ratio. Normally this field is not used, as specifying spacing ratios with the **LESIZE** command is recommended. If positive, *SPACE* is the nominal ratio of the last division size (at *P2*) to the first division size (at *P1*). If the ratio is greater than 1, the division sizes increase from *P1* to *P2*, and if less than 1, they decrease. If *SPACE* is negative, then  $|SPACE|$  is the nominal ratio of the center division size to those at the ends.

The following fields are used only if specified end slopes on the line are desired, otherwise zero curvature end slopes will be automatically calculated to produce a line which is "straight" in the active coordinate system. To specify end slopes, use the following fields to define a "slope vector" (one for each end of the line, if desired) that has its tail at the origin and its head at the point *XV*, *YV*, *ZV* in the active coordinate system [**CSYS**]. The corresponding end slope of the line will then be parallel to this "slope vector."

### **XV1**, **YV1**, **ZV1**

Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P1* end of the line. The tail of the vector is at the origin of the coordinate system.

### **XV2**, **YV2**, **ZV2**

Location of the head of the "slope vector" corresponding to the slope at the *P2* end of the line.

## Notes

Defines a line between two keypoints from *P1* to *P2*. The line shape may be generated as "straight" (in the active coordinate system) or curved. The line shape is invariant with coordinate system after it is generated. Note that solid modeling in a toroidal coordinate system is not recommended. A curved line is limited to 180°. Lines may be redefined only if not yet attached to an area.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Lines>Lines>In Active Coord**

**L2ANG**, *NL1*, *NL2*, *ANG1*, *ANG2*, *PHIT1*, *PHIT2***Generates a line at an angle with two existing lines.**

PREP7:Lines

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**NL1**

Number of the first line to be hit (touched by the end of the new line). If negative, assume *P1* (see below) is the second keypoint of the line instead of the first. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**NL2**

Number of the second line to be hit. If negative, assume *P3* is the second keypoint of the line instead of the first.

**ANG1**

Angle of intersection (usually zero or 180) of generated line with tangent to first line.

**ANG2**

Angle of intersection (usually zero or 180) of generated line with tangent to second line.

**PHIT1**

Number to be assigned to keypoint generated at hit location on first line (defaults to lowest available keypoint number [NUMSTR]).

**PHIT2**

Number to be assigned to keypoint generated at hit location on second line (defaults to lowest available keypoint number [NUMSTR]).

**Notes**

Generates a straight line (*PHIT1-PHIT2*) at an angle (*ANG1*) with an existing line *NL1* (*P1-P2*) and which is also at an angle (*ANG2*) with another existing line *NL2* (*P3-P4*). If the angles are zero the generated line is tangent to the two lines. The *PHIT1* and *PHIT2* locations on the lines are automatically calculated. Line *P1-P2* becomes *P1-PHIT1*, *P3-P4* becomes *P3-PHIT2*, and new lines *PHIT1-P2*, *PHIT2-P4*, and *PHIT1-PHIT2* are generated. Line divisions are set to zero (use **LESIZE**, etc. to modify).

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Angle to 2 Lines**

**Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Norm to 2 Lines**

**L2TAN**, *NL1*, *NL2***Generates a line tangent to two lines.**

PREP7:Lines

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**NL1**

Number of the first line generated line is tangent to. If negative, assume *P1* (see below) is the second keypoint of the line instead of the first. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).



**NL2**

Number of the second line generated line is tangent to. If negative, assume  $P3$  is the second keypoint of the line instead of the first.

**Notes**

Generates a line ( $P2-P3$ ) tangent at point  $P2$  to line  $NL1$  ( $P1-P2$ ) and tangent at point  $P3$  to line  $NL2$  ( $P3-P4$ ).

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Tan to 2 Lines**

**LANG**,  $NL1$ ,  $P3$ ,  $ANG$ ,  $PHIT$ ,  $LOCAT$ 

**Generates a straight line at an angle with a line.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NL1**

Number of the line to be hit (touched by the end of the new line). If negative, assume  $P1$  (see below) is the second keypoint of the line instead of the first. If  $NL1 = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**P3**

Keypoint at which generated line must end.

**ANG**

Angle of intersection of generated line  $PHIT-P3$  with tangent to line  $P1-P2$  at  $PHIT$ . If 0 (default), the generated line is tangent to  $NL1$  toward end  $P1$ ; if 90, the generated line is perpendicular to  $NL1$ . If 180, the generated line is tangent to  $NL1$  toward end  $P2$ .  $ANG$  can be any value, but is adjusted to the corresponding acute angle with respect to  $LOCAT$ . See "[Notes](#)" (p. 893) for a discussion of accuracy.

**PHIT**

Number to be assigned to keypoint generated at hit location (defaults to lowest available keypoint number [[NUMSTR](#)]).

**LOCAT**

Approximate location of  $PHIT$  in terms of the ratio of the distance along the line ( $NL1$ ) to the length of the line.  $LOCAT$  can range from 0 to 1. If  $LOCAT$  is blank, the point will be located with less speed and accuracy, and an arbitrary location may result.

**Notes**

Generates a straight line ( $PHIT-P3$ ) at an angle ( $ANG$ ) with a line  $NL1$  ( $P1-P2$ ). The location of  $PHIT$  on the line is automatically calculated. Line  $P1-P2$  becomes  $P1-PHIT$  and new lines  $PHIT-P2$  and  $PHIT-P3$  are generated. Line divisions are set to zero (use [LESIZE](#), etc. to modify).

$PHIT$  is positioned closest to  $LOCAT$  for the given angle,  $ANG$ . To ensure better performance, it is recommended that  $LOCAT$  be input, even if it is 0.

The program uses an iterative procedure to position *PHIT*. The procedure is not exact, with the result that the actual value of *ANG* will sometimes differ slightly from the specified value.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Lines>Lines>At angle to line**  
**Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Normal to Line**

---

## LARC, *P1*, *P2*, *PC*, *RAD*

Defines a circular arc.

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *P1*

Keypoint at one end of circular arc line. If *P1* = *P*, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### *P2*

Keypoint at other end of circular arc line.

### *PC*

Keypoint defining plane of arc and center of curvature side (with positive radius). Must not lie along the straight line from *P1* to *P2*. *PC* need not be at the center of curvature.

### *RAD*

Radius of curvature of the arc. If negative, assume center of curvature side is opposite to that defined by *PC*. If *RAD* is blank, *RAD* will be calculated from a curve fit through *P1*, *PC*, and *P2*.

## Notes

Defines a circular arc line from *P1* to *P2*. The line shape is generated as circular, regardless of the active coordinate system. The line shape is invariant with coordinate system after it is generated.

When dealing with a large radius arc (1e3), or if the location of the arc you create is far away from the origin of your coordinate system, anomalies may occur. You can prevent this by creating the arc at a smaller scale, and then scaling the model back to full size ([LSSCALE](#)).

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>By End KPs & Rad**  
**Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>Through 3 KPs**

---

**/LARC**, *XCENTR*, *YCENTR*, *XLRAD*, *ANGLE1*, *ANGLE2*

**Creates annotation arcs (GUI).**

GRAPHICS: Annotation

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***XCENTR***

Arc X center location ( $-1.0 < X < 1.0$ ).

***YCENTR***

Arc Y center location ( $-1.0 < Y < 1.0$ ).

***XLRAD***

Arc radius length.

***ANGLE1***

Starting angle of arc.

***ANGLE2***

Ending angle of arc. The arc is drawn counterclockwise from the starting angle, *ANGLE1*, to the ending angle, *ANGLE2*.

## Notes

Defines annotation arcs to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All arcs are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** command to set the attributes of the arc.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

**LAREA**, *P1*, *P2*, *NAREA*

**Generates the shortest line between two keypoints on an area.**

PREP7: Lines

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***P1***

First keypoint of line to be generated. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

***P2***

Second keypoint of line to be generated.

***NAREA***

Area containing *P1* and *P2*, or area to which generated line is to be parallel.

## Notes

Generates the shortest line between two keypoints,  $P1$  and  $P2$ , both of which lie on an area. The generated line will also lie on the area.  $P1$  and  $P2$  may also be equidistant (in global Cartesian space) from the area (and on the same side of the area), in which case a line parallel to the area is generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Overlaid on Area**

---

**LARGE**,  $IR$ ,  $IA$ ,  $IB$ ,  $IC$ ,  $Name$ ,  $--$ ,  $--$ ,  $FACTA$ ,  $FACTB$ ,  $FACTC$

**Finds the largest (the envelope) of three variables.**

[POST26: Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *IR*

Arbitrary reference number assigned to the resulting variable (2 to  $NV$  [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

### *IA, IB, IC*

Reference numbers of the three variables to be operated on. If only two, leave  $IC$  blank. If only one, leave  $IB$  blank also.

### *Name*

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

### *--, --*

Unused fields.

### *FACTA, FACTB, FACTC*

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

## Notes

Creates a new variable by finding the largest of up to three variables according to the operation:

$$IR = \text{Largest of } (FACTA \times IA, FACTB \times IB, FACTC \times IC)$$

The comparison is done at each time location, so that the new variable is the "envelope" of the three existing variables.

## Menu Paths

**Main Menu>TimeHist Postpro>Math Operations>Find Maximum**

---

**LATT**, *MAT*, *REAL*, *TYPE*, --, *KB*, *KE*, *SECNUM*

**Associates element attributes with the selected, unmeshed lines.**

PREP7: Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***MAT*, *REAL*, *TYPE***

Material number, real constant set number, and type number to be associated with selected, unmeshed lines.

--

Unused field.

***KB*, *KE***

Beginning and ending orientation keypoints to be associated with selected, unmeshed lines. ANSYS uses the location of these keypoints to determine how to orient beam cross sections during beam meshing. Beam elements may be created along a line with a constant orientation by specifying only one orientation keypoint (*KB*), or a pre-twisted beam may be created by selecting different orientation keypoints at each end of the line (*KB* and *KE*). (For a line bounded by two keypoints (*KP1* and *KP2*), the orientation vector at the beginning of the line extends from *KP1* to *KB*, and the orientation vector at the end of the line extends from *KP2* to *KE*. The orientation vectors are used to compute the orientation nodes of the elements.)

***SECNUM***

Section identifier to be associated with selected, unmeshed lines. For details, see the description of the **SECTYPE** and **SECNUM** commands.

## Notes

The element attributes specified by the **LATT** command will be used when the lines are meshed.

Lines subsequently generated from the lines will also have the attributes specified by *MAT*, *REAL*, *TYPE*, and *SECNUM*. If a line does not have these attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT**, **REAL**, **TYPE**, and **SECNUM** command settings.

In contrast, the values specified by *KB* and *KE* apply only to the selected lines; that is, lines subsequently generated from these lines will not share these attributes. Similarly, if a line does not have *KB* and *KE* attributes associated with it via the **LATT** command at the time it is meshed, ANSYS cannot obtain the attributes from elsewhere. See the discussion on beam meshing in [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide* for more information.

Reissue the **LATT** command (before lines are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association. If any of the arguments are defined as -1, then that value will be left unchanged in the selected set..

In some cases, ANSYS can proceed with a line meshing operation even when no logical element type has been assigned via **LATT,,,TYPE** or **TYPE**. See [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide* for more information about setting element attributes.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh Attributes>All Lines**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked Lines**

**LAYER**, *NUM*

**Specifies the element layer for which data are to be processed.**

POST1:Controls

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

**NUM**

Layer-processing mode:

***N***

The layer number to process. The default value is 0.

**FCMAX**

Processes the layer with the largest failure criteria.

## Command Default

The default is  $N = 0$ , meaning that the entire element is considered to be the default "layer." Accordingly, the results data are from the bottom of the bottom layer and the top of the top layer.

## Notes

Specifies the element layer for which results data are to be listed, plotted, or otherwise processed.

Applies to stress and strain data for layered elements [SHELL163](#), [SHELL181](#), [SHELL281](#), [ELBOW290](#), [SOLID185](#), [SOLID186](#), [SOLSH190](#), [SHELL208](#), [SHELL209](#), [REINF264](#), and [REINF265](#); heat flux and heat gradient for [SHELL131](#) and [SHELL132](#).

The **SHELL** command may then be used (with shell elements) to specify a location (TOP, MID, BOT) within the layer for output. (The **SHELL** command does not apply to thermal shell elements [SHELL131](#) and [SHELL132](#).) Transverse shear stresses for MID are linearly averaged from TOP and BOT, and do not reflect a parabolic distribution. Setting KEYOPT(8) = 2 for [SHELL181](#), [SHELL281](#), [SHELL208](#), [SHELL209](#), and [ELBOW290](#) writes the mid-surface values directly to the results file and yields more accurate values than linear averaging.

Because energy is a per-element quantity, you cannot use this command for energy output.

When using the **LAYER** command with [SHELL181](#), [SOLID185](#), [SOLID186](#), [SOLSH190](#), [SHELL208](#), [SHELL209](#), [SHELL281](#), and [ELBOW290](#), KEYOPT(8) must be set to 1 (or 2 for [SHELL181](#), [SHELL281](#), [ELBOW290](#), [SHELL208](#), and [SHELL209](#)) in order to store results for all layers.

When  $NUM = FCMAX$ , you must provide the failure criterion input. If using **FC** input, ANSYS processes all structural elements.

When  $NUM = FCMAX$ , you must provide the failure criterion input. For more information, see the documentation for the **FC** command.

Use this command with **RSYS**,**LSYS** to display results in the layer coordinate system for a particular layer.

For the ANSYS LS-DYNA product, this command works differently than described above. For [SHELL163](#), you must first use **EDINT** during the solution phase to define the integration points for which you want output data. Be aware that the output location for [SHELL163](#) data is always at the integration point, so "top" and "bottom" refer to the top or bottom integration point, not necessarily the top or bottom surface. For more information, see the [ANSYS LS-DYNA User's Guide](#).

## Menu Paths

Main Menu>General Postproc>Options for Outp  
Utility Menu>List>Results>Options

### LAYERP26, NUM

Specifies the element layer for which data are to be stored.

POST26: Controls

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

#### NUM

Layer-processing mode:

#### N

The layer number to process. The default value is 1.

## Command Default

The default is  $N = 1$ , meaning that results data are from the "first" layer. If KEYOPT(8) = 0, "first" layer actually means the bottom of the bottom layer and the top of the top layer.

## Notes

Defines the element layer for which results data are to be stored for postprocessing. Applies to stress and strain data for layered elements [BEAM161](#), [SHELL163](#), [SHELL181](#), [SOLID185](#), [SOLID186](#), [SOLSH190](#), [SHELL208](#), [SHELL209](#), [SHELL281](#), [REINF265](#), and [ELBOW290](#).

The **SHELL** command can be used (for shell elements) to specify a location (TOP, MID, BOT) within the layer for selection on the **ESOL** command. Transverse shear stresses for MID are linearly averaged from TOP and BOT, and do not reflect a parabolic distribution. Setting KEYOPT(8) = 2 for [SHELL181](#), [SHELL208](#), [SHELL209](#), [SHELL281](#), and [ELBOW290](#) writes the mid-surface values directly to the results file and yields more accurate values than linear averaging.

That this command cannot be used for energy output, as energy is a per-element quantity.

When using the **LAYERP26** command with [SHELL181](#), [SOLID185](#), [SOLID186](#), [SOLSH190](#), [SHELL208](#), or [SHELL209](#), KEYOPT(8) must be set to 1 (or 2 for [SHELL181](#), [SHELL208](#), [SHELL209](#), [SHELL281](#), and [ELBOW290](#)) in order to store results for all layers.

For the ANSYS LS-DYNA product, this command works differently than described above. For [SHELL163](#) and [BEAM161](#), you must first use **EDINT** during the solution phase to define the integration points for which you want output data. Be aware that the output location for [SHELL163](#) data is always at the integration point, so "top" and "bottom" refer to the top or bottom integration point, not necessarily the top or bottom surface. For more information, see the [ANSYS LS-DYNA User's Guide](#).

In POST26, the **ESOL** data stored is based on the active **LAYERP26** specification at the time the data is stored. To store data at various specifications (for example, layers 2 and 5), issue a **STORE** command before each new specification.

## Menu Paths

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

**LAYLIST**, *IEL*, *LAYR1*, *LAYR2*, *Mplab1*, *Mplab2*

**Lists real constants material properties for layered elements.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### *IEL*

Element number to be listed. If ALL, list all selected elements [**ESEL**] of the appropriate type. If blank and the current element type is a layered element type, list data from the current real constant table in the layered format.

### *LAYR1*, *LAYR2*

Range of layer numbers to be listed. If *LAYR1* is greater than *LAYR2*, a reverse order list is produced. *LAYR1* defaults to 1. *LAYR2* defaults to *LAYR1* if *LAYR1* is input or to the number of layers if *LAYR1* is not input.

### *Mplab1*, *Mplab2*

Material property labels (e.g., EX) to be listed along with the layer real constants.

## Notes

Lists real constants and any two material properties for layered shell and solid elements.

If matrix input is selected (KEYOPT(2) = 2 or 3), *LAYR1*, *LAYR2*, *Mplab1*, and *Mplab2* are not used.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Elements>Layered Elements

Utility Menu>List>Properties>Layer Data

**LAYPLOT**, *IEL*, *LAYR1*, *LAYR2*

**Displays the layer stacking sequence for layered elements.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### *IEL*

Element number for the display. If blank and the current element type is a layered element type, display data from the current real constant table.



**LAYR1, LAYR2**

Range of layer numbers to be displayed. If *LAYR1* is greater than *LAYR2*, a reversed order display is produced. Up to 20 layers may be displayed at a time. *LAYR1* defaults to 1. *LAYR2* defaults to *LAYR1* if *LAYR1* is input or to the number of layers (or to 19+*LAYR1*, if smaller) if *LAYR1* is not input.

**Notes**

Displays the layer-stacking sequence as defined in the real constant table for layered shell and solid elements in a form where the layers are visible (like a sheared deck of cards).

The element x-axis is shown as 0.0 degrees.

Layers are cross-hatched and color-coded for clarity. The hatch lines indicate the layer angle (real constant THETA) and the color coding is for material identification (real constant MAT).

The actual orientation of a specific layer in three-dimensional space can be seen using **/PSYMB,LAYR**. To use **/PSYMB,LAYR** with smeared reinforcing elements (**REINF265**), first set the vector-mode graphics option (**/DEVICE,VECTOR,1**).

Layer thickness can be displayed using the **/ESHAPE** and **ELOT** commands.

This command is valid in any processor.

**Menu Paths**

Utility Menu>Plot>Layered Elements

**LCABS, LCNO, KABS**

Specifies absolute values for load case operations.

POST1: Load Case Calculations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**LCNO**

Load case pointer number. If ALL, apply to all selected load cases [**LCSEL**].

**KABS**

Absolute value key:

**0**

Use algebraic values of load case *LCNO* in operations.

**1**

Use absolute values of load case *LCNO* in operations.

**Command Default**

Use algebraic values.

## Notes

Causes absolute values to be used in the load case operations [**LCASE** or **LCOPER**]. Absolute values are taken prior to assigning a load case factor [**LCFACT**] and are applied only to defined load cases [**LCDEF**].

## Menu Paths

**Main Menu>General Postproc>Load Case>Calc Options>Absolut Value**

---

### **LCASE**, *LCNO*

**Reads a load case into the database.**

[POST1: Load Case Calculations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *LCNO*

Load case pointer number [**LCDEF**,**STAT**]. Defaults to 1.

## Command Default

Load case 1.

## Notes

Reads a load case into the database. Load cases are created as described on the **LCDEF** or **LCWRITE** commands. The results portion of the database and the applied forces and displacements are cleared before reading the data in. Absolute values [**LCABS**] and scale factors [**LCFACT**] can be applied during the read operation.

## Menu Paths

**Main Menu>General Postproc>Load Case>Read Load Case**

---

### **LCCALC**

**Specifies "Load case settings" as the subsequent status topic.**

[POST1: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (*Jobname.LOG*) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

This command is also valid for [rezoning](#).

## Menu Paths

Utility Menu>List>Status>General Postproc>Load Case Calcs

### LCCAT, NL1, NL2

Concatenates multiple lines into one line for mapped meshing.

PREP7: Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### NL1, NL2

Lines to be concatenated. If  $NL1 = ALL$ ,  $NL2$  is ignored and all selected lines [**LSEL**] are concatenated.

If  $NL1 = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for  $NL1$  ( $NL2$  is ignored).

## Notes

Concatenates multiple, adjacent lines (the input lines) into one line (the output line) in preparation for mapped meshing. An area that contains too many lines for mapped meshing can still be mapped meshed if some of the lines in that area are first concatenated (see [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide* for details on mapped meshing restrictions).

**LCCAT** is meant to be used solely for meshing and cannot be used for any other purposes. Specifically, (a) the output line and any areas that have the output line on their line list [**ALIST**] cannot be used as input to any other solid modeling operation (not even another **LCCAT** command); and (b) the output line cannot accept solid model boundary conditions [**DL**, **SFL**].

The output line will take on the element divisions of the input lines and will not accept element divisions that are directly assigned [**LESIZE**]. The output line from the **LCCAT** operation will be coincident with the input lines and the input lines will be retained. Consider the **LCOMB** command instead of **LCCAT** if you wish to delete the input lines and if the lines to be combined have similar slopes at the common keypoint(s). When an **LCCAT** command is issued, area line lists [**ALIST**] that contain *all* of the input lines will be updated so that the area line lists refer to the output line instead of the input lines. Deletion of the output line [**LDELE**] effectively reverses the **LCCAT** operation and restores area line lists to their original condition.

You can use the **LSEL** command to select lines that were created by concatenation, and then follow it with an **LDELE,ALL** command to delete them. Also see [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide* for a discussion on how to easily select and delete concatenated lines in one step.

## Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>Concatenate>Lines

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>Concatenate>Lines

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Geometry Controls>Concatenate Lines

**LCDEF**, *LCNO*, *LSTEP*, *SBSTEP*, *KIMG*

**Creates a load case from a set of results on a results file.**

[POST1: Load Case Calculations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***LCNO***

Arbitrary pointer number (1-99) to be assigned to the load case specified by *LSTEP*, *SBSTEP* and by the **FILE** command. Defaults to 1 + previous value.

***LSTEP***

Load step number to be defined as the load case. Defaults to one.

***SBSTEP***

Substep number. Defaults to the last substep of the load step.

***KIMG***

Used only with results from complex analyses:

**0**

Use real part of complex solution

**1**

Use imaginary part.

**Notes**

Creates a load case by establishing a pointer to a set of results on a results file (written during the ANSYS solution phase). This pointer (*LCNO*) can then be used on the **LCASE** or **LCOPER** commands to read the load case data into the database.

Issue **LCDEF,ERASE** to delete *all* load case pointers (and *all* load case files, if any). Issue **LCDEF,LCNO,ERASE** to delete only the specific load case pointer *LCNO* (and its file, if any). With the ERASE options, all pointers are deleted; however only files with the default extension [**LCWRITE**] are deleted. Issue **LCDEF,STAT** for status of all selected load cases [**LCSEL**], or **LCDEF,STAT,ALL** for status of all load cases. The **STAT** command may be used to list all load cases. See also **LCFILE** to establish a pointer to a set of results on a load case file (written by **LCWRITE**). Harmonic element data read from a result file load case is stored at the zero-degree position.

**Menu Paths**

**Main Menu>General Postproc>Load Case>Create Load Case**

**Main Menu>General Postproc>Load Case>Erase Load Case**

**Main Menu>General Postproc>Load Case>List Load Cases**

**LCFACT**, *LCNO*, *FACT*

**Defines scale factors for load case operations.**

[POST1: Load Case Calculations](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***LCNO***

Load case pointer number. If ALL, apply to all selected load cases [**LCSEL**].

**FACT**

Scale factor applied to load case *LCNO*. Blank defaults to 1.0.

**Command Default**

All factors are 1.0.

**Notes**

Defines scale factors to be used in the load case operations [**LCASE** or **LCOPER**]. Scale factors are applied after an absolute value operation [**LCABS**] and are applied only to defined load cases [**LCDEF**].

**Menu Paths**

**Main Menu>General Postproc>Load Case>Calc Options>Scale Factor**

**LCFILE**, *LCNO*, *Fname*, *Ext*, --

**Creates a load case from an existing load case file.**

[POST1: Load Case Calculations](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***LCNO***

Arbitrary (1-99) pointer number assigned to this load case.

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to *Jobname*.

***Ext***

Filename extension (8 character maximum).

The extension defaults to the *LCNO* value preceded by an "L" (for values 10–99) or by an "L0" (for values 1–9).

--

Unused field.

**Notes**

Creates a load case by establishing a pointer to an existing load case file [**LCWRITE**]. This pointer (*LCNO*) can then be used on the **LCASE** or **LCOPER** commands to read the load case data into the database. This command is typically used to reestablish load case pointers in a new ANSYS session (pointers are not saved on the database file), or when more than one pointer to a single load case is desired. See the **LCDEF** command for status and erase operations. See also **LCDEF** to establish a pointer to a set of results on a results file (written during the ANSYS solution phase).

## Menu Paths

Main Menu>General Postproc>Load Case>Create Load Case

---

### LCLEAR, NL1, NL2, NINC

Deletes nodes and line elements associated with selected lines.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### NL1, NL2, NINC

Delete mesh for lines *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and the mesh for all selected lines [**LSEL**] is deleted. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

## Notes

Deletes *all* nodes and line elements associated with selected lines (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed lines and nodes associated with non-line elements will not be deleted. Attributes assigned as a result of **LATT** are maintained. In the program's response to the command, if a line or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

## Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Lines

---

### LCOMB, NL1, NL2, KEEP

Combines adjacent lines into one line.

[PREP7: Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### NL1

Number of the first line to be combined. If *NL1* = ALL, *NL2* is ignored and all selected lines [**LSEL**] are combined. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* is ignored).

#### NL2

Number of the second line to be combined.

#### KEEP

Specifies whether to keep the input entities:

**0**

Delete lines *NL1* and *NL2* and their common keypoint. Keypoints will not be deleted if they are meshed or if they are attached to other lines. Lines will not be deleted if they are attached to different areas.

1

Keep *NL1*, *NL2*, and their common keypoint. (The common keypoint will *not* be attached to the output line.)

## Notes

Combines adjacent lines into one line (the output line). This operation will effectively "undo" the **LDIV** operation. Line divisions are set to zero (use **LESIZE**, etc. to modify). Lines attached to the same area(s) can also be combined. See also the **LCCAT** command for line concatenation capability.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Add>Lines**

**LCOPER**, *Oper*, *LCASE1*, *Oper2*, *LCASE2*

**Performs load case operations.**

POST1: Load Case Calculations

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### *Oper*

Valid operations are:

#### **ZERO**

Zero results portion of database (*LCASE1* ignored).

#### **SQUA**

Square database values (*LCASE1* ignored).

#### **SQRT**

Square root of database (absolute) values (*LCASE1* ignored).

#### **LPRIN**

Recalculate line element principal stresses (*LCASE1* ignored). Stresses are as shown for the NMISC items of the **ETABLE** command for the specific line element type.

#### **ADD**

Add *LCASE1* to database values.

#### **SUB**

Subtract *LCASE1* from database values.

#### **SRSS**

Square root of the sum of the squares of database and *LCASE1*.

#### **MIN**

Compare and save in database the algebraic minimum of database and *LCASE1*.

#### **MAX**

Compare and save in database the algebraic maximum of database and *LCASE1*.

#### **ABMN**

Compare and save in database the absolute minimum of database and *LCASE1* (based on magnitudes, then apply the corresponding sign).

**ABMX**

Compare and save in database the absolute maximum of database and *LCASE1* (based on magnitudes, then apply the corresponding sign).

**LCASE1**

First load case in the operation (if any). See *LCNO* of the **LCDEF** command. If ALL, repeat operations using all selected load cases [**LCSEL**].

**Oper2**

Valid operations are:

**MULT**

Multiplication: *LCASE1\*LCASE2*

**CPXMAX**

This option does a phase angle sweep to calculate the true maximum of derived stresses for a complex solution where *LCASE1* is the real part and *LCASE2* is the imaginary part. The *Oper* field is not applicable with this option. Also, the **LCABS** and **SUMTYPE** commands have no effect on this option. The value of S3 will be a minimum. This option does not apply to derived displacement amplitude (USUM.)

**LCASE2**

Second load case. Used only with *Oper2* operations.

## Notes

**LCOPER** operates on the database and one or two load cases according to:

$$\text{Database} = \text{Database } \textit{Oper} (\textit{LCASE1 } \textit{Oper2 } \textit{LCASE2})$$

where operations *Oper* and *Oper2* are as described above. Absolute values and scale factors may be applied to the load cases before the operations [**LCABS**, **LCFACT**]. If *LCASE1* is not specified, only operation *Oper* is performed on the current database. If *LCASE2* is specified, operation *Oper2* will be performed *before* operation *Oper*. If *LCASE2* is not specified, operation *Oper2* is ignored. Solution items not contained [**OUTRES**] in either the database or the applicable load cases will result in a null item during a load case operation. Harmonic element data read from a result file load case are processed at zero degrees. All load case combinations are performed in the solution coordinate system, and the data resulting from load case combinations are stored in the solution coordinate system. The resultant data are then transformed to the active results coordinate system [**RSYS**] when listed or displayed.

If Oper2=CPXMAX, the derived stresses and strain calculation do not apply to line elements.

## Menu Paths

**Main Menu>General Postproc>Load Case>Add**  
**Main Menu>General Postproc>Load Case>Line Elem Stress**  
**Main Menu>General Postproc>Load Case>Min & Max**  
**Main Menu>General Postproc>Load Case>Square**  
**Main Menu>General Postproc>Load Case>Square Root**  
**Main Menu>General Postproc>Load Case>SRSS**  
**Main Menu>General Postproc>Load Case>Subtract**

---



**LCSEL**, *Type*, *LCMIN*, *LCMAX*, *LCINC*

**Selects a subset of load cases.**

POST1: Load Case Calculations

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of select:

#### **S**

Select a new set.

#### **R**

Reselect a set from the current set.

#### **A**

Additionally select a set and extend the current set.

#### **U**

Unselect a set from the current set.

#### **ALL**

Restore the full set.

#### **NONE**

Unselect the full set.

#### **INVE**

Invert the current set (selected becomes unselected and vice versa).

#### **STAT**

Display the current select status.

### *LCMIN*

Minimum value of load case pointer range.

### *LCMAX*

Maximum value of load case pointer range. *LCMAX* defaults to *LCMIN*.

### *LCINC*

Value increment within range. Defaults to 1. *LCINC* cannot be negative.

## Command Default

All load cases are selected.

## Notes

Selects a subset of load cases for other operations. For example, to select a new set of load cases based on load cases 1 through 7, use **LCSEL,S,1,7**. The subset is used when the ALL label is entered (or implied) on other commands, such as **LCFACT**, **LCABS**, **LCOPER**, etc. Load cases are flagged as selected and unselected; no load case pointers [**LCDEF**, **LCWRITE**, **LCFILE**] are actually deleted from the database.

## Menu Paths

**Main Menu>General Postproc>Load Case>Calc Options>Sele Ld Cases**

**LCSL**, *NL1, NL2, NL3, NL4, NL5, NL6, NL7, NL8, NL9*

**Divides intersecting lines at their point(s) of intersection.**

PREP7: Booleans

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NL1, NL2, NL3, . . . , NL9***

Numbers of lines to be intersected. If *NL1* = ALL, *NL2* to *NL9* are ignored and the intersection of all selected lines is found. If *NL1* = P, use graphical picking to specify lines (*NL2* to *NL9* are ignored).

**Notes**

Divides intersecting (classifies) lines at their point(s) of intersection. The original lines (and their corresponding keypoint(s)) will be deleted by default. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

**Menu Paths**

**This command cannot be accessed from a menu.**

**LCSUM**, *Lab*

**Specifies whether to process non-summable items in load case operations.**

POST1: Results

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***Lab***

Combination option

**(blank)**

Only combine summable items [default].

**ALL**

Combine all items including non summable items.

**Notes**

Allows non-summable items (e.g. plastic strains) to be included in load combinations. Issue **LCSUM,ALL** before the first load case operation (**LCXX** command). May also be used to include nonsummable items in the appending of a results file (**RAPPND** command).

**Menu Paths**

**This command cannot be accessed from a menu.**

**LCWRITE**, *LCNO*, *Fname*, *Ext*, --

**Creates a load case by writing results to a load case file.**

POST1: Load Case Calculations

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***LCNO***

Arbitrary pointer number (1-99) to be assigned to this load case.

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to the *LCNO* value preceded by an "L" (for values 10–99) or by an "L0" (for values 1–9).

--

Unused field.

## Notes

Creates a load case by writing the results data in the database to a load case file. The database remains unchanged by this operation. A pointer is also established to the written set of results on the load case file. This pointer (*LCNO*) can then be used on the **LCASE** or **LCOPER** commands to read the load case data into the database. By default, only summable results data (such as displacements, stresses, elastic strains) and constant results data (such as volume) are written to the load case file unless requested (**LCSUM** command). Non-summable results data (such as plastic strains, strain energy), boundary conditions, and nodal loads are not written to the load case file. The load case file may be named by default or by a user name. Rewriting to the same file overwrites the previous data. See the **LCDEF** command for status and erase operations.

## Menu Paths

**Main Menu>General Postproc>Load Case>Write Load Case**

## LCZERO

**Zeroes the results portion of the database.**

POST1: Load Case Calculations

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

## Notes

Often used before the **LCOPER** command. Same as **LCOPER,ZERO**.

## Menu Paths

Main Menu>General Postproc>Load Case>Zero Load Case

**LDELE**, *NL1*, *NL2*, *NINC*, *KSWP*

**Deletes unmeshed lines.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NL1*, *NL2*, *NINC***

Delete lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and all selected lines [**LSEL**] are deleted. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

***KSWP***

Specifies whether keypoints are also to be deleted:

**0**

Delete lines only.

**1**

Delete lines, as well as keypoints attached to lines but not attached to other lines.

## Notes

A line attached to an area cannot be deleted unless the area is first deleted.

## Menu Paths

Main Menu>Preprocessor>Meshing>Concatenate>Del Concats>Lines

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>Del Concats>Lines

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>Del Concats>Lines

Main Menu>Preprocessor>Modeling>Delete>Del Concats>Lines

Main Menu>Preprocessor>Modeling>Delete>Line and Below

Main Menu>Preprocessor>Modeling>Delete>Lines Only

**LDIV**, *NL1*, *RATIO*, *PDIV*, *NDIV*, *KEEP*

**Divides a single line into two or more lines.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NL1***

Number of the line to be divided. If negative, assume *P1* (see below) is the second keypoint of the line instead of the first for *RATIO*. If ALL, divide all selected lines [**LSEL**]. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

**RATIO**

Ratio of line length  $P1-PDIV$  to line length  $P1-P2$ . Must be between 0.0 and 1.0. Input ignored if  $NDIV > 2$ .

**PDIV**

Number to be assigned to keypoint generated at division location (defaults to lowest available keypoint number [**NUMSTR**]). Input ignored if  $NL1 = ALL$  or  $NDIV > 2$ . If  $PDIV$  already exists and lies on line  $NL1$ , divide line at  $PDIV$  ( $RATIO$  must also be 0.0). If  $PDIV$  already exists and does not lie on line  $NL1$ ,  $PDIV$  is projected and moved to the nearest point on line  $NL1$  (if possible).  $PDIV$  cannot be attached to another line, area, or volume.

**NDIV**

The number of new lines to be generated from old line (defaults to 2).

**KEEP**

Specifies whether to keep the input entities:

**0**

Modify old line to use new keypoints and slopes.

**1**

Do not modify old line. New lines will overlay old line and have unique keypoints.

## Notes

Divides a single line  $NL1$  (defined from keypoint  $P1$  to keypoint  $P2$ ) into two or more lines. Line  $NL1$  becomes the new line beginning with keypoint  $P1$  and new lines are generated ending at keypoint  $P2$ . If the line is attached to an area, the area will also be updated. Line divisions are set to zero (use **LESIZE**, etc. to modify).

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line into 2 Ln's**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line into N Ln's**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Lines w/ Options**

---

**LDRAG**,  $NK1, NK2, NK3, NK4, NK5, NK6, NL1, NL2, NL3, NL4, NL5, NL6$

**Generates lines by sweeping a keypoint pattern along path.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**$NK1, NK2, NK3, \dots, NK6$**

List of keypoints in the pattern to be dragged (6 maximum if using keyboard entry). If  $NK1 = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If  $NK1 = ALL$ , all selected keypoints (except those that define the drag path) will be swept along the path. A component name may also be substituted for  $NK1$ .

**$NL1, NL2, NL3, \dots, NL6$**

List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines.

## Notes

Generates lines (and their corresponding keypoints) by sweeping a given keypoint pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which the path lines are input (*NL1*, *NL2*, etc.). If the drag path is a single line (*NL1*), the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves. Keypoint and line numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]). For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Extrude>Keypoints>Along Lines**

**LDREAD**, *Lab*, *LSTEP*, *SBSTEP*, *TIME*, *KIMG*, *Fname*, *Ext*, --

**Reads results from the results file and applies them as loads.**

SOLUTION:FE Constraints

SOLUTION:FE Body Loads

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lab*

Valid load label:

#### **TEMP**

Temperatures from a thermal analysis are applied as body force nodal loads (**BF**) in a structural analysis, an explicit dynamic analysis, or other type of analysis.

If the thermal analysis uses [SHELL131](#) or [SHELL132](#), temperatures are applied as body force element loads (**BFE**). In most cases, only the top and bottom temperatures from [SHELL131](#) and [SHELL132](#) are used by the structural shell elements; any interior temperatures are ignored.

All temperatures are used for [SHELL181](#) using section input, and [SHELL281](#) using section input; for these elements, therefore, the number of temperature points at a node generated in the thermal model must match the number of temperature points at a node needed by the structural model.

When using [SHELL131](#) or [SHELL132](#) information for the **LDREAD** operation, all element types should specify the same set of thermal degrees of freedom.

Can also be temperatures applied as nodal loads or initial conditions (see *KIMG* description).

#### **FORC**

Forces from an electromagnetic analysis are applied as force loads (**F**) in a structural analysis or a FLOTRAN analysis. **LDREAD,FORC** reads coupling forces. When transferring static magnetic forces from a [SOLID117](#) model, ensure that all nodes and elements of the bodies intended to receive the forces are selected prior to transfer. See the discussion on [force computation](#) in the *Low-Frequency Electromagnetic Analysis Guide*.

For a full harmonic magnetic analysis, FORC represents the time-averaged force (use in conjunction with  $KIMG = 2$ ). Values are in the nodal coordinate system for the force loads (**F**).

**HGEN**

Heat generations from a magnetic analysis are applied as body force loads (**BFE**) in a thermal or a FLOTRAN analysis. For a full harmonic analysis, HGEN represents the time-averaged heat generation load (use in conjunction with  $KIMG = 2$ ).

**HFLU**

Heat fluxes from a FLOTRAN analysis are applied as surface loads in a thermal analysis.

**EHFLU**

Surface losses from a high frequency electromagnetic analysis are applied as time-average surface heat flux in a thermal analysis.

**JS**

Source current density from a current-conduction analysis are applied as body force loads (**BFE**). Values are in the global Cartesian coordinate system.

**EF**

Electric field element centroid values from an electrostatic analysis are applied as body force loads (**BFE**) in a magnetic analysis. Values are in the global Cartesian coordinate system.

**PRES**

Pressures from a FLOTRAN analysis are applied as surface loads (**SFE**) in a structural analysis. For shell elements, use the  $KIMG$  parameter to establish the face on which to apply the pressure.

**REAC**

Reaction loads from any analysis are applied as force loads (**F**) in any analysis. Values are in the nodal coordinate system.

**HFLM**

FLOTRAN predicted film coefficient (and associated bulk temperature) are applied as surface loads (film coefficient and bulk temperature) in a thermal analysis (**SFE**). FLOTRAN film coefficients may be either positive or negative, depending on the direction of heat transfer, but ANSYS thermal analyses require positive film coefficients. If the FLOTRAN film coefficient is negative, **LDREAD** adjusts the bulk temperature to be equal to twice the wall temperature minus the bulk temperature. The adjusted bulk temperature and the now positive film coefficient make the amount and direction of heat transfer consistent with the FLOTRAN analysis.

**H**

Nodal magnetic field values from magneto static analysis are applied as nodal loads (**BF**) in a high-frequency electromagnetic analysis. Values are in the global Cartesian coordinate system.

**FLRZ**

Lorentz forces ( $F = J * B$ ). Used only with **SOLID117**.

**LSTEP**

Load step number of the data set to be read. Defaults to 1. If LAST, ignore *SBSTEP* and *TIME* and read the last data set.

**SBSTEP**

Substep number (within *LSTEP*). If zero (or blank), *LSTEP* represents the last substep of the load step.

**TIME**

Time-point identifying the data set to be read. Used only if both *LSTEP* and *SBSTEP* are zero (or blank). If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. If *TIME* is beyond the last time point on the file, use the last time point.

**KIMG**

When used with results from harmonic analyses (**ANTYPE,HARMIC**) *KIMG* establishes which set of data to read:

- 0**  
Read the real part of the solution. Valid also for *Lab* = EHFLU to read in time-average heat flux.
- 1**  
Read the imaginary part of the solution.
- 2**  
Calculate and read the time-average part. Meaningful for *Lab* = HGEN or FORC.

When used with the PRES label, *KIMG* represents the shell element face on which to apply the pressure:

- 1**  
Apply pressure to face 1
- 2**  
Apply pressure to face 2

When used with the TEMP label, *KIMG* indicates how temperatures are to be applied. In an explicit dynamic analysis, *KIMG* = 0 is the only valid option for applying temperature loads.

- 0**  
Apply temperatures as body loads
- 1**  
Apply temperatures as nodal loads
- 2**  
Apply temperatures as initial conditions.

**Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

**Ext**

Filename extension (8 character maximum).

The extension defaults to RST if *Fname* is blank.

--

Unused field.

## Notes

Reads results data from the results file and applies them as loads. You also can use **LDREAD** to apply results from an analysis defined with one physics environment as loads on a second analysis using a different physics environment. The values are applied as loads for field coupling effects (such as output temperatures from a thermal analysis as input to a structural analysis). Nodal loads are applied only to selected nodes. Element loads are applied only to selected elements. Element surface loads are applied only to selected elements where all face nodes for that surface are selected. Additionally, to assure proper distribution of the surface loads, select *only* the nodes on the element face where the surface load is to be applied. Scaling and accumulation specifications are applied as the loads are read (**BFCUM** for body force loads (heat gener-



ation loads are not accumulated), **SFCUM** for surface loads, and **FCUM** for force loads). These commands do not work for tabular boundary conditions or temperature loads applied to an explicit dynamic analysis via **LDREAD**. Use the appropriate list command to list the results (**BFLIST** or **BFELIST** for body force loads, **SFELIST** for surface loads, and **FLIST** for force loads). Values may be redefined after being read by issuing **LDREAD** again with a different load step and substep, or time value.

When you use **LDREAD** in an explicit dynamic analysis to read in temperatures, you cannot use the **EDLOAD,,TEMP** command to apply temperature loading. Furthermore, any temperature loading defined by **LDREAD** cannot be listed or deleted by the **EDLOAD** command.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>From Therm Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>From Mag Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Forces>Forces>From Mag Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>Temp from ANSYS**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>Temp from Fluid**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>From Therm Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>From Elec An**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>Electric Field>From Elec An**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>From Mag Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>From Reactions**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>From Fluid Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Temperature>From Therm Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>From Fluid Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>From EMAG Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>From Fluid Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>From Mag Analy**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>From ANSYS**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>From Flotran**  
**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Temp From ANSYS**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>From Therm Analy**  
**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>From Mag Analy**  
**Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Forces>Forces>From Mag Analy**  
**Main Menu>Solution>Define Loads>Apply>Initial Condit'n>Temp from ANSYS**  
**Main Menu>Solution>Define Loads>Apply>Initial Condit'n>Temp from Fluid**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>From Therm Analy**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>From Elec An**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Other>Electric Field>From Elec An**  
**Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>From Mag Analy**  
**Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>From Reactions**  
**Main Menu>Solution>Define Loads>Apply>Structural>Pressure>From Fluid Analy**  
**Main Menu>Solution>Define Loads>Apply>Structural>Temperature>From Therm Analy**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Convection>From Fluid Analy**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>From EMAG Analy**

**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>From Fluid Analy**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>From Mag Analy**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>From ANSYS**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>From Flotran**  
**Main Menu>Solution>Loading Options>Temp From ANSYS**

**LESIZE**, *NL1*, *SIZE*, *ANGSIZ*, *NDIV*, *SPACE*, *KFORC*, *LAYER1*, *LAYER2*, *KYNDIV*

**Specifies the divisions and spacing ratio on unmeshed lines.**

PREP7: Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### **NL1**

Number of the line to be modified. If ALL, modify all selected lines [LSEL]. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

#### **SIZE**

If *NDIV* is blank, *SIZE* is the division (element edge) length. The number of divisions is automatically calculated from the line length (rounded upward to next integer). If *SIZE* is zero (or blank), use *ANGSIZ* or *NDIV*.

#### **ANGSIZ**

The division arc (in degrees) spanned by the element edge (except for straight lines, which always result in one division). The number of divisions is automatically calculated from the line length (rounded upward to next integer).

#### **NDIV**

If positive, *NDIV* is the number of element divisions per line. If -1 (and *KFORC* = 1), *NDIV* is assumed to be zero element divisions per line. TARGE169 with a rigid specification ignores *NDIV* and will always mesh with one element division.

#### **SPACE**

Spacing ratio. If positive, nominal ratio of last division size to first division size (if > 1.0, sizes increase, if < 1.0, sizes decrease). If negative, |*SPACE*| is nominal ratio of center division(s) size to end divisions size. Ratio defaults to 1.0 (uniform spacing). For layer-meshing, a value of 1.0 normally is used. If *SPACE* = FREE, ratio is determined by other considerations

#### **KFORC**

*KFORC* 0-3 are used only with *NL1* = ALL. Specifies which selected lines are to be modified.

**0**

Modify only selected lines having undefined (zero) divisions.

**1**

Modify all selected lines.

**2**

Modify only selected lines having fewer divisions (including zero) than specified with this command.

**3**

Modify only selected lines having more divisions than specified with this command.

**4**

Modify only nonzero settings for *SIZE*, *ANGSIZ*, *NDIV*, *SPACE*, *LAYER1*, and *LAYER2*. If *KFORC* = 4, blank or 0 settings remain unchanged.

***LAYER1***

Layer-meshing control parameter. Distance which defines the thickness of the inner mesh layer, *LAYER1*. Elements in this layer are uniformly-sized with edge lengths equal to the specified element size for the line (either through *SIZE* or line-length/*NDIV*). A positive value for *LAYER1* is interpreted as an absolute length, while a negative value is interpreted as a multiplier on the specified element size for the line. As a general rule, the resulting thickness of the inner mesh layer should be greater than or equal to the specified element size for the line. If *LAYER1* = OFF, layer-meshing control settings are cleared for the selected lines. The default value is 0.0

***LAYER2***

Layer-meshing control parameter. Distance which defines the thickness of the outer mesh layer, *LAYER2*. Elements in this layer transition in size from those in *LAYER1* to the global element size. A positive value of *LAYER2* is interpreted as an absolute length, while a negative value is interpreted as a mesh transition factor. A value of *LAYER2* = -2 would indicate that elements should approximately double in size as the mesh progresses normal to *LAYER1*. The default value is 0.0.

***KYNDIV***

0, No, and Off means that SmartSizing cannot override specified divisions and spacing ratios. Mapped mesh fails if divisions do not match. This defines the specification as "hard".

1, Yes, and On means that SmartSizing can override specified divisions and spacing ratios for curvature or proximity. Mapped meshing can override divisions to obtain required matching divisions. This defines the specification as "soft".

**Notes**

Defines the number of divisions and the spacing ratio on selected lines. Lines with previously specified divisions may also be changed.

This command is also valid for [rezoning](#).

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Clr Layers**  
**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Picked Lines**  
**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>All Lines**  
**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Clr Size**  
**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Copy Divs**  
**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Flip Bias**  
**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Picked Lines**

**LEXTND**, *NL1*, *NK1*, *DIST*, *KEEP*

**Extends a line at one end by using its slope.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NL1**

Number of the line to be extended. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**NK1**

Number of keypoint at the end of line *NL1* to be extended.

**DIST**

Distance that the line will be extended.

**KEEP**

Specifies whether to keep the input entities:

**0**

Modify old line to use new keypoints and slopes.

**1**

Do not modify old line. New line will overlay old line and have unique keypoints.

## Notes

Extends a line at one end by using its slope. Lines may be redefined only if not yet attached to an area. Line divisions are set to zero (use [LESIZE](#), etc. to modify). Note that solid modeling in a toroidal coordinate system is not recommended.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Extend Line**

---

**LFILLT**, *NL1*, *NL2*, *RAD*, *PCENT*

**Generates a fillet line between two intersecting lines.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NL1**

Number of the first intersecting line. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**NL2**

Number of the second intersecting line.

**RAD**

Radius of fillet to be generated. Radius should be less than the lengths of the two lines specified with *NL1* and *NL2*.

***PCENT***

Number to be assigned to generated keypoint at fillet arc center. If zero (or blank), no keypoint is generated.

**Notes**

Generates a fillet line between two intersecting lines *NL1* (*P1-PINT*) and *NL2* (*P2-PINT*). Three keypoints may be generated, two at the fillet tangent points (*PTAN1* and *PTAN2*) and one (optional) at the fillet arc center (*PCENT*). Line *P1-PINT* becomes *P1-PTAN1*, *P2-PINT* becomes *P2-PTAN2*, and new arc line *PTAN1-PTAN2* is generated. Generated keypoint and line numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]). Line divisions are set to zero (use **LESIZE**, etc. to modify).

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Lines>Line Fillet**

**LFSURF, *SLINE*, *TLINE***

**Generates surface elements overlaid on the edge of existing solid elements and assigns the extra node as the closest fluid element node.**

[PREP7: Elements](#)

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

***SLINE***

Component name for the surface lines of the meshed solid areas.

***TLINE***

Component name for the target lines meshed with fluid elements.

**Notes**

This command macro is used to generate surface effect elements overlaid on the surface of existing plane elements and, based on proximity, to determine and assign the extra node for each surface element. The underlying areas of the solid region and the fluid lines must be meshed prior to calling this command macro. The active element type must be **SURF151** with appropriate settings for **KEYOPT(4)**, **KEYOPT(5)**, **KEYOPT(6)**, and **KEYOPT(8)**.

The surface lines of the solid and the target lines of the fluid are grouped into components and named using the **CM** command. The names must be enclosed in single quotes (e.g., '*SLINE*') when the **LFSURF** command is manually typed in.

When using the GUI method, node and element components are created through the picking dialog boxes associated with this command.

The macro is applicable for the **SURF151** and **FLUID116** element types.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Area to Fluid**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Line to Fluid**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Node to Fluid**

**LGEN**, *ITIME*, *NL1*, *NL2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

**Generates additional lines from a pattern of lines.**

PREP7:Lines

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ***ITIME***

Do this generation operation a total of *ITIME*s, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be > 1 for generation to occur.

#### ***NL1*, *NL2*, *NINC***

Generate lines from pattern beginning with *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and pattern is all selected lines [LSEL]. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

#### ***DX*, *DY*, *DZ***

Keypoint location increments in the active coordinate system (--, D $\theta$ , DZ for cylindrical, --, D $\theta$ , -- for spherical).

#### ***KINC***

Keypoint increment between generated sets. If zero, the lowest available keypoint numbers are assigned [NUMSTR].

#### ***NOELEM***

Specifies if elements and nodes are also to be generated:

- 0**  
Generate nodes and elements associated with the original lines, if they exist.
- 1**  
Do not generate nodes and elements.

#### ***IMOVE***

Specifies whether to redefine the existing lines:

- 0**  
Generate additional lines as requested with the *ITIME* argument.
- 1**  
Move original lines to new position retaining the same keypoint numbers (*ITIME*, *KINC*, and *NOELM* are ignored). Valid only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

## **Notes**

Generates additional lines (and their corresponding keypoints and mesh) from a given line pattern. The MAT, TYPE, REAL, and ESYS attributes are based upon the lines in the pattern and not upon the current settings. End slopes of the generated lines remain the same (in the active coordinate system) as those of the given pattern. For example, radial slopes remain radial, etc. Generations which produce lines of a size or shape

different from the pattern (i.e., radial generations in cylindrical systems, radial and phi generations in spherical systems, and theta generations in elliptical systems) are not allowed. Note that solid modeling in a toroidal coordinate system is not recommended. New line numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]).

## Menu Paths

**Main Menu>Preprocessor>Modeling>Copy>Lines**

**Main Menu>Preprocessor>Modeling>Move / Modify>Lines**

**LGLUE**, *NL1, NL2, NL3, NL4, NL5, NL6, NL7, NL8, NL9*

**Generates new lines by "gluing" lines.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NL1, NL2, NL3, . . . , NL9*

Numbers of the lines to be glued. If *NL1* = ALL, all selected lines will be glued (*NL2* to *NL9* will be ignored). If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

## Notes

Use of the **LGLUE** command generates new lines by "gluing" input lines. The glue operation redefines the input lines so that they share keypoints at their common ends. The new lines encompass the same geometry as the original lines. This operation is only valid if the intersections of the input lines are keypoints at the ends of those lines. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

The **LGLUE** command results in the merging of keypoints at the common end of the lines. The keypoints of the lower numbered line will be kept. This means one must be aware of line numbering when multiple **LGLUE** commands are applied to avoid any "ungluing" of geometry.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Glue>Lines**

**LGWRITE**, *Fname, Ext, --, Kedit*

**Writes the database command log to a file.**

[SESSION: Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `LGW` if `Fname` and `Ext` are blank.

--

Unused field.

***Kedit***

Flag to suppress nonessential commands:

**NONE**

Do not suppress any commands (default).

**COMMENT**

Write nonessential commands as comments (starting with !).

**REMOVE**

Do not write nonessential commands or comments.

## Notes

Writes the database command log to a named file. The database command log contains all commands that were used to create the current database. These commands are recorded in the database as they are issued, and saved in the database file (`File.DB`) whenever the database is saved. The **LGWRITE** command extracts these commands from the database and writes them to a file. Nonessential commands (for listing, graphics displays, help, etc.) can be excluded from the file by using the *Kedit* field. The file resulting from **LGWRITE** can be used as command input to the program. This command is most useful if the session log file (`File.LOG`), which is normally saved during an interactive session, has been lost or corrupted. **LGWRITE** cannot be used after design optimization looping because the database is cleared [**CLEAR**] automatically at each loop; use `File.LOG` instead.

This command is valid in any processor.

## Menu Paths

Utility Menu>File>Write DB Log File

---

**/LIGHT**, *WN*, *NUM*, *INT*, *XV*, *YV*, *ZV*, *REFL*

**Specifies the light direction for the display window.**

GRAPHICS:Style

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***WN***

Window number (or ALL) to which command applies (defaults to 1).

***NUM***

Ambient or directional light key:

**0**

Ambient light (default).



**1**

Directional light.

**INT**

Light intensity factor (defaults to 0.3 for ambient, 1.0 for directional). This option is valid only for 3-D devices).

**XV, YV, ZV**

Light direction (valid only for *NUM* = 1). The directional light source is parallel to the line from point *XV*, *YV*, *ZV* to the origin, in the global Cartesian system origin. Defaults to the viewing direction [**/VIEW**].

**REFL**

Light reflectance factor (valid only for *NUM* = 1 and 3-D devices).

**Command Default**

Use ambient light.

**Notes**

Defines the light direction for the window. Use this command only with 3-D graphics devices or 2-D devices when Z-buffering is used [**/TYPE**,,(6 or 7)]. The ambient light has no direction, only an intensity. You can position the directional light source by defining a point (in the global Cartesian coordinate system) representing a point along the light directional line. This point, and the global Cartesian coordinate system origin, define the line along which the light is positioned looking toward the origin. You can use any point along the light line; for example, both (1.,1.,1.) and (2.,2.,2.) give the same light effect. For 3-D graphics devices only, the directional light source also has intensity and reflectance factors.

By choosing the highest intensity ambient light for 3-D graphics devices (via the command **/LIGHT,WN,0,1**), you can nullify color shading and other effects of directional lighting.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>PlotCtrls>Style>Light Source**

**LINA, NL, NA**

**Finds the intersection of a line with an area.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NL**

Number of line to be intersected. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**NA**

Number of area to be intersected.

## Notes

Finds the intersection of a line with an area. New lines will be generated where the lines intersect the areas. If the regions of intersection are only points, new keypoints will be generated instead. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Line with Area**

---

## LINE

Specifies "Lines" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**/LINE**, *X1*, *Y1*, *X2*, *Y2*

Creates annotation lines (GUI).

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**X1**

Line X starting location (-1.0 < X < 2.0).

**Y1**

Line Y starting location (-1.0 < Y < 1.0).

**X2**

Line X ending location (-1.0 < X < 2.0).

**Y2**

Line Y ending location (-1.0 < Y < 1.0).

## Notes

Defines annotation lines to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All lines are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** command to set the attributes of the line.

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Annotation>Create Annotation**

---

### **LINES, *N***

**Specifies the length of a printed page.**

[POST26: Listing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*N*

Number of lines per page (defaults to 20). (Minimum allowed = 11).

## Command Default

20 lines per page.

## Notes

Specifies the length of a printed page (for use in reports, etc.).

## Menu Paths

**Main Menu>TimeHist Postpro>Settings>List**

---

### **LINL, *NL1, NL2, NL3, NL4, NL5, NL6, NL7, NL8, NL9***

**Finds the common intersection of lines.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NL1, NL2, NL3, . . . , NL9*

Numbers of lines to be intersected. If *NL1* = ALL, find the intersection of all selected lines and *NL2* to *NL9* are ignored. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

## Notes

Finds the common (not pairwise) intersection of lines. The common intersection is defined as the regions shared (in common) by *all* lines listed on this command. New lines will be generated where the original lines intersect. If the regions of intersection are only points, new keypoints will be generated instead. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Common>Lines**

---

**LINP**, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

**Finds the pairwise intersection of lines.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NL1*, *NL2*, *NL3*, . . . , *NL9*

Numbers of lines to be intersected pairwise. If *NL1* = ALL, find the pairwise intersection of all selected lines and *NL2* to *NL9* are ignored. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NL1*.

## Notes

Finds the pairwise intersection of lines. The pairwise intersection is defined as any and all regions shared by at least two lines listed on this command. New lines will be generated where the original lines intersect pairwise. If the regions of pairwise intersection are only points, new keypoints will be generated. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Pairwise>Lines**

---

**LINV**, *NL*, *NV*

**Finds the intersection of a line with a volume.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NL*

Number of line to be intersected. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

*NV*

Number of volume to be intersected.

## Notes

Finds the intersection of a line with a volume. New lines will be generated where the lines intersect the volumes. If the regions of intersection are only points, new keypoints will be generated instead. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Line with Volume**

## LIST, LEVEL

**Lists out the sets in the results file.**

[AUX3: Results Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This command lists the results set number, the load step, substep, and time step for each set. It also shows all sets marked for deletion.

## Menu Paths

**This command cannot be accessed from a menu.**

## \*LIST, Fname, Ext, --

**Displays the contents of an external, coded file.**

[SESSION: Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

### ***Ext***

Filename extension (8 character maximum).

### **--**

Unused field.

## Notes

Displays the contents of an external, coded file. The file to be listed cannot be in use (open) at the time (except for the error file, `File.ERR`, which may be displayed with **\*LIST,ERR**).

Use caution when you are listing active ANSYS files via the **List> Files> Other** and **File> List> Other** menu paths. File I/O buffer and system configurations can result in incomplete listings unless the files are closed.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

### LLIST, *NL1*, *NL2*, *NINC*, *Lab*

**Lists the defined lines.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *NL1*, *NL2*, *NINC*

List lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL (default), *NL2* and *NINC* are ignored and all selected lines [**LSEL**] are listed. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

#### *Lab*

Determines what type of listing is used (one of the following):

##### **(blank)**

Prints information about all lines in the specified range.

##### **RADIUS**

Prints the radius of certain circular arcs, along with the keypoint numbers of each line. Straight lines, non-circular curves, and circular arcs not internally identified as arcs (which depends upon how each arc is created) will print a radius value of zero.

##### **LAYER**

Prints layer-mesh control specifications.

##### **HPT**

Prints information about only those lines that contain hard points. HPT is not supported in the GUI.

##### **ORIENT**

Prints a list of lines, and identifies any orientation keypoints and any cross section IDs that are associated with the lines. Used for beam meshing with defined orientation nodes and cross sections.

## Notes

There are 2 listings for the number of element divisions and the spacing ratio. The first listing shows assignments from **LESIZE** only, followed by the "hard" key (*KYNDIV*). See **LESIZE** for more information. The second listing shows *NDIV* and *SPACE* for the existing mesh, if one exists. Whether this existing mesh and the mesh generated by **LESIZE** match at any given point depends upon meshing options and the sequence of meshing operations.

A "-1" in the "nodes" column indicates that the line has been meshed but that there are no interior nodes.

An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **LATT** command (and will not be reset to zero if the mesh is cleared);

one listed as a negative value indicates that the attribute was assigned using the attribute pointer [**TYPE**, **MAT**, **REAL**, or **ESYS**] that was active during meshing (and will be reset to zero if the mesh is cleared).

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Lines

---

**LMATRIX**, *SYMFAC*, *Coilname*, *Curname*, *Indname*

**Calculates an inductance matrix and the total flux linkage for an N-winding coil system.**

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

### ***SYMFAC***

Geometric symmetry factor. Inductance terms are scaled by this factor which represents the fraction of the total device modeled. Default is 1.

### ***Coilname***

Alphanumeric prefix identifier for coil label used in defining named element coil components. Default is 'coil.'

### ***Curname***

Name of a predefined parameter array containing the nominal coil currents of the system. The array must be defined (see **\*DIM** command) prior to calling the **LMATRIX** macro. Default is 'cur.'

### ***Indname***

Name of the array parameter to be created by **LMATRIX** containing the calculated inductance matrix and the flux linkage in each coil. A text file of the same name with an extension .TXT is created containing the matrix data. Default is 'lmatrix.'

## Notes

**LMATRIX** calculates the differential inductance matrix for an  $N$ -winding system where  $N$  is the number of coils in the system, and calculates the total flux linkage in each coil. **LMATRIX** may only be executed after the solution of a problem with nominal currents applied to the coils at a desired "operating point." The array *Indname* has  $N$  rows and  $N+1$  columns. The  $N \times N$  block is the differential inductance matrix; the  $N+1$ th column contains the total flux linkage, with the  $i$ th row corresponding to the  $i$ th coil. See the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for more details.

To invoke the **LMATRIX** macro, for the classical formulations, the elements for each coil must be grouped into a component using the **CM** command. Each set of independent coil elements is assigned a component name with the prefix *Coilname* followed by the coil number. For the solenoidal formulations, you must make the exciting node with a **F,AMPS** load a node component using the **CM** command. The classical and solenoidal formulations cannot be mixed.

To invoke the **LMATRIX** macro, the vector array parameter *Curname* with dimension  $N$  must be defined and named using the **\*DIM** command. You must set each vector array entry equal to the nominal current per turn in the corresponding coil at the operating point. Zero current values must be approximated by a negligibly small applied current.

Do not apply (or remove) inhomogeneous loads before using the **LMATRIX** command. Inhomogeneous loads are those created by:

- Degree of freedom commands (**D**, **DA**, etc.) specifying nonzero degrees of freedom values on nodes or solid model entities
- Any **CE** command with a nonzero constant term

Do not put any loads (for example, current) on elements not contained in the element component.

Operating solutions must be obtained through static analysis before calling **LMATRIX**. All name-strings must be enclosed in single quotes in the **LMATRIX** command line. The geometric symmetry factor, *SymFac*, represents the fraction of the device modeled, disregarding any current source primitives.

**LMATRIX** works only with magnetic field elements: **PLANE53**, **SOLID96**, and **SOLID97**, and with **SOLID117/SOURC36** solenoidal formulations. You can use interface element **INTER115** to connect **SOLID96** and **SOLID97** regions.

For more information, see **LMATRIX** in the *Low-Frequency Electromagnetic Analysis Guide*.

See the *Theory Reference for the Mechanical APDL and Mechanical Applications* and *Electric and Magnetic Macros* in the *Low-Frequency Electromagnetic Analysis Guide* for details.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Solve>Electromagnet>Static Analysis>Induct Matrix**

---

## **LMESH**, *NL1*, *NL2*, *NINC*

**Generates nodes and line elements along lines.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NL1*, *NL2*, *NINC*

Mesh lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and all selected lines [**LSEL**] are meshed. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

## Notes

Generates nodes and line elements along lines. Missing nodes required for the generated elements are created and assigned the lowest available numbers.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh>Lines**

---



**LNSRCH**, *Key*

**Activates a line search to be used with Newton-Raphson.**

**SOLUTION: Nonlinear Options**

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

**Key**

Line search key:

**OFF**

Do not use a line search.

**ON**

Use a line search. Note, adaptive descent is suppressed when **LNSRCH** is on unless explicitly requested on the **NROPT** command. *Having line search on and adaptive descent on at the same time is not recommended.*

**AUTO**

ANSYS automatically switches line searching ON and OFF between substeps of a load step as needed. This option is recommended.

**Command Default**

Line search off, unless contact elements are present.

**Notes**

The default values given for this command assume **SOLCONTROL,ON** (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL,ON** and **SOLCONTROL,OFF**.

Activates a line search to be used with the Newton-Raphson method [**NROPT**]. Line search is an alternative to adaptive descent (see [Line Search](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*).

**LNSRCH,AUTO** can be very efficient for problems in which **LNSRCH** is needed at only certain substeps.

You cannot use line search [**LNSRCH**], automatic time stepping [**AUTOTS**], or the DOF solution predictor [**PRED**] with the arc-length method [**ARCLEN, ARCTRM**]. If you activate the arc-length method after you set **LNSRCH, AUTOTS**, or **PRED**, a warning message appears. If you choose to proceed with the arc-length method activation, ANSYS disables your line search, automatic time stepping, and DOF predictor settings.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Line Search**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear**

**Main Menu>Solution>Load Step Opts>Nonlinear>Line Search**

**LOCAL**, *KCN*, *KCS*, *XC*, *YC*, *ZC*, *THXY*, *THYZ*, *THZX*, *PAR1*, *PAR2*

**Defines a local coordinate system by a location and orientation.**

DATABASE: Coordinate System

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KCN**

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

**KCS**

Coordinate system type:

**0 or CART**

Cartesian

**1 or CYLIN**

Cylindrical (circular or elliptical)

**2 or SPHE**

Spherical (or spheroidal)

**3 or TORO**

Toroidal

**XC, YC, ZC**

Location (in the global Cartesian coordinate system) of the origin of the new coordinate system.

**THXY**

First rotation about local Z (positive X toward Y).

**THYZ**

Second rotation about local X (positive Y toward Z).

**THZX**

Third rotation about local Y (positive Z toward X).

**PAR1**

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

**PAR2**

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

## Notes

Defines a local coordinate system by origin location and orientation angles. The local coordinate system is parallel to the global Cartesian system unless rotated. Rotation angles are in degrees and redefine any previous rotation angles. See the **CLOCAL**, **CS**, **CSWPLA**, and **CSKP** commands for alternate definitions. This local system becomes the active coordinate system [**CSYS**]. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

## Menu Paths

Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>At Specified Loc

---

**LOVLAP**, *NL1, NL2, NL3, NL4, NL5, NL6, NL7, NL8, NL9*

Overlaps lines.

[PREP7:Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NL1, NL2, NL3, . . . , NL9*

Numbers of lines to be overlapped. If *NL1* = ALL, *NL2* to *NL9* are ignored and all selected lines are overlapped. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

## Notes

Overlaps lines. Generates new lines which encompass the geometry of all the input lines. The new lines are defined by the regions of intersection of the input lines, and by the complementary (non-intersecting) regions. See the [Modeling and Meshing Guide](#) for an illustration. This operation is only valid when the region of intersection is a line. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Overlap>Lines

---

**LPLOT**, *NL1, NL2, NINC*

Displays the selected lines.

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NL1, NL2, NINC*

Display lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL (default), *NL2* and *NINC* are ignored and display all selected lines [**LSEL**].

## Notes

Mesh divisions on plotted lines are controlled by the *LDIV* option of the **/PSYMB** command.

This command is valid in any processor.

## Menu Paths

Utility Menu>Plot>Lines  
Utility Menu>Plot>Specified Entities>Lines

---

**LPRT**, *PORTNUM*, *LNUM*, *Ltype*, *CS*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

**Defines impedance and calibration lines for excitation eigenfield.**

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

***PORTNUM***

Port number. If *PORTNUM* = LIST, lists lines. If *PORTNUM* = DELE, deletes lines.

***LNUM***

Line number. A maximum of 20 lines can be defined for each port. The line number should be between 1 and 1000.

***Ltype***

Type of line:

**IMPD**

Impedance line.

**CIMP**

Impedance and calibration line.

***CS***

Local coordinate system number at port.

***X1*, *Y1*, *Z1***

The local coordinates of the starting point.

***X2*, *Y2*, *Z2***

The local coordinates of the ending point.

## Notes

The direction of a line is from starting point to ending point.

## Menu Paths

**This command cannot be accessed from a menu.**

**LPTN**, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

**Partitions lines.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NL1*, *NL2*, *NL3*, . . . , *NL9***

Numbers of lines to be operated on. If *NL1* = ALL, *NL2* to *NL9* are ignored all selected lines are used.

If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NL1*.

## Notes

Partitions lines. Generates new lines which encompass the geometry of all the input lines. The new lines are defined by both the regions of intersection of the input lines and the complementary (non-intersecting) regions. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Partition>Lines**

**LREFINE**, *NL1*, *NL2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

**Refines the mesh around specified lines.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NL1*, *NL2*, *NINC***

Lines (*NL1* to *NL2* in increments of *NINC*) around which the mesh is to be refined. *NL2* defaults to *NL1*, and *NINC* defaults to 1. If *NL1* = ALL, *NL2* and *NINC* are ignored and all selected lines are used for refinement. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

### ***LEVEL***

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

### ***DEPTH***

Depth of mesh refinement in terms of the number of elements outward from the indicated lines (defaults to 1).

### ***POST***

Type of postprocessing to be done after element splitting, in order to improve element quality:

#### **OFF**

No postprocessing will be done.

#### **SMOOTH**

Smoothing will be done. Node locations may change.

#### **CLEAN**

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

### ***RETAIN***

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

#### **ON**

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

**OFF**

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

**Notes**

**LREFINE** performs local mesh refinement around the specified lines. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

**LREFINE** refines all area elements and tetrahedral volume elements that are adjacent to the specified lines. Any volume elements that are adjacent to the specified lines, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. For additional restrictions on mesh refinement, see [Revising Your Model](#) in the *Modeling and Meshing Guide*.

This command is also valid for [rezoning](#).

**Menu Paths**

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Lines

---

**LREVERSE**, *LNUM*, *NOEFLIP*

Reverses the normal of a line, regardless of its connectivity or mesh status.

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***LNUM***

Line number of the line whose normal direction is to be reversed. If *LNUM* = ALL, the normals of all selected lines will be reversed. If *LNUM* = P, graphical picking is enabled. A component name may also be substituted for *LNUM*.

***NOEFLIP***

Indicates whether you want to change the normal direction of the existing elements on the reversed line(s) so that they are consistent with each line's new normal direction.

**0**

Make the normal direction of existing elements on the reversed line(s) consistent with each line's new normal direction (default).

**1**

Do not change the normal direction of existing elements on the reversed line(s).

**Notes**

You cannot use the **LREVERSE** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

For more information, see [Revising Your Model](#) in the *Modeling and Meshing Guide*.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Reverse Normals>of Lines**

---

**LROTAT**, *NK1, NK2, NK3, NK4, NK5, NK6, PAX1, PAX2, ARC, NSEG*

**Generates circular lines by rotating a keypoint pattern about an axis.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NK1, NK2, NK3, . . . , NK6***

List of keypoints in the pattern to be rotated (6 maximum if using keyboard entry). If *NK1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NK1* = ALL, all selected keypoints (except *PAX1* and *PAX2*) will define the pattern to be rotated. A component name may also be substituted for *NK1*.

***PAX1, PAX2***

Keypoints defining the axis about which the keypoint pattern is to be rotated.

***ARC***

Arc length (in degrees). Positive follows right-hand rule about *PAX1-PAX2* vector. Defaults to 360.

***NSEG***

Number of lines (8 maximum) around circumference. Defaults to minimum required for 90° (maximum) arcs, i.e., 4 for 360°, 3 for 270°, etc.

## Notes

Generates circular lines (and their corresponding keypoints) by rotating a keypoint pattern about an axis. Keypoint patterns are generated at regular angular locations (based on a maximum spacing of 90°). Line patterns are generated at the keypoint patterns. Keypoint and line numbers are automatically assigned (beginning with the lowest available values [\[NUMSTR\]](#)).

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Extrude>Keypoints>About Axis**

---

**LSBA**, *NL*, *NA*, *SEPO*, *KEEPL*, *KEEPA***Subtracts areas from lines.**

PREP7: Booleans

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**NL**

Line (or lines, if picking is used) to be subtracted from. If ALL, use all selected lines. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL*.

**NA**

Area (or areas, if picking is used) to be subtracted. If ALL, use all selected areas. A component name may also be substituted for *NA*.

**SEPO**

Behavior if the intersection of the lines and the areas is a keypoint or keypoints:

**(blank)**

The resulting lines will share keypoint(s) where they touch.

**SEPO**

The resulting lines will have separate, but coincident keypoint(s) where they touch.

**KEEPL**

Specifies whether *NL* lines are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NL* lines after **LSBA** operation (override **BOPTN** command settings).

**KEEP**

Keep *NL* lines after **LSBA** operation (override **BOPTN** command settings).

**KEEPA**

Specifies whether *NA* areas are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete areas after **LSBA** operation (override **BOPTN** command settings).

**KEEP**

Keep areas after **LSBA** operation (override **BOPTN** command settings).

**Notes**

Generates new lines by subtracting the regions common to both *NL* lines and *NA* areas (the intersection) from the *NL* lines. The intersection can be a line(s) or keypoint(s). If the intersection is a keypoint and *SEPO* is blank, the *NL* line is divided at the keypoint and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL* is divided into two unconnected lines with separate keypoints where they touch. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.



## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by Area

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by Area

**\*LSBAC**, *EngineName*, *RhsVector*, *SolVector*

**Performs the solve (forward/backward substitution) of a factorized linear system.**

APDL: Matrix Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *EngineName*

Name used to identify this engine. Must have been previously created using **\*LSENGINE** and factorized using **\*LSFACTOR**.

### *RhsVector*

Name of vector containing the right-hand side (load) vectors as input. Must have been previously defined as a **\*VEC** vector or a **\*DMAT** matrix.

### *SolVector*

Name of vector that will contain the solution vectors upon completion. Must be predefined as a **\*VEC** vector or **\*DMAT** matrix.

## Notes

This command performs forward and back substitution to obtain the solution to the linear matrix equation  $Ax = b$ . The matrix engine must have been previously defined using **\*LSENGINE**, and the matrix factored using **\*LSFACTOR**.

You can use the **\*DMAT,,,COPY** (or **\*VEC,,,COPY**) command to copy the load vector to the solution vector in order to predefine it with the appropriate size.

## Menu Paths

**This command cannot be accessed from a menu.**

**LSBL**, *NL1*, *NL2*, *SEPO*, *KEEP1*, *KEEP2*

**Subtracts lines from lines.**

PREP7: Booleans

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NL1*

Line (or lines, if picking is used) to be subtracted from. If ALL, use all selected lines. Lines specified in this argument are not available for use in the *NL2* argument. If P, graphical picking is enabled (valid only in the GUI) and all remaining fields are ignored. A component name may also be substituted for *NL1*.

**NL2**

Line (or lines, if picking is used) to subtract. If ALL, use all selected lines (except those included in the *NL1* argument). A component name may also be substituted for *NL2*.

**SEPO**

Behavior if the intersection of the *NL1* lines and the *NL2* lines is a keypoint or keypoints:

**(blank)**

The resulting lines will share keypoint(s) where they touch.

**SEPO**

The resulting lines will have separate, but coincident keypoint(s) where they touch.

**KEEP1**

Specifies whether *NL1* lines are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NL1* lines after **LSBL** operation (override **BOPTN** command settings).

**KEEP**

Keep *NL1* lines after **LSBL** operation (override **BOPTN** command settings).

**KEEP2**

Specifies whether *NL2* lines are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NL2* lines after **LSBL** operation (override **BOPTN** command settings).

**KEEP**

Keep *NL2* lines after **LSBL** operation (override **BOPTN** command settings).

## Notes

Generates new lines by subtracting the regions common to both *NL1* and *NL2* lines (the intersection) from the *NL1* lines. The intersection can be a line(s) or point(s). If the intersection is a point and *SEPO* is blank, the *NL1* line is divided at the point and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL1* is divided into two unconnected lines with separate keypoints where they touch. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **LSBL,ALL,ALL** will have no effect since all the lines (in *NL1*) will be unavailable as *NL2* lines.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by Line**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by Line**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Lines**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Lines**

**LSBV**, *NL*, *NV*, *SEPO*, *KEEPL*, *KEEPV***Subtracts volumes from lines.**

PREP7: Booleans

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***NL***

Line (or lines, if picking is used) to be subtracted from. If ALL, use all selected lines. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL*.

***NV***

Volume (or volumes, if picking is used) to be subtracted. If ALL, use all selected volumes. A component name may also be substituted for *NV*.

***SEPO***

Behavior if the intersection of the *NL* lines and the *NV* volumes is a keypoint or keypoints:

**(blank)**

The resulting lines will share keypoint(s) where they touch.

**SEPO**

The resulting lines will have separate, but coincident keypoint(s) where they touch.

***KEEPL***

Specifies whether *NL* lines are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NL* lines after **LSBV** operation (override **BOPTN** command settings).

**KEEP**

Keep *NL* lines after **LSBV** operation (override **BOPTN** command settings).

***KEEPV***

Specifies whether *NV* volumes are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NV* volumes after **LSBV** operation (override **BOPTN** command settings).

**KEEP**

Keep *NV* volumes after **LSBV** operation (override **BOPTN** command settings).

**Notes**

Generates new lines by subtracting the regions common to both *NL* lines and *NV* volumes (the intersection) from the *NL* lines. The intersection can be a line(s) or point(s). If the intersection is a point and *SEPO* is blank, the *NL1* line is divided at the point and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL1* is divided into two unconnected lines with separate keypoints where they touch. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **LSBL**, ALL, ALL will have no effect since all the lines (in *NL1*) will be unavailable as *NL2* lines.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by Volume

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by Volume

Main Menu>Preprocessor>Modeling>Operate>Divide>Line by Volu

**LSBW**, *NL*, *SEPO*, *KEEP*

Subtracts the intersection of the working plane from lines (divides lines).

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NL*

Line (or lines, if picking is used) to be subtracted from. If *NL* = ALL, use all selected lines. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be input for *NL*.

### *SEPO*

Behavior of the created boundary.

#### (blank)

The resulting lines will share keypoint(s) where they touch.

#### SEPO

The resulting lines will have separate, but coincident keypoint(s).

### *KEEP*

Specifies whether *NL* lines are to be deleted:

#### (blank)

Use the setting of KEEP on the **BOPTN** command.

#### DELETE

Delete *NL* lines after **LSBW** operation (override **BOPTN** command settings).

#### KEEP

Keep *NL* lines after **LSBW** operation (override **BOPTN** command settings).

## Notes

Generates new lines by subtracting the intersection of the working plane from the *NL* lines. The intersection will be a keypoint(s). The working plane must not be in the same plane as the *NL* line(s). If *SEPO* is blank, the *NL* line is divided and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL* is divided into two unconnected lines with separate keypoints. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. Areas that completely contain the input lines will be updated if the lines are divided by this operation.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by WrkPlane

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by WrkPlane

**Main Menu>Preprocessor>Modeling>Operate>Divide>Line by WrkPlane**

---

## LSCLEAR, Lab

**Clears loads and load step options from the database.**

[SOLUTION: Load Step Operations](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### Lab

Label identifying the data to be cleared:

#### **SOLID**

Delete only solid model loads.

#### **FE**

Delete only finite element loads.

#### **INER**

Delete only inertia loads (**ACEL**, etc.).

#### **LFACT**

Initialize only load factors (on **DCUM**, **FCUM**, **SFCUM**, etc.).

#### **LSOPT**

Initialize only load step options.

#### **ALL**

Delete all loads and initialize all load step options and load factors.

## Notes

Loads are deleted, and load step options are initialized to their default values.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All F.E. Loads**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Inertia Lds**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Loads & Opts**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All SolidMod Lds**

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Reset Factors**

**Main Menu>Preprocessor>Loads>Load Step Opts>Reset Options**

**Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>All**

**Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Areas**

**Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Elements**

**Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Keypoints**

**Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Lines**

**Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Nodes**

**Main Menu>Solution>Define Loads>Delete>All Load Data>All F.E. Loads**

**Main Menu>Solution>Define Loads>Delete>All Load Data>All Inertia Lds**

**Main Menu>Solution>Define Loads>Delete>All Load Data>All Loads & Opts**

**Main Menu>Solution>Define Loads>Delete>All Load Data>All SolidMod Lds**

**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Reset Factors**  
**Main Menu>Solution>Load Step Opts>Reset Options**

---

## LSDELE, *LSMIN*, *LSMAX*, *LSINC*

**Deletes load step files.**

[SOLUTION: Load Step Operations](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *LSMIN*, *LSMAX*, *LSINC*

Range of load step files to be deleted, from *LSMIN* to *LSMAX* in steps of *LSINC*. *LSMAX* defaults to *LSMIN*, and *LSINC* defaults to 1. If *LSMIN* = ALL, all load step files are deleted (and *LSMAX* and *LSINC* are ignored). The load step files are assumed to be named Jobname.S*n*, where *n* is a number assigned by the **LSWRITE** command (01--09,10,11, etc.). On systems with a 3-character limit on the extension, the "S" is dropped for numbers > 99.

## Notes

Deletes load step files in the current directory (written by the **LSWRITE** command).

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Operate>Delete LS Files**  
**Main Menu>Solution>Define Loads>Operate>Delete LS Files**

---

## LSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP*

**Selects a subset of lines.**

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of select:

**S**

Select a new set (default).

**R**

Reselect a set from the current set.

**A**

Additionally select a set and extend the current set.

**U**

Unselect a set from the current set.

**ALL**

Restore the full set.

**NONE**

Unselect the full set.

**INVE**

Invert the current set (selected becomes unselected and vice versa).

**STAT**

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

**Item**

Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to LINE.

**Comp**

Component of the item (if required). Valid component labels are shown in the table below.

**VMIN**

Minimum value of item range. Ranges are line numbers, coordinate values, attribute numbers, etc., as appropriate for the item. If *VMIN* = 0.0, a tolerance of  $\pm 1.0E-6$  is used, or  $\pm 0.005 \times VMIN$  if *VMIN* = *VMAX*. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, ESYS, or NDIV and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **LLIST** command for a discussion of signed attributes.

**VMAX**

Maximum value of item range. *VMAX* defaults to *VMIN*.

**VINC**

Value increment within range. Used only with integer ranges (such as for line numbers). Defaults to 1. *VINC* cannot be negative.

**KSWP**

Specifies whether only lines are to be selected:

**0**

Select lines only.

**1**

Select lines, as well as keypoints, nodes, and elements associated with selected lines. Valid only with *Type* = S.

## Command Default

All lines are selected.

## Notes

Selects lines based on values of a labeled item and component. For example, to select a new set of lines based on line numbers 1 through 7, use **LSEL,S,LINE,,1,7**. The subset is used when the ALL label is entered (or implied) on other commands, such as **LLIST,ALL**. Only data identified by line number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

If *Item* = LCCA, the command selects only those lines that were created by concatenation. The *KSWP* field is processed, but the *Comp*, *VMIN*, *VMAX*, and *VINC* fields are ignored.

If *Item* = HPT, the command selects only those lines that contain hard points.

*Item* = RADIUS is only valid for lines that are circular arcs.

**LSEL** is valid in any processor.

For selections based on non-integer numbers (coordinates, results, etc.), items that are within the range  $VMIN - Toler$  and  $VMAX + Toler$  are selected. The default tolerance *Toler* is based on the relative values of *VMIN* and *VMAX* as follows:

- If  $VMIN = VMAX$ ,  $Toler = 0.005 \times VMIN$ .
- If  $VMIN = VMAX = 0.0$ ,  $Toler = 1.0E-6$ .
- If  $VMAX \neq VMIN$ ,  $Toler = 1.0E-8 \times (VMAX - VMIN)$ .

Use the **SELTOL** command to override this default and specify *Toler* explicitly.

**Table 224 LSEL - Valid Item and Component Labels**

**Valid Item and Component Labels LSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP***

Item	Comp	Description
LINE		Line number.
EXT		Line numbers on exterior of selected area (ignore remaining fields).
LOC	X,Y,Z	X, Y, or Z center location in the active coordinate system.
TAN1	X,Y,Z	Unit vector component of outward tangent at beginning of line.
TAN2	X,Y,Z	Unit vector component of outward tangent at end of line.
NDIV		Number of divisions within the line.
SPACE		Spacing ratio of line divisions.
MAT		Material number associated with the line.
TYPE		Element type number associated with the line.
REAL		Real constant set number associated with the line.
ESYS		Element coordinate system associated with the line.
SEC		Cross section ID number. [ <b>SECNUM</b> ]
LENGTH		Length of the line.
RADIUS		Radius of the line.
HPT		Line number (selects only lines with associated hard points).
LCCA		Concatenated lines (selects only lines that were created by concatenation [ <b>LCCAT</b> ]).

## Menu Paths

Utility Menu>Select>Entities



**\*LSENGINE**, *Type, EngineName, Matrix, Option*

**Creates a linear solver engine.**

APDL: Matrix Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Type*

Specifies the algorithm to be used:

#### **BCS --**

Boeing sparse solver (default if applied to sparse matrices).

#### **DSS --**

MKL sparse linear solver (Intel Windows and Linux systems only).

#### **LAPACK --**

LAPACK dense matrix linear solver (default if applied to dense matrices).

#### **DSP --**

Distributed sparse solver.

### *EngineName*

Name used to identify this engine. Must be specified.

### *Matrix*

Name of the matrix to solve.

### *Option*

Option to control the memory mode of the DSS solver (used only if *Type* = DSS):

#### **INCORE --**

In-core memory mode.

#### **OUTOFCORE --**

Out-of-core memory mode.

## Notes

This command creates a linear solver engine.

The BCS, DSS, and DSP solvers can only be used with sparse matrices. For dense matrices, use the LAPACK solver.

## Menu Paths

**This command cannot be accessed from a menu.**

**\*LSFACTOR**, *EngineName*, *Option*

**Performs the numerical factorization of a linear solver system.**

[APDL: Matrix Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *EngineName*

Name used to identify this engine. Must have been previously created using **\*LSENGINE**.

### *Option*

Option to invert the matrix, used only with an LAPACK engine (**\*LSENGINE,LAPACK**):

**INVERT --**

Invert the matrix.

## Notes

Performs the computationally intensive, memory intensive factorization of a matrix specified by **\*LSENGINE**, using the solver engine also specified by **\*LSENGINE**.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**LSLA**, *Type*

**Selects those lines contained in the selected areas.**

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of line select:

**S**

Select a new set (default).

**R**

Reselect a set from the current set.

**A**

Additionally select a set and extend the current set.

**U**

Unselect a set from the current set.

## Notes

This command is valid in any processor.

## Menu Paths

Utility Menu>Select>Entities

---

**LSLK**, *Type*, *LSKEY*

**Selects those lines containing the selected keypoints.**

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of line select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

### *LSKEY*

Specifies whether all contained line keypoints must be selected [**KSEL**]:

- 0**  
Select line if any of its keypoints are in the selected keypoint set.
- 1**  
Select line only if all of its keypoints are in the selected keypoint set.

## Notes

This command is valid in any processor.

## Menu Paths

Utility Menu>Select>Entities

---

## LSOPER

**Specifies "Load step operations" as the subsequent status topic.**

[SOLUTION: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This

command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>Solution>Load Step Operations**

---

**/LSPEC**, *LCOLOR*, *LINSTL*, *XLNWID*

**Specifies annotation line attributes (GUI).**

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *LCOLOR*

Line color:

- 0** Black
- 1** Red-Magenta
- 2** Magenta
- 3** Blue-Magenta
- 4** Blue
- 5** Cyan-Blue
- 6** Cyan
- 7** Green-Cyan
- 8** Green
- 9** Yellow-Green
- 10** Yellow
- 11** Orange
- 12** Red

- 13**  
Dark Gray
- 14**  
Light Gray
- 15**  
White

**LINSTL**

Line style:

- 0**  
Solid line.
- 1**  
Dashed line.

**XLNWID**

Line width multiplier (1.0 to 20.0). Defaults to 1.0.

**Notes**

Specifies annotation line attributes to control certain characteristics of the lines created via the **/LINE**, **/LARC**, **/LSYMBOL**, **/POLYGON**, **/PMORE**, **/PCIRCLE**, and **/PWEDGE** commands. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is valid in any processor.

**Menu Paths**

Utility Menu>PlotCtrls>Annotation>Create Annotation

**LSREAD, LSNUM**

**Reads load and load step option data into the database.**

[SOLUTION: Load Step Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**LSNUM**

Identification number of the load step file to be read. Defaults to 1 + highest number read in the current session. Issue **LSREAD,STAT** to list the current value of *LSNUM*. Issue **LSREAD,INIT** to reset *LSNUM* to 1. The load step files are assumed to be named `Jobname.Sn`, where *n* is a number assigned by the **LSWRITE** command (01--09,10,11, etc.). On systems with a 3-character limit on the extension, the "S" is dropped for *LSNUM* > 99.

**Notes**

Reads load and load step option data from the load step file into the database. **LSREAD** *will not* clear the database of *all* current loads. However, if a load is respecified with **LSREAD**, then it will overwrite the existing

load. See the **LSWRITE** command to write load step files, and the **LSDELE** command to delete load step files. **LSREAD** removes any existing **SFGRAD** specification.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Read LS File

Main Menu>Solution>Load Step Opts>Read LS File

## LSSCALE, NL1, NL2, NINC, RX, RY, RZ, KINC, NOELEM, IMOVE

Generates a scaled set of lines from a pattern of lines.

PREP7:Lines

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NL1, NL2, NINC

Set of lines (*NL1* to *NL2* in steps of *NINC*) that defines the pattern to be scaled. *NL2* defaults to *NL1*, *NINC* defaults to 1. If *NL1* = ALL, *NL2* and *NINC* are ignored and the pattern is defined by all selected lines. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

### RX, RY, RZ

Scale factors to be applied to the X, Y, Z keypoint coordinates in active coordinate system (*RR*, *R $\theta$* , *RZ* for cylindrical; *RR*, *R $\theta$* , *R $\Phi$*  for spherical). Note that the *R $\theta$*  and *R $\Phi$*  scale factors are interpreted as angular offsets. For example, for **CSYS**,1, *RR*, *R $\theta$* , *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

### KINC

Increment to be applied to keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned [**NUMSTR**].

### NOELEM

Specifies whether nodes and elements are also to be generated:

0

Nodes and line elements associated with the original lines will be generated (scaled) if they exist.

1

Nodes and line elements will *not* be generated.

### IMOVE

Specifies whether lines will be moved or newly defined:

0

Additional lines will be generated.

1

Original lines will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

## Notes

Generates a scaled set of lines (and their corresponding keypoints and mesh) from a pattern of lines. The MAT, TYPE, REAL, and ESYS attributes are based on the lines in the pattern and not the current settings. Scaling is done in the active coordinate system. Lines in the pattern could have been generated in any coordinate system.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Lines

## LSSOLVE, LSMIN, LSMAX, LSINC

Reads and solves multiple load steps.

[SOLUTION: Load Step Operations](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *LSMIN, LSMAX, LSINC*

Range of load step files to be read and solved, from *LSMIN* to *LSMAX* in steps of *LSINC*. *LSMAX* defaults to *LSMIN*, and *LSINC* defaults to 1. If *LSMIN* is blank, a brief command description is displayed. The load step files are assumed to be named *Jobname.Sn*, where *n* is a number assigned by the **LSWRITE** command (01--09,10,11, etc.). On systems with a 3-character limit on the extension, the "S" is dropped for numbers > 99.

## Notes

**LSSOLVE** invokes an ANSYS macro to read and solve multiple load steps. The macro loops through a series of load step files written by the **LSWRITE** command. The macro file called by **LSSOLVE** is called `LS-SOLVE.MAC`.

**LSSOLVE** cannot be used with the birth-death option.

**LSSOLVE** is not supported for cyclic symmetry analyses.

**LSSOLVE** does not support restarts.

**LSSOLVE** does not apply to FLOTRAN.

## Menu Paths

Main Menu>Solution>Solve>From LS Files

## LSTR, P1, P2

Defines a straight line irrespective of the active coordinate system.

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### P1

Keypoint at the beginning of line. If  $P1 = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI)

### P2

Keypoint at the end of line.

## Notes

Defines a straight line from  $P1$  to  $P2$  using the global Cartesian coordinate system. The active coordinate system will be ignored. The line shape is invariant with the coordinate system after it is generated. Lines may be redefined only if not yet attached to an area.

## Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Straight Line

---

## LSUM

Calculates and prints geometry statistics of the selected lines.

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Calculates and prints geometry statistics (length, centroid, moments of inertia, etc.) associated with the selected lines. Geometry items are reported in the global Cartesian coordinate system. A unit density is assumed, irrespective of any material associations [**LATT**, **MAT**]. Items calculated by **LSUM** and later retrieved by a **\*GET** or **\*VGET** command are valid only if the model is not modified after the **LSUM** command is issued.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Lines

---



## LSWRITE, *LSNUM*

**Writes load and load step option data to a file.**

[SOLUTION: Load Step Operations](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *LSNUM*

Number to be assigned to the load step file name for identification purposes. Defaults to 1 + highest *LSNUM* used in the current session. Issue **LSWRITE,STAT** to list the current value of *LSNUM*. Issue **LSWRITE,INIT** to reset to 1. The load step file will be named `JOBNAME.Sn`, where *n* is the specified *LSNUM* value (preceded by "0" for values 1-9). On systems with a 3-character limit on the file name extension, the "S" is dropped for *LSNUM* > 99.

## Command Default

The default behavior of **LSWRITE** is dependent on the **SOLCONTROL** command. When solution control is on (default behavior), the **LSWRITE** command does NOT write the default values for commands affected by solution control. When solution control is off (**SOLCONTROL,OFF**), **LSWRITE** does write the default values for commands affected by solution control. See the **SOLCONTROL** documentation for a list of those commands.

## Notes

Writes all load and load step option data for the selected model to a load step file for later use. **LSWRITE** does *not* capture changes made to real constants (**R**), material properties (**MP**), couplings (**CP**), or constraint equations (**CE**). Solid model loads will not be saved if the model is not meshed. Solid model loads, if any, are transferred to the finite element model. Issue **LSCLEAR,FE** to delete finite element loads. One file is written for each load step. Use the **LSREAD** command to read a single load step file, and the **LSDELE** command to delete load step files. Use the **LSSOLVE** command to read and solve the load steps sequentially.

**LSWRITE** does not support the following commands: **DJ**, **FJ**, **GSDATA**, **GSGDATA**, **ESTIF**, **EKILL**, **EALIVE**, **MPCHG**, and **OUTRES**. These commands will not be written to the load step file.

**LSWRITE** cannot be used with the birth-death option.

**LSWRITE** does not apply to FLOTRAN.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Write LS File**

**Main Menu>Solution>Load Step Opts>Write LS File**

**/LSYMBOL**, *X*, *Y*, *SYMANG*, *SYMTYP*, *SYMSIZ*, *KEYBMP*

**Creates annotation symbols (GUI).**

GRAPHICS: Annotation

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**X**

X location for symbol ( $-1.0 < X < 2.0$ ).

**Y**

Y location for symbol ( $-1.0 < Y < 1.0$ ).

**SYMANG**

Symbol orientation angle.

**SYMTYP**

Symbol type:

**1**

Arrow.

**2**

Tee.

**3**

Circle.

**4**

Triangle.

**5**

Star.

**SYMSIZ**

Symbol size multiplier (0.1 to 20.0). Defaults to 1.0.

**KEYBMP**

If *KEYBMP* = 1, the annotation is a bitmap. *SYMTYP* will then be a number from 1-99, indicating the bitmap type (see notes), and *X* and *Y* will define the lower left corner of the bitmap. The *SYMANG*, *SYMSIZ* arguments are ignored. If *KEYBMP* = 0, or blank, then the argument definitions above apply.

## Notes

Defines annotation symbols to be written directly onto the display at a specified location. This is a command generated by the GUI and will appear in the log file (*Jobname*.LOG) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All symbols are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** command to set the attributes of the symbol.

The *KEYBMP* argument reads the *sytype* argument to determine which bitmap to insert. This bitmap is defined by an integer between 1 and 99. Numbers 1 through 40 correspond to the standard texture values found in the **/TXTRE** command, while numbers 51 through 99 correspond to user supplied bitmaps, as defined using the Filename option of the **/TXTRE** command. Numbers 51 through 57 are predefined (the logos, clamps and arrows available from the GUI) but can be overridden. Numbers 41 through 50 are reserved.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

**LSYMM**, *Ncomp*, *NL1*, *NL2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

**Generates lines from a line pattern by symmetry reflection.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***Ncomp***

Symmetry key:

**X**

X symmetry (default).

**Y**

Y symmetry.

**Z**

Z symmetry.

### ***NL1*, *NL2*, *NINC***

Reflect lines from pattern beginning with *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and pattern is all selected lines [**LSEL**]. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

### ***KINC***

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

### ***NOELEM***

Specifies whether nodes and elements are also to be generated:

**0**

Generate nodes and elements associated with the original lines, if they exist.

**1**

Do not generate nodes and elements.

### ***IMOVE***

Specifies whether areas will be moved or newly defined:

**0**

Generate additional lines.

**1**

Move original lines to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

## Notes

Generates a reflected set of lines (and their corresponding keypoints and mesh) from a given line pattern by a symmetry reflection (see analogous node symmetry command, **NSYM**). The MAT, TYPE, REAL, and ESYS attributes are based upon the lines in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. The active coordinate system must be Cartesian. Lines in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Lines are generated as described in the **LGEN** command.

See the **ESYM** command for additional information about symmetry elements.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Reflect>Lines**

---

**LTAN**, *NL1*, *P3*, *XV3*, *YV3*, *ZV3*

**Generates a line at the end of, and tangent to, an existing line.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NL1**

Number of the line the generated line is tangent to. If negative, assume *P1* (see below), instead of *P2*, is the second keypoint of line *NL1*. If *NL1* = *P*, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**P3**

Keypoint at which generated line must end.

The following fields are used only if a specified slope on the end of the new line is desired, otherwise a zero curvature end slope will be automatically calculated. To specify the end slope, use the following fields to define a "slope vector" that has its tail at the origin and its head at the point *XV*, *YV*, *ZV* in the active coordinate system [**CSYS**]. The corresponding end slope of the line will then be parallel to this "slope vector."

**XV3**, **YV3**, **ZV3**

Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P3* end of the line. The tail of the vector is at the coordinate system origin.

## Notes

Generates a line (*P2-P3*) tangent at end point (*P2*) of line *NL1* (*P1-P2*).

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Tangent to Line**

---

**LTRAN**, *KCNTO*, *NL1*, *NL2*, *NINC*, *KINC*, *NOELEM*, *IMOVE***Transfers a pattern of lines to another coordinate system.**

PREP7:Lines

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**KCNTO**

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system. The coordinate system type and parameters of *KCNTO* must be the same as the active system.

**NL1, NL2, NINC**

Transfer lines from pattern beginning with *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and pattern is all selected lines [LSEL]. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

**KINC**

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [NUMSTR].

**NOELEM**

Specifies whether nodes and elements are also to be generated:

**0**

Generate nodes and elements associated with the original lines, if they exist.

**1**

Do not generate nodes and elements.

**IMOVE**

Specifies whether lines will be moved or newly defined:

**0**

Generate additional lines.

**1**

Move original lines to new position retaining the same keypoint numbers (*KINC* and *NOELM* are ignored). Valid only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

**Notes**

Transfers a pattern of lines (and their corresponding keypoints and mesh) from one coordinate system to another (see analogous node transfer command, **TRANSFER**). The MAT, TYPE, REAL, and ESYS attributes are based upon the lines in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Coordinate and slope values are interpreted in the active coordinate system and are transferred directly. Lines are generated as described in the **LGEM** command.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Lines**

**LUMPM**, *Key*

**Specifies a lumped mass matrix formulation.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**Key**

Formulation key:

**OFF**

Use the element-dependent default mass matrix formulation (default).

**ON**

Use a lumped mass approximation.

**Command Default**

Use the default element mass matrix.

**Notes**

This command is also valid in PREP7. If used in SOLUTION, this command is valid only within the first load step.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Preprocessor>Loads>Analysis Type>New Analysis**

**Main Menu>Solution>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>New Analysis**

---

**LVSCALE**, *FACT*, *LDSTEP*

**Scales the load vector for mode superposition analyses.**

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**FACT**

Scale factor applied to both the real and imaginary (if they exist) components of the load vector. Defaults to 0.0.

**LDSTEP**

Specifies the load step number (from the modal analysis) to be used in MSUP transient or harmonic analyses. Defaults to 1.

**Command Default**

No load vector is applied.

## Notes

Specifies the scale factor for the load vector that was created in a modal (**ANTYPE**,MODAL) analysis. Applies only to the mode superposition transient (**ANTYPE**,TRANS), mode superposition harmonic response (**ANTYPE**,HARMIC), and PSD spectrum (pressure PSD) analyses.

The **LVSCALE** command supports tabular boundary conditions (%TABNAME\_X%) for *FACT* input values only as a function of time in the mode superposition transient (**ANTYPE**,TRANS) or as a function of frequency in mode superposition harmonic response (**ANTYPE**,HARMIC).

**MPC contact** generates constraint equations that can include constant terms (included on the right-hand side of the system equation). The **LVSCALE** command scales the constant terms.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For Mode Super**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For PSD**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For Mode Super**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For PSD**  
**Main Menu>Solution>Define Loads>Apply>Load Vector>For Mode Super**  
**Main Menu>Solution>Define Loads>Apply>Load Vector>For PSD**  
**Main Menu>Solution>Define Loads>Delete>Load Vector>For Mode Super**  
**Main Menu>Solution>Define Loads>Delete>Load Vector>For PSD**

## LWPLAN, WN, NL1, RATIO

Defines the working plane normal to a location on a line.

DATABASE: Working Plane

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### WN

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified.

### NL1

Number of line to be used. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### RATIO

Location on *NL1*, specified as a ratio of the line length. Must be between 0.0 and 1.0. If *RATIO* = P, use graphical picking to specify location on the line.

## Command Default

Working plane is parallel to the Global X-Y plane at  $Z = 0.0$ .

## Notes

Defines a working plane (to assist in picking operations) normal to a location on a line. See **WPSTYL** command to set the style of working plane display.

This command is valid in any processor.

## Menu Paths

**Utility Menu>WorkPlane>Align WP with>Plane Normal to Line**

---



---

## M Commands

**M**, *NODE*, *Lab1*, *NEND*, *NINC*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

**Defines master degrees of freedom for reduced and superelement generation analyses.**

SOLUTION: Master DOF

```
MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS
```

### **NODE**

Node number at which master degree of freedom is defined. If ALL, define master degrees of freedom at all selected nodes (**NSEL**). If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

### **Lab1**

Valid degree of freedom label. If ALL, use all appropriate labels. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations). Thermal labels: TEMP, TBOT, TE2, TE3, . . . , TTOP (temperature). Electric labels: VOLT (voltage).

### **NEND, NINC**

Define all nodes from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1) as master degrees of freedom in the specified direction.

### **Lab2, Lab3, Lab4, . . . , Lab6**

Additional master degree of freedom labels. The nodes defined are associated with each label specified.

## Notes

Defines master degrees of freedom (MDOF) for reduced (dynamic and superelement generation) analyses. If defined for other analyses, MDOF are ignored. If used in SOLUTION, this command is valid only within the first load step. Scalar MDOF (VOLT) are not permitted in structural analyses if mass or damping matrices are reduced. Reduced analyses normally apply only to the UX, UY, UZ, ROTX, ROTY, and ROTZ labels.

Repeat **M** command for additional master degrees of freedom. The limit for the number of master nodes used is determined by the maximum system memory available.

The reduced modal (**ANTYPE**,MODAL), reduced transient (**ANTYPE**,TRANS), reduced harmonic response (**ANTYPE**,HARMIC), and the substructure (**ANTYPE**,SUBSTR) analyses utilize the matrix condensation technique to reduce the structure matrices to those characterized by a set of master degrees of freedom.

Master degrees of freedom are identified by a list of nodes and their nodal directions. The actual degree of freedom directions available for a particular node depends upon the degrees of freedom associated with element types (**ET**) at that node. There must be some mass (or stress stiffening in the case of the buckling analysis) associated with each master degree of freedom (except for the VOLT label). The mass may be due either to the distributed mass of the element or due to discrete lumped masses at the node. If a master degree of freedom is specified at a constrained point, it is ignored. If a master degree of freedom is specified at a coupled node, it should be specified at the prime node of the coupled set. Master degrees of freedom can also be generated automatically (during solution) by issuing the **TOTAL** command in PREP7 or SOLUTION.

Transient displacements and forces, used to apply motion to a structure in the reduced transient or reduced harmonic response analysis, must be applied at a master degree of freedom. Substructure analysis connection points must be defined as master degrees of freedom.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Master DOFs>User Selected>Define**

**Main Menu>Preprocessor>Modeling>CMS>CMS Interface>Define**

**Main Menu>Solution>Master DOFs>User Selected>Define**

---

**MADAPT**, *ERRTARGET*, *NADAPT*, *NMAX*, *KPLT*, *Ksmooth*, *KLST*, *KCD*, *DEVICE*

**Adaptively meshes and solves an edge-based model.**

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> EM <> <> <> <> EME <>

### ***ERRTARGET***

Target percentage for Zienkiewicz Zhu magnetic flux error (defaults to 5).

### ***NADAPT***

Maximum number of adaptive steps (defaults to 5).

### ***NMAX***

Maximum number of elements at which the iterations may continue (defaults to 50,000). Limits the number of elements that can be chosen for refinement.

### ***KPLT***

Plotting options:

**0**

No plot (default)

**1**

Elements and H

**2**

BERR error estimates

**3**

BDSG, BEPC error estimates

**4**

Adaptive details

If *KPLT* > 0, then **MADAPT** disables PowerGraphics for the adaptive solution.

### ***Ksmooth***

Smoothing options for refinement

**0**

No postprocessing will be done (default).

**1**

Smoothing will be done. Node locations may change.

- 2**  
Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change.

***KLST***

Listing options

- 0**  
No printing (default)
- 1**  
Final report
- 2**  
Report at each iteration step
- 3**  
Report Zienkiewicz Zhu magnetic errors BERR and BSGD
- 4**  
Regular details
- 5**  
Full details
- 6**  
Refine details
- 7**  
Track

***KCD***

Allows you to issue a **CDWRITE** or **CDREAD** at every iteration.

- 0**  
Do not issue **CDWRITE** or **CDREAD** (default).
- 1**  
Issue **CDWRITE** at every iteration (to save every mesh variation). This option issues **CDWRITE,geom**, writing the information to `jobnameN.cdb`.
- 2**  
Issue **CDREAD** at every iteration (to read every mesh variation). Reads from an existing `job-nameN.cdb`.

***DEVICE***

Defines the output device for plotting.

- 0**  
Screen only (default)
- 1**  
JPEG frames. Each frame is written to a file (`jobnameN.jpg` by default). See **/SHOW**.

**Notes**

**MADAPT** invokes a predefined ANSYS macro for adaptive meshing and solution of edge-based magnetic analyses. The macro causes repeated runs of the PREP7, SOLUTION, and POST1 phases of the ANSYS program with mesh density refinements based upon the percentage error in energy norm.

The **MADAPT** command macro requires a second, user-defined macro, which must be named `madaptld.mac` and must reside in the same directory where ANSYS is being run. This `madaptld` macro must contain loads and boundary conditions, based on permanent geometry or solid model features (such as sides or vertices). Loads specified in the `madaptld` macro cannot be based on node numbers because the node numbers will change throughout the refinement process. This secondary macro is required because the **MADAPT** macro process must delete all loads and boundary conditions at every refinement step.

**MADAPT** refines tetrahedral volume elements based on error. Hexahedra, wedges, and pyramids are not refined (see [NREFINE](#)).

This command is also valid at the Begin level.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

## MAGOPT, Value

Specifies options for a 3-D magnetostatic field analysis.

[SOLUTION: Load Step Options](#)

MP ME <> <> <> <> <> EM <> <> PP <> EME MFS

### Value

Option key:

**0**

Calculate a complete H field solution in the entire domain using a single (reduced) potential.

### Caution

When used in problems with both current sources and iron regions, errors may result due to numerical cancellation.

**1**

Calculate and store a preliminary H field in "iron" regions ( $\mu_r \neq 1$ ). Requires flux-parallel boundary conditions to be specified on exterior iron boundaries. Used in conjunction with subsequent solutions with *VALUE* = 2 followed by *VALUE* = 3. Applicable to multiply-connected iron domain problems.

**2**

Calculate and store a preliminary H field in "air" regions ( $\mu_r = 1$ ). The air-iron interface is appropriately treated internally by the program. Used in conjunction with a subsequent solution with *VALUE* = 3. Applicable to singly-connected iron domain problems (with subsequent solution with *VALUE* = 3) or to multiply-connected iron domain problems (when preceded by a solution with *VALUE* = 1 and followed by a solution with *VALUE* = 3).

**3**

Use the previously stored H field solution(s) and calculate the complete H field.

## Notes

Specifies the solution sequence options for a 3-D magnetostatic field analysis using a scalar potential (MAG). The solution sequence is determined by the nature of the problem.

You cannot use constraint equations with *Value* = 1.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** The **MAGOPT,3** option is not supported in Distributed ANSYS when the following contact elements are present in the model: [CONTA173](#), [CONTA174](#), [CONTA175](#), [CONTA176](#), or [CONTA177](#).

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>DSP Method**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>GSP Method**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>RSP Method**  
**Main Menu>Solution>Load Step Opts>Magnetics>Options Only>DSP Method**  
**Main Menu>Solution>Load Step Opts>Magnetics>Options Only>GSP Method**  
**Main Menu>Solution>Load Step Opts>Magnetics>Options Only>RSP Method**

---

## MAGSOLV, OPT, NRAMP, CNVCSG, CNVFLUX, NEQIT, BIOT, CNVTOL

**Specifies magnetic solution options and initiates the solution.**

[SOLUTION: Load Step Options](#)

MP ME <> <> <> <> <> EM <> <> <> EME MFS

### OPT

Static magnetic solution option:

- 0** Vector potential (MVP) or edge formulation (default).
- 1** Combined vector potential and reduced scalar potential (MVP-RSP).
- 2** Reduced scalar potential (RSP).
- 3** Difference scalar potential (DSP).
- 4** General scalar potential (GSP).

### NRAMP

Number of ramped substeps for the first load step of a nonlinear MVP or MVP-RSP solution. Defaults to 3. If *NRAMP* = -1, ignore the ramped load step entirely. *NRAMP* is ignored for linear magnetostatics.

### CNVCSG

Tolerance value on the program-calculated reference value for the magnetic current-segment convergence. Used for the MVP, the MVP-RSP, and the edge formulation solution options (*OPT* = 0 and 1). Defaults to 0.001.

**CNVFLUX**

Tolerance value on the program-calculated reference value for the magnetic flux convergence. Used for all scalar potential solution options ( $OPT = 2, 3, 4$ ). Defaults to 0.001.

**NEQIT**

Maximum number of equilibrium iterations per load step. Defaults to 25.

**BIOT**

Option to force execution of a Biot-Savart integral solution [**BIOT,NEW**] for the scalar potential options. Required if multiple load steps are being performed with different current source primitives (**SOURC36** elements).

**0**

Do not force execution of Biot-Savart calculation (default); Biot-Savart is automatically calculated only for the first solution.

**1**

Force execution of Biot-Savart calculation.

**CNVTOL**

Sets the convergence tolerance for AMPS reaction. Defaults to 1e-3. Applicable only when **KEYOPT(1) = 5** on **SOLID117**.

## Notes

**MAGSOLV** invokes an ANSYS macro which specifies magnetic solution options and initiates the solution. The macro is applicable to any ANSYS magnetostatic analysis using the magnetic vector potential (MVP), reduced scalar potential (RSP), difference scalar potential (DSP), general scalar potential (GSP), or combined MVP-RSP formulation options. Results are only stored for the final converged solution. (In **POST1**, issue **\*SET,LIST** to identify the load step of solution results.) The macro internally determines if a nonlinear analysis is required based on magnetic material properties.

If you use the **BIOT** option and issue **SAVE** after solution or postprocessing, the Biot-Savart calculations are saved to the database, but will be overwritten upon normal exit from the program. To save this data after issuing **SAVE**, use the **/EXIT,NOSAVE** command. You can also issue the **/EXIT,SOLU** command to exit ANSYS and save all solution data, including the Biot-Savart calculations, in the database. Otherwise, when you issue **RESUME**, the Biot-Savart calculation will be lost (resulting in a zero solution).

The MVP, MVP-RSP, and edge formulation options perform a two-load-step solution sequence. The first load step ramps the applied loads over a prescribed number of substeps (**NRAMP**), and the second load step calculates the converged solution. For linear problems, only a single load step solution is performed. The ramped load step can be bypassed by setting **NRAMP** to -1.

The RSP option solves in a single load step using the adaptive descent procedure. The DSP option uses two load steps, and the GSP solution uses three load steps.

The following analysis options and nonlinear options are controlled by this macro: **KBC**, **NEQIT**, **NSUBST**, **CNVTOL**, **NROPT**, **MAGOPT**, and **OUTRES**.

You cannot use constraint equations with  $OPT = 4$ .

## Menu Paths

**Main Menu>Solution>Solve>Electromagnet>Static Analysis>Opt&Solv**

---

**/MAIL**, --, *Address*, *Fname*, *Ext*

**Mails file to the specified address.**

APDL: Macro Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

--

Unused field.

### **Address**

Email address (up to 64 characters) of the intended recipient of the file.

### **Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

### **Ext**

Filename extension (8 character maximum).

## Notes

Issue the **/MAIL** command to alert someone when a long-running job has completed, as shown in this example:

```
...
SOLVE
/MAIL, ,yourname@yourdomain.com, jobdone,txt
```

If you are running ANSYS in a Microsoft Windows environment, you must configure BLAT (a freeware program provided with ANSYS). Before issuing the **/MAIL** command, enter this command from the C:\Program Files\Ansys Inc\V130\Ansys\Bin\Intel directory:

If you are running ANSYS in a Windows x64 environment, enter this command from the C:\Program Files\Ansys Inc\V130\Ansys\Bin\Winx64 directory:

```
blat -install smtphost.bar.com you@bar.com
```

For more information about the BLAT program, see the BLAT\_README.TXT file in the same directory.

Because Windows cannot operate on a busy file, you cannot mail standard ANSYS output. Instead, redirect the output to a file and then mail that file, as shown in this example:

```
...
/PREP7
/OUT,scratch.out
:
:
FINISH
/OUT
/MAIL, ,YourName@YourDomain.com,scratch.out
FINISH
```

## Menu Paths

**This command cannot be accessed from a menu.**

**MAPSOLVE**, MAXSBSTEP

Maps solved node and element solutions from an original mesh to a new mesh.

SOLUTION: Rezoning

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**MAXSBSTEP**

The maximum number of substeps for rebalancing the residuals. The default value is 5.

**Notes**

Used during the [rezoning](#) process, the **MAPSOLVE** command maps solved node and element solutions from the original mesh to the new mesh and achieves equilibrium based on the new mesh.

Additional substeps are necessary to reduce the residuals to zero.

During the rebalancing stage, the external loads and time remain unchanged.

The **MAPSOLVE** command is valid for manual rezoning (**REZONE**,MANUAL) only.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>Solution>Manual Rezoning>Map Results

---

**MASTER**

Specifies "Master DOF" as the subsequent status topic.

SOLUTION: Status

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

**Notes**

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

**Menu Paths**

Utility Menu>List>Status>Solution>Master DOF

---



**MAT**, *MAT*

Sets the element material attribute pointer.

[PREP7: Meshing](#)  
[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**MAT**

Assign this material number to subsequently defined elements (defaults to 1).

**Notes**

Identifies the material number to be assigned to subsequently defined elements. This number refers to the material number (*MAT*) defined with the material properties [**MP**]. Material numbers may be displayed [**/PNUM**].

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs**

**Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes**

**MATER**

Specifies "Material properties" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (*Jobname.LOG*) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

**Menu Paths**

**Utility Menu>List>Status>Preprocessor>Materials**

**MCHECK**, *Lab*

Checks mesh connectivity.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Lab**

Operation:

**ESEL**

Unselects the valid elements.

**Notes**

Wherever two area or volume elements share a common face, **MCHECK** verifies that the way the elements are connected to the face is consistent with their relative normals or integrated volumes. (This may detect folds or otherwise overlapping elements.)

**MCHECK** verifies that the element exterior faces form simply-connected closed surfaces. (This may detect unintended cracks in a mesh.)

**MCHECK** warns if the number of element facets in a 2-D loop or 3-D shell is not greater than a computed limit. This limit is the smaller of either three times the number of faces on one element, or one-tenth the total number of element faces in the model. (This may detect holes in the middle of a mesh.)

The **MCHECK** command will perform a number of validity checks on the selected elements, including:

1. **Normal check:** Wherever two area elements share a common edge, **MCHECK** verifies that the ordering of the nodes on each element is consistent with their relative normals.
2. **Volume check:** Wherever two volume elements share a common face, **MCHECK** verifies that the sign of the integrated volume of each element is consistent.
3. **Closed surface check:** **MCHECK** verifies that the element exterior faces form simply-connected closed surfaces (this may detect unintended cracks in a mesh).
4. **Check for holes in the mesh:** **MCHECK** warns if the number of element faces surrounding an interior void in the mesh is small enough to suggest one or more accidentally omitted elements, rather than a deliberately formed hole. For this test, the number of faces around the void is compared to the smaller of a) three times the number of faces on one element, or b) one-tenth the total number of element faces in the model.

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Check Mesh>Connectivity>Ck Connectivity**  
**Main Menu>Preprocessor>Meshing>Check Mesh>Connectivity>Sel Bad Connt**

**MDAMP**, *STLOC*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

**Defines the damping ratios as a function of mode.**

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**STLOC**

Starting location in table for entering data. For example, if *STLOC* = 1, data input in the *V1* field applies to the first constant in the table. If *STLOC* = 7, data input in the *V1* field applies to the seventh constant in the table, etc. Defaults to the last location filled + 1.

**V1, V2, V3, ..., V6**

Data assigned to six locations starting with *STLOC*. If a value is already in this location, it will be redefined. Blank values for *V2* to *V6* leave the corresponding previous value unchanged.

## Notes

Defines the damping ratios as a function of mode. Table position corresponds to mode number. These ratios are added to the **DMPRAT** value, if defined. Use **STAT** command to list current values. Applies to the mode superposition harmonic response (**ANTYPE,HARMIC**), the mode superposition linear transient dynamic (**ANTYPE,TRANS**), and the spectrum (**ANTYPE,SPECTR**) analyses. Repeat **MDAMP** command for additional constants (10000 maximum).

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping**

**MDELE**, *NODE*, *Lab1*, *NEND*, *NINC*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

**Deletes master degrees of freedom.**

**SOLUTION: Master DOF**

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

**NODE**, *Lab1*, **NEND**, **NINC**

Delete master degrees of freedom in the *Lab1* direction [**M**] from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1). If *NODE* = ALL, *NEND* and *NINC* are ignored and masters for all selected nodes [**NSEL**] are deleted. If *Lab1* = ALL, all label directions will be deleted. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

*Lab2*, *Lab3*, *Lab4*, . . . , *Lab6*

Delete masters in these additional directions.

## Notes

Deletes master degrees of freedom. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Master DOFs>User Selected>Delete**

**Main Menu>Solution>Master DOFs>User Selected>Delete**

**MDPLOT**, *Function, Dmpname, Scale***Plots frequency-dependent modal damping coefficients calculated by DMPEXT.**

SOLUTION: Dynamic Options

MP ME &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

**Function**

Function to display.

**d\_coeff**

Damping coefficient

**s\_coeff**

Squeeze coefficient

**d\_ratio**

Damping ratio

**s\_ratio**

Squeeze stiffness ratio

**Dmpname**

Array parameter name where damping information is stored. Defaults to d\_damp.

**Scale**

Indicates whether to perform a linear or a double logarithmic plot.

**LIN**

Perform a linear plot. Default

**LOG**

Perform a double logarithmic plot.

**Notes**See [Thin Film Analysis](#) for more information on thin film analyses.**Menu Paths****Main Menu>General Postproc>Plot Results>ThinFilm>Plot Dmp Parm****MEMM**, *Lab, Kywrd***Allows the current session to keep allocated memory**

SESSION: Run Controls

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**Lab**When *Lab* = KEEP, the memory manager's ability to acquire and keep memory is controlled by *Kywrd***Kywrd**

Turns the memory "keep" mode on or off

**ON**

Keep any memory allocated during the analysis.

**OFF**

Use memory dynamically and free it up to other users after use (default).

**Command Default**

*Kywrđ* = OFF.

**Notes**

You can use the **MEMM** command to ensure that memory intensive operations will always have the same memory available when the operations occur intermittently. Normally, if a large amount of memory is allocated for a specific operation, it will be returned to the system once the operation is finished. This option always maintains the highest level used during the analysis until the analysis is finished.

The **MEMM** command does not affect the value you specify with the `-m` switch. When you allocate memory with the `-m` switch, that amount will always be available. However, if dynamic memory allocation in excess of the `-m` value occurs, you can use the **MEMM** command to ensure that amount is retained until the end of your analysis.

**Menu Paths**

**This command cannot be accessed from a menu.**

**/MENU**, *Key*

**Activates the Graphical User Interface (GUI).**

**SESSION: Run Controls**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Key**

Activation key:

**ON**

Activates the menu system (device dependent).

**GRPH**

Enters non-GUI graphics mode. This option is intended for use on graphics devices that do not support the full Motif-based GUI.

**Command Default**

GUI is on if entering the ANSYS program through the launcher. GUI is off if entering using the ANSYS execution command.

**Notes**

Activates the Graphical User Interface (GUI).

### Caution

if you include the **/MENU,ON** command in your start130.ans, it should be the *last* command in the file. Any commands after **/MENU,ON** may be ignored. (It is not necessary to include the **/SHOW** and **/MENU,ON** commands in start130.ans if you will be using the launcher to enter the ANSYS program.)

This command is valid in any processor.

### Menu Paths

**This command cannot be accessed from a menu.**

---

## MESHING

Specifies "Meshing" as the subsequent status topic.

PREP7: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

### Menu Paths

**Utility Menu>List>Status>Preprocessor>Meshing**

---

## MFANALYSIS, Key

Turns an ANSYS Multi-field solver analysis on or off.

SOLUTION: Multi-field Global Controls

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions

### Key

Multifield analysis key:

#### ON

Activates an ANSYS Multi-field solver analysis.

#### OFF

Deactivates an ANSYS Multi-field solver analysis (default).

## Notes

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>Select method**  
**Main Menu>Solution>Multi-field Set Up>Select method**

---

## MFBUCKET, *Key*, *Value*

**Turns a bucket search on or off.**

[SOLUTION: Multi-field Interface Mapping](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
[Product Restrictions](#)

### *Key*

Bucket search key:

#### **ON**

Activates a bucket search (default).

#### **OFF**

Deactivates a bucket search. A global search is then activated.

### *Value*

Scaling factor (%) used to determine the number of buckets for a bucket search. Defaults to 50%.

## Notes

A bucket search will more efficiently compute the mapping of surface and volumetric interpolation data across field interfaces (flagged by the FSIN label using **SF**, **SFA**, **SFE**, or **SFL** or the FVIN label using **BFE**).

The number of buckets used to partition a flagged interface is equal to the scaling factor (%) times the total number of interface elements. For example, for the default scaling factor of 50% and a 10,000 element interface, 5,000 buckets are used.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Setup>Global**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Mapping**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Setup>Global**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Mapping**

## MFCALC, *FNUMB*, *FREQ*

**Specifies a calculation frequency for a field in an ANSYS Multi-field solver analysis.**

*SOLUTION:* Multi-field Time Controls

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *FNUMB*

Field number set by the **MFELEM** command.

### *FREQ*

Perform calculation every Nth ANSYS Multi-field solver time step. Defaults to 1 for every time step.

## Notes

This command only applies to a harmonic analysis of the specified field. It is useful when a field contributes negligible field interaction within a single ANSYS Multi-field solver time step.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Frequency**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Frequency**



**MFCI**, *VAL1*, *VAL2*

Sets the control parameters used by the conservative (CPP) interpolation scheme.

[SOLUTION: Multi-field Interface Mapping](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**VAL1**

Controls the pixel resolution. The higher the resolution, the more accurate and more expensive the conservative (CPP) interpolation will be. Valid values are 10 to 256; defaults to 100.

**VAL2**

The separation factor to handle any gap between the two surfaces. It is a relative measure of the gap, normalized by the averaged element face sizes from both sides of the interface. Defaults to 0.1.

**Notes**

In a conservative (CPP) interpolation scheme as specified on the **MFLCOMM** command, each element face is first divided into  $n$  number of faces, where  $n$  is the number of nodes on the face. The three-dimensional faces are then converted onto a two-dimensional polygon made up of rows and columns of dots called pixels. By default, these pixels have a resolution of 100 x 100; use *VAL1* to increase the resolution and improve the accuracy of the algorithm. See [Load Interpolation](#) in the *Coupled-Field Analysis Guide* for more information on interpolation schemes and adjusting the pixel resolution for the conservative interpolation scheme.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Mapping**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Mapping**

**MFCLEAR**, *Option*, *Value*

Deletes ANSYS Multi-field solver analysis settings.

[SOLUTION: Multi-field Global Controls](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> <>

[Product Restrictions](#)

**Option****SOLU**

Resets all ANSYS solution commands except **KBC** to their default states. This option clears analysis options when setting up different fields for an ANSYS Multi-field solver analysis.

**FIELD**

Deletes all ANSYS Multi-field solver specifications for the specified field number.

**SINT**

Deletes all ANSYS Multi-field solver specifications for the specified surface interface number.

**VINT**

Deletes all ANSYS Multi-field solver specifications for the volumetric interface number.

**ORD**

Deletes the analysis order specified by the **MFORDER** command.

**EXT**

Deletes external fields specified by the **MFEXTER** command

**MFLC**

Deletes load transfers specified by the **MFLCOMM** command

**Value**

Use only for *Option* = FIELD, SINT, or VINT.

*Option* = FIELD deletes all ANSYS Multi-field solver specifications for the field number *Value*. *Value* defaults to 0 (no deletions). A *Value* of -1 deletes the ANSYS Multi-field solver settings for all fields.

*Option* = SINT deletes surface interface specifications for the surface interface number *Value*. *Value* defaults to 0 (no deletions). A *Value* of -1 deletes all surface interface specifications. This command deletes the ANSYS Multi-field solver commands, not the boundary conditions themselves.

*Option* = VINT deletes all volumetric interface specifications for the volume interface number *Value*. *Value* defaults to 0 (no deletions). A *Value* of -1 deletes all volumetric interface specifications. This command deletes the ANSYS Multi-field solver commands, not the boundary conditions themselves.

**Notes**

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Product Restrictions**

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

**Menu Paths**

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Clear**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Clear**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Solution Ctrl**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Clear**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Clear**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Solution Ctrl**

---

**MFCMMAND**, *FNUMB*, *Fname*, *Ext*

**Captures field solution options in a command file.**

SOLUTION: Multi-field Definitions

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**FNUMB**

Field number specified by the **MFELEM** command.

***Fname***

Command file name specified for the field number. Defaults to field "FNUMB".

***Ext***

Extension for *Fname*. Defaults to .cmd.

**Notes**

All relevant solution option commands for the specified field are written to a file with the extension .cmd. Refer to the commands in the following tables in the [Command Reference: Analysis Options, Nonlinear Options, Dynamic Options, and Load Step Options](#).

This command is also valid in PREP7.

See [Multi-field Commands](#) in the [Coupled-Field Analysis Guide](#) for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Capture**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Capture**

**MFCONV**, *Lab*, *TOLER*, *--*, *MINREF*

**Sets convergence values for an ANSYS Multi-field solver analysis.**

[SOLUTION: Multi-field Convergence Controls](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions

***Lab***

Valid labels:

Force labels: FX, FY, FZ

Displacement labels: UX, UY, UZ

Temperature label: TEMP

Heat flux label: HFLU (for surface load transfer)

Heat generation label: HGEN (for volume load transfer)

If *Lab* = ALL, **MFCONV** applies the convergence value to all nine variables.

***TOLER***

Convergence tolerance about program calculated reference value (the L2 norm of the new load in a multi-field analysis). Defaults to 0.01 (1%) for all labels. Must be less than 1.0.

*--*

Unused field.

***MINREF***

The minimum value allowed for the program calculated reference value. If negative, no minimum is enforced. Defaults to 1.0e-6 for all labels. Not available in the GUI. *MINREF* corresponds to  $\|\phi_{\text{new}}\|$  as defined in [Set up Stagger Solution](#) in the [Coupled-Field Analysis Guide](#).

## Notes

**MFCONV** sets convergence values for variables at the ANSYS Multi-field solver interface.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Stagger>Convergence**  
**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Convergence**  
**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Stagger>Convergence**  
**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Convergence**

---

## MFDTIME, DTIME, DTMIN, DTMAX, Carry

Sets time step sizes for an ANSYS Multi-field solver analysis.

[SOLUTION: Multi-field Time Controls](#)

[MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS](#)  
[Product Restrictions](#)

### **DTIME**

Multi-field time step size. If automatic time stepping is being used [see Notes below], *DTIME* is the starting time step.

### **DTMIN**

Minimum time step. Defaults to *DTIME*.

### **DTMAX**

Maximum time step. Defaults to *DTIME*.

### **Carry**

Time step carryover key.

#### **OFF**

Use *DTIME* as the starting time step for the next restart run (default).

#### **ON**

Use the final time step from the previous run as the starting time step for the next restart run.

## Notes

This command specifies time step sizes for an ANSYS Multi-field solver analysis. If either *DTMIN* or *DTMAX* is not equal to *DTIME*, auto time-stepping is turned on for the multi-field loop. ANSYS will automatically

adjust the time step size for the next multi-field step between *DTMIN* and *DTMAX*, based on the status of the current convergence, the number of target stagger iterations (specified by **MFITER**), and the actual number of iterations needed to reach convergence at the current time step.

If auto time-stepping is turned off, the time step size must be evenly divisible into the end time (specified by **MFTIME**) minus the start time (0 for a new analysis or a restart time specified by **MFRSTART**).

You can use a smaller time step within each ANSYS field analysis. This is called subcycling. Use the **DELTIM** and **AUTOTS** commands to subcycle a structural, thermal, or electromagnetic analysis. Use the **FLDATA4** command to subcycle a fluid analysis.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Time Ctrl**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Time Ctrl**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Time Ctrl**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Time Ctrl**

---

## MFELEM, FNUMB, ITYPE1, ITYPE2, ITYPE3, ITYPE4, ITYPE5, ITYPE6, ITYPE7, ITYPE8, ITYPE9, ITYPE10

**Defines a field by grouping element types.**

[SOLUTION: Multi-field Definitions](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### **FNUMB**

Field number for a group of element types.

### **ITYPE1, ITYPE2, ITYPE3, . . . , ITYPE10**

Element types defined by the **ET** command.

## Notes

You can define up to ten element types per field.

Define only element types that contain elements in the field. Do not include **MESH200** because it is a “mesh-only” element that does not contribute to the solution.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Define>Define**  
**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Define>Define**

**MFEM**, *FNUMB*, *ITYPE1*, *ITYPE2*, *ITYPE3*, *ITYPE4*, *ITYPE5*, *ITYPE6*, *ITYPE7*, *ITYPE8*, *ITYPE9*, *ITYPE10*

**Add more element types to a previously defined field number.**

[SOLUTION: Multi-field Definitions](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *FNUMB*

Existing field number defined by the **MFELEM** command.

*ITYPE1*, *ITYPE2*, *ITYPE3*, . . . , *ITYPE10*

Element types defined by the **ET** command.

## Notes

You can add up to ten element types per **MFEM** command. This command should not be used after an initial solution.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Define>Add elems**  
**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Define>Add elems**

**MFEXTER**, *FNUMB1*, *FNUMB2*, *FNUMB3*, *FNUMB4*, *FNUMB5*, *FNUMB6*, *FNUMB7*, *FNUMB8*, *FNUMB9*, *FNUMB10*,  
*FNUMB11*, *FNUMB12*, *FNUMB13*, *FNUMB14*, *FNUMB15*, *FNUMB16*, *FNUMB17*, *FNUMB18*, *FNUMB19*, *FNUMB20*

**Defines external fields for an ANSYS Multi-field solver analysis.**

[SOLUTION: Multi-field Definitions](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

*FNUMB1*, *FNUMB2*, *FNUMB3*, . . . , *FNUMB20*

External field numbers defined by the **MFELEM** command.

## Notes

This command specifies external field numbers to be used for load transfer in an ANSYS Multi-field solver analysis. Use the **MFIMPORT** command to import the external fields.

Use the **MFELEM** command to specify external field numbers. Use the **MFORDEF** command to specify the solution order for the external fields.

You can define a maximum of 20 fields.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Setup>External**  
**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Setup>External**

**MFFNAME**, *FNUMB*, *Fname*

**Specifies a file name for a field in an ANSYS Multi-field solver analysis.**

[SOLUTION: Multi-field Definitions](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***FNUMB***

Field number specified by the **MFELEM** command.

***Fname***

File name. Defaults to field "*FNUMB*".

## Notes

All files created for the field will have this file name with the appropriate extensions.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Define>Define**  
**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Define>Define**

**MFFR**, *Fname*, *Lab*, *RFINI*, *RFMIN*, *RFMAX*

### Setup Multi-Field relaxation factors for field solutions.

SOLUTION: Multi-field Global Controls

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions

#### ***Fname***

Field name (MFX) or number (MFS). Must be the ANSYS field (cannot be a FLOTRAN or CFX field).

#### ***Lab***

Label name. Valid values are DISP and TEMP.

#### ***RFINI***

Initial relaxation factor. Defaults to 0.75.

#### ***RFMIN***

Minimum relaxation factor. Defaults to *RFINI*.

#### ***RFMAX***

Maximum relaxation factor. Defaults to *RFINI*.

## Notes

Use this command to relax the field solutions in fluid-solid interaction analyses and thermal-thermal analyses for a better convergence rate in coupled problems, especially cases that need dynamic relaxation. The ANSYS field that has the MFFR command applied will do only one nonlinear stagger iteration within each multi-field stagger; the convergence of the ANSYS field solver will be satisfied through multiple multi-field staggers. Note that the CFX field solver can have multiple iterations within the field solver; see the CFX documentation for more details. ANSYS will not terminate the nonlinear field solution until the ANSYS field solver converges or reaches the maximum number of multi-field staggers as specified on **MFITER**.

The interface load relaxation (**MFRELAX**) will be automatically turned off for the corresponding surface loads that have **MFFR** applied. The automatic change of the relaxation factor for accelerating the nonlinear convergence of the coupled field solution is based on Aitken's acceleration scheme.

This command is valid only with coupled problems involving surface load transfer only. No subcycling is allowed for the field solver if using this command.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**



**MFIMPORT**, *FNUMB*, *Option*, *Fname*, *Ext*

**Imports a new field into a current ANSYS Multi-field solver analysis.**

[PREP7: Database](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***FNUMB***

Field number specified by the **MFELEM** command.

***Option***

Selects data to read.

**DB**

Reads a CDB file. The CDB file name and extension are specified by *Fname* and *Ext*.

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to CDB if *Fname* is blank.

## Notes

The field to be imported should be written to a CDB file (**CDWRITE** command). This file is read into the database, offsetting all existing element type numbers, node numbers, etc. in order to accommodate the imported field. (See the **NUMOFF** command for information on offset capabilities.) It then updates all of the previously issued MFxx commands to the new element type numbers. A new field is created using the specified field number, which must not currently exist. If there are no ANSYS Multi-field solver command files written for the existing fields in the database, one will be written for each field with the default name (see the **MFCMMAND** command). A **MFCMMAND** will be issued for the imported field as well.

Repeat the **MFIMPORT** command to import additional fields. If you are importing a FLOTRAN fluid field along with other fields, you must import the FLOTRAN fluid field last to ensure the FLOTRAN elements have material type 1. FLOTRAN requires its elements to have material type 1.

If the fluid field is imported last and **MP** commands were used to define the FLOTRAN material properties, the non-FLOTRAN elements will have material numbers greater than 1. However, if **MP** commands were not used to define the FLOTRAN material properties, some non-FLOTRAN elements may also have material number 1. This material numbering anomaly can occur, because if a **MP** command is not present in the fluid field's CDB file, then **MFIMPORT** will not offset the material numbers when importing the fluid field. Thus, it is possible for FLOTRAN and non-FLOTRAN elements to share material number 1. The solution will not be affected, because the MFS internally manages the material numbers to ensure that the correct material properties are used with each field. However, the ability to select elements based on their material properties may be compromised. If unique material numbers are desired, **NUMOFF,MAT,1** can be explicitly included in the fluid field's CDB file to force the material numbers for the non-FLOTRAN elements to be offset.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Import**

---

### **MFINTER**, *Option*

**Specifies the interface load transfer interpolation option for an ANSYS Multi-field solver analysis.**

[SOLUTION: Multi-field Global Controls](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

#### *Option*

Interface load transfer option:

#### **CONS**

Conservative formulation for load transfer.

#### **NONC**

Nonconservative formulation for load transfer (default).

## Notes

This command only applies to the interpolation method for forces, heat flux, and heat generation. Displacement and temperature transfers are always nonconservative.

For more information on conservative and nonconservative load transfer, see [Load Transfer](#) in the *Coupled-Field Analysis Guide*.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Setup>Global**  
**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Setup>Global**

---

**MFITER**, *MAXITER*, *MINITER*, *TARGET*

Sets the number of stagger iterations for an ANSYS Multi-field solver analysis.

**SOLUTION:** Multi-field Convergence Controls

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions

**MAXITER**

Maximum number of iterations. Defaults to 10.

**MINITER**

Minimum number of iterations. Defaults to 1.

**TARGET**

Target number of iterations. Defaults to 5.

## Notes

The number of stagger iterations applies to each time step in an ANSYS Multi-field solver analysis. *MINITER* and *TARGET* are valid only when multi-field auto time stepping is on (**MFDTIME**).

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Stagger>Iterations**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Iterations**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Stagger>Iterations**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Iterations**

**MFLCOMM**, *Type*, *Fname1*, *Inname1*, *Label1*, *Fname2*, *Inname2*, *Label2*, *Option*

Defines a load transfer for code coupling analyses.

**SOLUTION:** Multi-field Load Transfer

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**Type**

Set to SURF for a surface load transfer. Only surface load transfers are available for MFX.

**Fname1**

Sets the field solver name for the server (sending) code with a case-sensitive character string of up to 80 characters.

***Inname1***

Sets the interface name or number for the field solver of the server code. ANSYS interfaces are numbered and are defined by the SF family of commands (**SF**, **SFA**, or **SFE**) with the FSIN surface load label. CFX interfaces use names, which are set in CFX-Pre.

***Label1***

Sets the surface load label for the field solver of the server code with a character string of up to 80 characters. ANSYS uses a combination of the label and option to determine what data is transferred (e.g., heat flows and not fluxes are sent with the label/option pair HFLU/ CPP). ANSYS cannot serve total force or total force density to CFX for either formulation. CFX will send the data requested by the label regardless of the option. CFX labels that have more than one word must be enclosed in single quotes. Note that this field is case-sensitive; i.e., FORC will work, but forc will not.

<b>CFX Label</b>	<b>ANSYS Label</b>	<b>Option</b>	<b>Definition</b>
'Total Force'	FORC	CPP	Sets the surface load label to force.
'Total Force Density'	FORC	NONC	Sets the surface load label to the total force per unit area associated with a mesh node.
'Total Mesh Displacement'	DISP	NONC, CPP	Sets the surface load label to displacement.
Temperature	TEMP	NONC, CPP	Sets the surface load label to temperature.
'Wall Heat Flow'	HFLU	CPP	Sets the surface load label to total wall heat flow.
'Wall Heat Flux'	HFLU	NONC	Sets the surface load label to heat flux.

***Fname2***

Sets the field solver name for the client (receiving) code with a character string of up to 80 characters.

***Inname2***

Sets the interface name or number for the field solver of the client code with a character string of up to 80 characters. ANSYS interfaces are numbered and are defined by the SF family of commands (**SF**, **SFA**, or **SFE**) with the FSIN surface load label. CFX interfaces use names, which are set in CFX-Pre.

***Label2***

Sets the surface load label for the field solver of the client code with a character string of up to 80 characters. ANSYS uses a combination of the label and option to determine what data is transferred (e.g., heat flows and not fluxes are sent with the label-option pair HFLU/ CPP). CFX will send the data requested by the label regardless of the option. CFX labels that have more than one word must be enclosed in single quotes. Note that this field is case-sensitive; i.e., FORC will work, but forc will not.

***Option*****NONC**

Profile preserving: Sets the interface load transfer to the nonconservative formulation (default for displacement and temperature). In the nonconservative formulation, the force density (or heat flux) is transferred across the interface, preserving the density profile between the two fields.

**CPP**

Conservative: Uses a local conservative formulation while preserving the density profile (default for total force and wall heat flow). In the conservative formulation, total force (or heat flow) must be transferred across the interface from the CFX field solver to the ANSYS field solver.

**Notes**

ANSYS input should always be in consistent units for its model.

ANSYS uses a combination of the label and option to determine what data to transfer. CFX will send exactly the data requested by the label, regardless of the option. However, for the NONC option, the CFX label must be Total Force Density or Wall Heat Flux and for the CPP option, the CFX label must be Total Force or Wall Heat Flow.

For more information on profile preserving and conservative load transfer, see [Load Interpolation](#) in the *Coupled-Field Analysis Guide*. [Mapping Diagnostics](#) are also available; however, if the improperly-mapped nodes are based on the CFX mesh, you should ignore the ANSYS-generated components because the CFX nodes are not present in the ANSYS database.

If you are working interactively, you can choose two pre-defined combinations, **Mechanical** or **Thermal**, or you can choose a **Custom** option. If you choose the **Mechanical** load type, then the Total Force Density and Total Mesh Displacement data (corresponding to the ANSYS FORC and DISP labels, respectively) is transferred. If you choose the **Thermal** load type, then the Temperature and Wall Heat Flux data (corresponding to the ANSYS TEMP and HFLU labels, respectively) is transferred. If you choose **Custom**, you can select any valid combination of label and option as described above.

The ANSYS Multi-field solver does not allow you to switch the load transfer direction for the same load quantity across the same interfaces for a restart run. For example, if Field1 sends temperature to and receives heat flow from Field2 across Interface 1 in a previous solution, then you cannot make Field1 send heat flow to and receive temperatures from Field2 across the same interface in a restart run, even if you cleared the corresponding load transfer command.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Load Transfer**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Load Transfer**

**MFLIST**, *Option*, *Value*

**Lists the settings for an ANSYS Multi-field solver analysis.**

**SOLUTION: Multi-field Global Controls**

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions

***Option*****ALL**

Lists all ANSYS Multi-field solver analysis options.

**SOLU**

Lists all solution-related ANSYS Multi-field solver options.

**FIELD**

Lists all ANSYS Multi-field solver options related to the specified field number.

**SINT**

Lists all surface interface information for the specified surface interface number.

**VINT**

Lists all volumetric interface information for the specified volumetric interface number.

***Value***

Use only for *Option* = FIELD, SINT, or VINT.

*Option* = FIELD lists all ANSYS Multi-field solver options for the field number *Value*. Value defaults to 0, which lists information for all fields.

*Option* = SINT lists all surface interface information for the interface number *Value*. Value defaults to 0, which lists information for all surface interfaces.

*Option* = VINT lists all volumetric interface information for interface number *Value*. Value defaults to 0, which lists information for all volumetric interfaces.

**Notes**

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Product Restrictions**

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

**Menu Paths**

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Status**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Status**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Status**

**MFMAP**, *Lab1*, *Lab2*, *Filename*, *Opt*

Calculates, saves, resumes, or deletes mapping data in an ANSYS Multi-field solver analysis.

[SOLUTION: Multi-field Interface Mapping](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**Lab1**

Operation label:

**CALC**

Calculate mapping data and keep it in memory (default).

**SAVE**

Calculate mapping data, keep it in memory, and save it to a file. (If MFMAP,CALC or MFMAP,RESU have been issued, just save it to a file.)

**RESU**

Resume the mapping from a file and keep it in memory.

**DELE**

Free the mapping memory.

**Lab2**

Applicable mapping label:

**ALL**

Surface and volumetric mapping.

**SURF**

Surface mapping only.

**VOLU**

Volumetric mapping only.

**Filename**

The file name for a mapping data file (*filename.sur* for surface mapping and *filename.vol* for volumetric mapping). Defaults to Jobname. Applies to the commands MFMAP,SAVE and MFMAP,RESU only.

**Opt**

File format:

**BINA**

Binary file (default).

**ASCI**

ASCII file.

## Notes

This command calculates, saves, resumes, or deletes mapping data. It defaults to calculating the mapping data. If MFMAP has not been previously issued, the mapping data will be automatically calculated during the solution process. On the other hand, the ANSYS Multi-field solver solver will use previously created mapping data. Resumed mapping files must have load transfer specifications that are consistent with those of the current **MFSURFACE** and **MFVOLUME** commands and the ANSYS database.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Interface>Mapping**  
**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Interface>Mapping**

---

**MFORDER**, *FNUMB1, FNUMB2, FNUMB3, FNUMB4, FNUMB5, FNUMB6, FNUMB7, FNUMB8, FNUMB9, FNUMB10, FNUMB11, FNUMB12, FNUMB13, FNUMB14, FNUMB15, FNUMB16, FNUMB17, FNUMB18, FNUMB19, FNUMB20*

**Specifies field solution order for an ANSYS Multi-field solver analysis.**

[SOLUTION: Multi-field Global Controls](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

*FNUMB1, FNUMB2, FNUMB3, . . . , FNUMB20*

Field numbers defined by the [MFELEM](#) command .

## Notes

You can define up to twenty fields in an ANSYS Multi-field solver analysis.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Setup>Order**  
**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Setup>Order**

---

**MFOUTPUT**, *FREQ*

**Specifies results file output frequency for an ANSYS Multi-field solver analysis.**

[SOLUTION: Multi-field Time Controls](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
[Product Restrictions](#)

*FREQ*

*N*

Write solution every *N*th (and the last) time step. Defaults to 1, for every time step.



**-N**

Writes up to  $-N$  equally spaced results (for multifield auto time stepping).

**NONE**

Suppresses writing of results for all multifield time steps.

**ALL**

Writes results for every multifield time step (default).

**LAST**

Writes results for only the last multifield time step.

**%array%**

Where **%array%** is the name of an  $n \times 1 \times 1$  dimensional array parameter defining  $n$  key times, the data for the specified solution results item is written at those key times. Key times in the array parameter must appear in ascending order. Value must be greater than or equal to the ending time values for the load step.

For restart runs (see **MFRSTART** command), either change the parameter values to fall between the beginning and ending time values of the load step, or erase the current settings and reissue the command with a new array parameter.

For more information about defining array parameters, see the **\*DIM** command documentation.

## Notes

A **MFOUTPUT** setting overrides any other output frequency setting (**OUTRES**). To select the solution items, use the **OUTRES** command.

For the case of  $\text{Freq} = -n$  and  $\text{Freq} = \text{\%array\%}$ , the results at the time points which first time reaches or exceeds the targeting output time points will be written.

FLOTRAN ignores a **MFOUTPUT** setting. You should write out every time step in a FLOTRAN analysis.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Frequency**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Frequency**

**\*MFOURI**, *Oper*, *COEFF*, *MODE*, *ISYM*, *THETA*, *CURVE*

**Calculates the coefficients for, or evaluates, a Fourier series.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Oper*

Type of Fourier operation:

#### **FIT** --

Calculate Fourier coefficients *COEFF* from *MODE*, *ISYM*, *THETA*, and *CURVE*.

#### **EVAL** --

Evaluate the Fourier curve *CURVE* from *COEFF*, *MODE*, *ISYM* and *THETA*

### *COEFF*

Name of the array parameter vector containing the Fourier coefficients (calculated if *Oper* = FIT, required as input if *Oper* = EVAL). See **\*SET** for name restrictions.

### *MODE*

Name of the array parameter vector containing the mode numbers of the desired Fourier terms.

### *ISYM*

Name of the array parameter vector containing the symmetry key for the corresponding Fourier terms. The vector should contain keys for each term as follows:

#### **0** or **1** --

Symmetric (cosine) term

#### **-1** --

Antisymmetric (sine) term.

### *THETA*, *CURVE*

Names of the array parameter vectors containing the theta vs. curve description, respectively. Theta values should be input in degrees. If *Oper* = FIT, one curve value should be supplied with each theta value. If *Oper* = EVAL, one curve value will be calculated for each theta value.

## Notes

Calculates the coefficients of a Fourier series for a given curve, or evaluates the Fourier curve from the given (or previously calculated) coefficients. The lengths of the *COEFF*, *MODE*, and *ISYM* vectors must be the same--typically two times the number of modes desired, since two terms (sine and cosine) are generally required for each mode. The lengths of the *CURVE* and *THETA* vectors should be the same or the smaller of the two will be used. There should be a sufficient number of points to adequately define the curve--at least two times the number of coefficients. A starting array element number (1) must be defined for each array parameter vector. The vector specifications **\*VLEN**, **\*VCOL**, **\*VABS**, **\*VFACT**, and **\*VCUM** do not apply to this command. Array elements should not be skipped with the **\*VMASK** and the *NINC* value of the **\*VLEN** specifications. The vector being calculated (*COEFF* if *Oper* is FIT, or *CURVE* if *Oper* is EVAL) must exist as a dimensioned array [**\*DIM**].

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Array Operations>Matrix Fourier

**MFPSIMUL**, *gname*, *Fname1*, *Fname2*

Sets up a field solver group to simultaneously process with code coupling analyses.

[SOLUTION: Multi-field Global Controls](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
[Product Restrictions](#)

### *gname*

Sets the group name with a character string of up to 80 characters.

### *Fname1*, *Fname2*

Sets the field solver 1 and field solver 2 names, which are processed simultaneously, with a character string of up to 80 characters.

## Notes

This command is used to define a group of simultaneously-processed field solvers in an MFX analysis. For example, to define group *g1* with field solvers *ansys-code* and *cfx-code*, enter **MFPS ,g1 ,ansys-code ,cfx-code**.

To indicate groups of sequentially-processed field solvers for your MFX analysis, create two groups (*g1* and *g2*).

A field solver refers to a specific instance of an ANSYS or CFX solver execution that is defined by the respective input file(s) referenced when starting the solver (through the launcher or from the command line). The field solver names that are referenced in several MFX commands must be consistent with the names that will be used when starting the coupled simulation.

---

### Note

When running MFX from the launcher, you must use ANSYS and CFX (uppercase) as the field solver names (**MFPSIMUL**) in your input file.

See [Multi-field Commands](#) in the [Coupled-Field Analysis Guide](#) for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Solution Ctrl

**MFRC**, *FREQ*, *MAXFILES*

**Controls file writing for multiframe restarts for the ANSYS Multi-field solver.**

[SOLUTION: Multi-field Time Controls](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
[Product Restrictions](#)

***FREQ***

Frequency at which the .Xnnn files are written.

**0**

Do not write any .Xnnn files for this simulation

**LAST**

Write the .Xnnn files for the last multifield time step (default).

**N**

If N is a positive number, write the .Xnnn file every Nth multifield time step.

If N is a negative number write N equally spaced .Xnnn file for the entire simulation time. If auto time stepping is on, write the file at the multifield time step which first exceeds one of the equally spaced time points.

***MAXFILES***

Maximum number of .Xnnn files to save for a multifield analysis.

**0**

Do not overwrite existing .Xnnn files (default). The total maximum number of .Xnnn files for one run is 999. If this number is reached before the analysis is complete, the analysis will continue, but will no longer write .Xnnn files.

**N**

The maximum number of .Xnnn file to keep for this multifield simulation. When this number of .Rnnn have been written, ANSYS will overwrite the first .Xnnn file of the subsequent multifield time step.

**Notes**

This command sets up the restart parameters for a multiframe restart, allowing you to restart an analysis from any multifield time step for which there is a .Rnnn file. You can perform a multiframe restart only for nonlinear static and full transient structural, thermal and thermal- structural (see **RESCONTROL** for details).

If you have many multifield time steps, and are writing .Xnnn files frequently, use *MAXFILES* to limit the number of .Xnnn files saved, since these files can fill up your disk quickly.

For a CFX analysis, you must also configure the **MFOUTPUT** and **MFRSTART** settings to ensure consistent time points for postprocessing and restart simulation.

For MFX simulation, the **RESCONTROL** command will be ignored.

**Default Behavior**

In nonlinear static and full transient analyses, the default behavior is multiframe restart. (**MFRC**, LAST). By default, the .Rnnn file will be written at the last multifield time step. A .Rnnn file and corresponding load set of .ldhi files is also written at the multifield time step prior to the abort point of the run if an abort file was used or if the job terminated because of a failure to converge or some other solution error. No information at the aborted multifield time step will be saved in either the .Rnnn file or the .ldhi file.

This command cannot be issued after restarting a multifield analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

This command cannot be accessed from a menu.

---

**MFRELAX**, *Lab*, *VALUE*, *Option*

**Sets relaxation values for an ANSYS Multi-field solver analysis.**

[SOLUTION: Multi-field Convergence Controls](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
[Product Restrictions](#)

### *Lab*

Valid labels:

- Force label: FORC
- Displacement label: DISP
- Temperature label: TEMP
- Heat flux label: HFLU
- Heat generation label: HGEN

If *Lab* = ALL, **MFRELAX** applies the relaxation value to all variables.

### *VALUE*

Relaxation value. Defaults to 0.75 for all labels.

### *Option*

Valid options are:

#### **RELX**

Uses relaxation method for load transfer (default).

#### **LINT**

Uses a linear interpolation for load transfer.

## Notes

**MFRELAX** sets relaxation values for the load transfer variables at a surface or volume interface. *Option* = RELX will usually give you a more stable and smooth load transfer and is suitable for strongly coupled problems (such as FSI problems). *Option* = LINT is suitable for weakly coupled problems because it will transfer the full load in fewer stagger iterations.

See the **MFRR** and **MFITER** commands for more information on relaxation in the ANSYS Multi-field solver.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Stagger>Relaxation**  
**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Relaxation**  
**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Solution Ctrl**  
**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Stagger>Relaxation**  
**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Relaxation**  
**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Solution Ctrl**

## MFRSTART, TIME

**Specifies restart status for an ANSYS Multi-field solver analysis.**

[SOLUTION: Multi-field Time Controls](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
[Product Restrictions](#)

### TIME

Restart time

**0**

New analysis (Default)

**-1**

Restart from the last result set from a previous run.

**n**

Specify any positive number for the actual time point from which the ANSYS Multi-field solver will restart. ANSYS checks the availability of the result set and database file.

## Notes

For MFX analyses, ANSYS always passes an actual time value to CFX (zero for a new analysis or a positive value for a restart run) and CFX verifies the consistency with the initial results file. For more details about ANSYS restart capabilities, please see [Restarting an Analysis](#) in the *Basic Analysis Guide*.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Time Ctrl**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Time Ctrl**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Time Ctrl**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Time Ctrl**

## MFSORDER, *gname1*, *gname2*

**Sets up the solution sequence of simultaneous field solver groups for code coupling analyses.**

*SOLUTION: Multi-field Global Controls*

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions

*gname1*, *gname2*

Specifies the group name for groups defined by the **MFPSIMUL** command with a character string of up to 80 characters.

## Notes

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Solution Ctrl**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Solution Ctrl**

**MFSURFACE**, *INUMB*, *FNUMB1*, *Label*, *FNUMB2*

**Defines a surface load transfer for an ANSYS Multi-field solver analysis.**

**SOLUTION:** Multi-field Load Transfer

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***INUMB***

Interface number for load transfer. The interface number corresponds to the interface number specified by the surface flag FSIN (SFxxcommands).

***FNUMB1***

Field number of sending field.

***Label***

Valid surface load labels:

Force label: FORC

Displacement label: DISP

Temperature label: TEMP

Heat flux label: HFLU

***FNUMB2***

Field number for receiving field.

**Notes**

This command is also valid in PREP7.

The ANSYS Multi-field solver solver does not allow you to switch the load transfer direction for the same load quantity across the same interfaces for a restart run. For example, if Field1 sends temperature to and receives heat flow from Field2 across Interface 1 in a previous solution, then you cannot make Field1 send heat flow to and receive temperatures from Field2 across the same interface in a restart run, even if you cleared the corresponding load transfer command.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Interface>Surface**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Interface>Surface**

---



**MFTIME**, *TIME*

Sets end time for an ANSYS Multi-field solver analysis.

[SOLUTION: Multi-field Time Controls](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
[Product Restrictions](#)

**TIME**

End time of an ANSYS Multi-field solver analysis. Defaults to 1.

**Notes**

A **MFTIME** setting overrides any other end time setting (**TIME** or **FLDATA4**).

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Product Restrictions**

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

**Menu Paths**

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Time Ctrl**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Time Ctrl**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Time Ctrl**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Time Ctrl**

**MFTOL**, *Key, Value, Toler*

Turns normal distance checking on for surface mapping in an ANSYS Multi-field solver analysis.

[SOLUTION: Multi-field Interface Mapping](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
[Product Restrictions](#)

**Key**

Normal distance key

**ON**

Activates normal distance checking.

**OFF**

Deactivates normal distance checking (default).

**Value**

The normal distance tolerance for surface mapping. Defaults to 1.0e-6. If *Toler* = REL, *Value* is dimensionless. If *Toler* = ABS, *Value* has the dimensions of length.

### *Toler*

Tolerance definition key

#### **REL**

Activates relative gap tolerance, which is independent of units (default).

#### **ABS**

Activates absolute gap tolerance.

## Notes

For a dissimilar mesh interface, the nodes of one mesh are mapped to the local coordinates of an element in the other mesh. When normal distance checking is activated, the mapping tool checks the normal distance from the node to the nearest element. The node is considered improperly mapped if the normal distance exceeds the tolerance value. The mapping tool creates a component to graphically display the improperly mapped nodes. See [Mapping Diagnostics](#) in the *Coupled-Field Analysis Guide* for more information.

When using relative gap tolerance ( $Toler = REL$ ), the normal distance tolerance is derived from the product of the relative tolerance *Value* and the largest dimension of the Cartesian bounding box for a specific interface. Therefore, each interface will have a different normal distance tolerance, even though **MFTOL** is a global command.

This command is also valid in PREP7.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Product Restrictions

ANSYS Mechanical is available only for an MFX analysis. ANSYS Multiphysics is valid for both MFS and MFX analyses.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Setup>Global**

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Mapping**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Setup>Global**

**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Advanced Set Up>Mapping**

---

\***MFUN**, *ParR*, *Func*, *Par1*

**Copies or transposes an array parameter matrix.**

[APDL: Array Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *ParR*

The name of the resulting array parameter matrix. See **\*SET** for name restrictions.

**Func**

Copy or transpose function:

**COPY --**

*Par1* is copied to *ParR*

**TRAN --**

*Par1* is transposed to *ParR*. Rows (m) and columns (n) of *Par1* matrix are transposed to resulting *ParR* matrix of shape (n,m).

**Par1**

Array parameter matrix input to the operation.

**Notes**

Operates on one input array parameter matrix and produces one output array parameter matrix according to:

$$ParR = f(Par1)$$

where the function (f) is either a copy or transpose, as described above.

Functions are based on the standard FORTRAN definitions where possible. *ParR* may be the same as *Par1*. Starting array element numbers must be defined for each array parameter matrix if it does not start at the first location. For example, **\*MFUN**,A(1,5),**COPY**,B(2,3) copies matrix B (starting at element (2,3)) to matrix A (starting at element (1,5)). The diagonal corner elements for each submatrix must be defined: the upper left corner by the array starting element (on this command), the lower right corner by the current values from the **\*VCOL** and **\*VLEN** commands. The default values are the (1,1) element and the last element in the matrix. No operations progress across matrix planes (in the 3rd dimension). Absolute values and scale factors may be applied to all parameters [**\*VABS**, **\*VFACT**]. Results may be cumulative [**\*VCUM**]. Array elements should not be skipped with the **\*VMASK** and the *NINC* value of the **\*VLEN** specifications. The number of rows [**\*VLEN**] applies to the *Par1* array. See the **\*VOPER** command for details.

This command is valid in any processor.

**Menu Paths**

Utility Menu>Parameters>Array Operations>Matrix Functions

**MFVOLUME**, *INUMB*, *FNUMB1*, *Label*, *FNUMB2*

**Defines a volume load transfer for an ANSYS Multi-field solver analysis.**

SOLUTION: Multi-field Load Transfer

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**INUMB**

Interface number for load transfer. The interface number corresponds to the interface number specified by the volume flag *FVIN* (**BFE** command).

**FNUMB1**

Field number of sending field.

**Label**

Valid volume load labels:

Force label: FORC

Displacement label: DISP

Temperature label: TEMP

Heat generation: HGEN

**FNUMB2**

Field number for receiving field.

**Notes**

This command is also valid in PREP7.

The ANSYS Multi-field solver does not allow you to switch the load transfer direction for the same load quantity across the same interfaces for a restart run. For example, if Field1 sends temperature to and receives heat flow from Field2 across Interface 1 in a previous solution, then you cannot make Field1 send heat flow to and receive temperatures from Field2 across the same interface in a restart run, even if you cleared the corresponding load transfer command.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Multi-field Set Up>MFS-Single Code>Interface>Volume**

**Main Menu>Solution>Multi-field Set Up>MFS-Single Code>Interface>Volume**

---

**MFWRITE, Fname, Ext**

**Writes an ANSYS master input file for MFX multiple code coupling.**

[SOLUTION: Multi-field Global Controls](#)

[MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions](#)

**Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

*Fname* defaults to Jobname.

**Ext**

Filename extension (8 character maximum).

*Ext* defaults to dat.

## Notes

When working interactively, you need to issue this command as the last step in your setup process. This command will write out the input file that you will then use to submit the MFX analysis. This file will include the **/SOLU**, **SOLVE**, and **FINISH** commands.

See [Multi-field Commands](#) in the *Coupled-Field Analysis Guide* for a list of all ANSYS Multi-field solver commands and their availability for MFS and MFX analyses.

## Product Restrictions

ANSYS Mechanical allows MFX analyses but not MFS analyses. ANSYS Multiphysics allows both MFS and MFX analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Multi-field Set Up>MFX-ANSYS/CFX>Write input**  
**Main Menu>Solution>Multi-field Set Up>MFX-ANSYS/CFX>Write input**

## MGEN, *ITIME*, *INC*, *NODE1*, *NODE2*, *NINC*

**Generates additional MDOF from a previously defined set.**

**SOLUTION:** Master DOF

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *ITIME*, *INC*

Do this generation operation a total of *ITIME*s, incrementing all nodes in the set by *INC* each time after the first. *ITIME* must be > 1 for generation to occur. All previously defined master degree of freedom directions are included in the set. A component name may also be substituted for *ITIME*.

### *NODE1*, *NODE2*, *NINC*

Generate master degrees of freedom from set beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and set is all selected nodes [**NSSEL**]. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

## Notes

Generates additional master degrees of freedom from a previously defined set. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Master DOFs>User Selected>Copy**  
**Main Menu>Solution>Master DOFs>User Selected>Copy**

**MIDTOL**, *KEY*, *TOLERB*, *RESFQ*

**Sets midstep residual criterion values for structural transient analyses.**

**SOLUTION:** Dynamic Options

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

**KEY**

Midstep residual criterion activation key.

**ON or 1**

Activate midstep residual criterion in a structural transient analysis (default).

**OFF or 0**

Deactivate midstep residual criterion in a structural transient analysis.

**STAT**

List the current midstep residual criterion setting.

**TOLERB**

Midstep residual tolerance or reference value for *bisection*. Defaults to 100 times the *TOLER* setting of the **CNVTOL** command.

If *TOLERB* > 0, it is used as a tolerance about the typical force and/or moment to compare midstep residual force and/or moment for convergence.

If *TOLERB* < 0, it is used as a reference force value against which the midstep residual force is compared for convergence. The reference force value is used to compute a reference moment value for midstep residual moment comparison.

If midstep residual force and/or moment has not converged and **AUTOTS,ON** is used, then *TOLERB* is also used to predict time step size for bisection.

**RESFQ**

Key to use response frequency computation along with midstep residual criterion for automatic time stepping (**AUTOTS,ON**).

**OFF or 0**

Do not calculate response frequency and do not consider it in the automatic time stepping (default).

**ON or 1**

Calculate response frequency and consider it in the automatic time stepping.

**Command Default**

For transient structural analysis, the out-of-balance residual is not checked at the midstep.

**Notes**

When *TOLERB* is input as a tolerance value (*TOLERB* > 0), the typical force and/or moment from the regular time step is used in the midstep residual force and/or moment comparison.

In a structural transient analysis, the suggested tolerance range of *TOLERB* (*TOLERB* > 0) is as follows:

*TOLERB* = 1 to 10 times the *TOLER* setting of the **CNVTOL** command for high accuracy solution.

*TOLERB* = 10 to 100 times the *TOLER* setting of the **CNVTOL** command for medium accuracy solution.

*TOLERB* = more than 100 times the *TOLER* setting of the **CNVTOL** command for low accuracy solution.

If the structural transient analysis is elastic and linear, and the load is constant or changes slowly, use a smaller value of *TOLERB* to achieve an accurate solution. If the analysis involves large amounts of energy dissipation, such as elastic-plastic material, *TOLERB* can be larger. If the analysis includes contact or rapidly varying loads, a smaller value of *TOLERB* should be used if high frequency response is important; otherwise, a larger value of *TOLERB* may be used to enable faster convergence with larger time step sizes.

For more information on how the midstep criterion is used by the program, see [Midstep Residual for Structural Dynamic Analysis](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Transient**

**/MKDIR**, *Dir*

**Creates a directory.**

APDL: Macro Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

*Dir*

The directory to create (248 characters maximum on Unix and Linux; 233 on Windows). If no path is provided, it will be created in the current working directory. Must be a valid name (and path) for the system you are working on.

## Notes

Creates a directory on the computer ANSYS is currently running on.

## Menu Paths

**This command cannot be accessed from a menu.**

**MLIST**, *NODE1*, *NODE2*, *NINC*

**Lists the MDOF of freedom.**

SOLUTION: Master DOF

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

*NODE1*, *NODE2*, *NINC*

List master degrees of freedom from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL (default), *NODE2* and *NINC* are ignored and masters for all selected nodes [**NSEL**] are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored

(valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

## Notes

Lists the master degrees of freedom. Master degrees of freedom generated from the **TOTAL** command cannot be listed until after the first load step.

## Menu Paths

**Main Menu>Preprocessor>Loads>Master DOFs>User Selected>List All**

**Main Menu>Preprocessor>Loads>Master DOFs>User Selected>List Picked**

**Main Menu>Solution>Master DOFs>User Selected>List All**

**Main Menu>Solution>Master DOFs>User Selected>List Picked**

**Utility Menu>List>Other>Master DOF>At All Nodes**

**Utility Menu>List>Other>Master DOF>At Picked Nodes**

## MMASS, Option, ZPA

**Specifies the missing mass response calculation.**

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### Option

Flag to activate or deactivate missing mass response calculation.

#### 1 (ON or YES)

Activate.

#### 0 (OFF or NO)

Deactivate (this is the default value)

### ZPA

Zero Period Acceleration Value. If a scale factor *FACT* is defined in the *SVTYP* command, it applies to this value.

## Notes

For more information on spectrum analysis combination methods, see [Combination of Modes](#)

Missing mass calculation is valid only for single point excitation response spectrum analysis (**SPOPT**, *SPRS*) and for multiple point response spectrum analysis (**SPOPT**, *MPRS*) performed with base excitation using acceleration response spectrum loading. Missing mass is supported in a spectrum analysis only when the preceding modal analysis is performed with Block Lanczos, PCG Lanczos or Supernode eigensolver (Method =LANB, LANPCG, or SNODE on the **MODOPT** command). Velocity and acceleration solutions are not available (*Label=VELO* or *ACEL* on the combination command: **SRSS**, **CQC**...).

This command is also valid in PREP7.

## Menu Paths

Menu paths generated with the final build.



## MMF

Calculates the magnetomotive force along a path.

[POST1: Magnetics Calculations](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

## Notes

**MMF** invokes an ANSYS macro which calculates the magnetomotive force (mmf) along a predefined path [**PATH**]. It is valid for both 2-D and 3-D magnetic field analyses. The calculated mmf value is stored in the parameter **MMF**.

A closed path [**PATH**], passing through the magnetic circuit for which mmf is to be calculated, must be defined before this command is issued. A counterclockwise ordering of points on the **PPATH** command will yield the correct sign on the mmf. The mmf is based on Ampere's Law. The macro makes use of calculated values of field intensity (H), and uses path operations for the calculations. All path items are cleared upon completion. The **MMF** macro sets the "ACCURATE" mapping method and "MAT" discontinuity option of the **PMAP** command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>MMF

**MODCONT**, *MLSkey*, *EnforcedKey*, *IgnoreThermalStrain*

Specify additional modal analysis options.

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN DS DSS <> <> EH <> PP <> EME MFS

### *MLSkey*

Multiple Load Step Key:

#### **OFF**

Perform the modal analysis for each load step. (default)

#### **ON**

Perform the modal analysis only for the first load step; form the load vector for each load step and write it to the `Jobname.MODE` file for downstream mode superposition analyses.

### *EnforcedKey*

Enforced Motion Key:

#### **OFF**

Do not calculate enforced static modes. (default)

#### **ON**

Calculate enforced static modes and write them to the `Jobname.MODE` file.

### *IgnoreThermalStrain*

Key to ignore thermal strains in the load vector. Only available if *MLSkey*=ON.

**OFF**

Include any thermal strains in the modal load vector. (default)

**ON**

Ignore any thermal strains.

## Notes

Specifies additional modal analysis (**ANTYPE**,**MODAL**) options.

Temperatures applied in the modal analysis are used by default to evaluate material properties and contribute to the load vector if the temperature does not equal the reference temperature and a coefficient of thermal expansion is specified. Use the *IgnoreThermalStrain* key if you wish to evaluate the material properties but not contribute to the load vector. This is particularly useful when performing a downstream harmonic analysis where you do not want to include harmonically varying thermal loads.

Use the **LVSCALE** command to apply the desired load in a mode superposition transient or harmonic analysis.

Multiple load steps in **MODCONT** are not supported for the **PSOLVE** command.

This command applies only to the Block Lanczos, PCG Lanczos, and Supernode modal methods.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## **MODE**, *MODE*, *ISYM*

**Specifies the harmonic loading term for this load step.**

[SOLUTION: Load Step Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**MODE**

Number of harmonic waves around circumference for this harmonic loading term (defaults to 0).

**ISYM**

Symmetry condition for this harmonic loading term (not used when *MODE* = 0):

**1**

Symmetric (UX, UY, ROTZ, TEMP use cosine terms; UZ uses sine term) (default).

**-1**

Antisymmetric (UX, UY, ROTZ, temp use sine terms; UZ uses cosine term).

## Command Default

*MODE* = 0, *ISYM* = 1.

## Notes

Used with axisymmetric elements having nonaxisymmetric loading capability (e.g., [PLANE25](#), [SHELL61](#), [FLUID81](#), etc.). For analysis types [ANTYPE](#), MODAL, HARMIC, TRANS, and SUBSTR, the term must be defined in the first load step and may not be changed in succeeding load steps.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>For Harmonic Ele**  
**Main Menu>Solution>Load Step Opts>Other>For Harmonic Ele**

## MODIFY, SET, LSTEP, ITER, CUMIT, TIME, Ktitle

Changes the listed values of the data in a set.

[AUX3: Results Files](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**SET**  
 Set of data in results file to be modified.

**LSTEP**  
 The new load step number.

**ITER**  
 The new load substep number.

**CUMIT**  
 The new cumulative iteration.

**TIME**  
 The new time/frequency value.

**Ktitle**  
 Indicates if the set title should be modified.

**0**  
 Keep the original title.

**1**  
 Change the title to the title specified with the most current [/TITLE](#) command.

## Notes

Use this command to change the listed values in a data set in a results file. Using this command does not change any actual model data; it affects only the values listed in the results file.

For example, if you start with the following results file:

```

SET    TIME/FREQ    LOAD STEP    SUBSTEP    CUMULATIVE
1      1.0000       1            1          1
first load set

2      2.0000       2            1          2
second load set
    
```

```
3      3.0000      3      1      3
third load set

4      4.0000      4      1      4
fourth load set
```

and you then issue the following commands:

```
/title, modified title for set number 3
modify,3,5,2,5,4.5,1
```

The modified results file would look like this:

```
SET    TIME/FREQ    LOAD STEP    SUBSTEP    CUMULATIVE
1      1.0000      1           1          1
first load set

2      2.0000      2           1          2
second load set

3      4.5000      5           2          5
modified title for set number 3

4      4.0000      4           1          4
fourth load set
```

## Menu Paths

**Main Menu>Preprocessor>Loads>Other>For Harmonic Ele**  
**Main Menu>Solution>Other>For Harmonic Ele**

---

## MODMSH, Lab

Controls the relationship of the solid model and the **FE** model.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Lab

Relationship key:

#### STAT

Gives status of command (default). This applies only to the CHECK option (no status is provided for the DETACH option).

#### NOCHECK

Deactivates the checking of the solid model and the finite element model. Allows elements and nodes generated with the mesh commands to be modified directly (**EMODIF**, **NMODIF**, **EDELE**, **NDELE**, etc.). Also deactivates solid model hierarchical checking so that areas attached to volumes may be deleted etc.

#### CHECK

Reactivates future checking of the solid model.

#### DETACH

Releases all associativity between the current solid model and finite element model. ANSYS deletes any element attributes that were assigned to the affected solid model entities through default attributes (that is, through the **TYPE**, **REAL**, **MAT**, **SECNUM**, and **ESYS** command settings and a subsequent

meshing operation). However, attributes that were assigned directly to the solid model entities (via the **KATT**, **LATT**, **AATT**, and **VATT** commands) are not deleted.

## Notes

Affects the relationship of the solid model (keypoints, lines, areas, volumes) and the finite element model (nodes, elements, and boundary conditions).

Specify *Lab* = NOCHECK carefully. By deactivating checking, the solid model database can be corrupted by subsequent operations.

After specifying *Lab* = DETACH, it is no longer possible to select or define finite element model items in terms of the detached solid model or to clear the mesh.

## Menu Paths

**Main Menu>Preprocessor>Checking Ctrl's>Model Checking**

**MODOPT**, *Method*, *NMODE*, *FREQB*, *FREQE*, *Cpxmod/PRMODE*, *Nrmkey*, *--*, *BlockSize*

**Specifies modal analysis options.**

[SOLUTION: Dynamic Options](#)

[MP ME ST PR PRN DS DSS <> <> EH <> PP <> EME MFS](#)  
[Product Restrictions](#)

### *Method*

Mode-extraction method to be used for the modal analysis.

#### **LANB**

Block Lanczos

#### **LANPCG**

PCG Lanczos

#### **SNODE**

Supernode modal solver

#### **REDUC**

Householder (reduced)

#### **UNSYM**

Unsymmetric matrix

#### **DAMP**

Damped system

#### **QRDAMP**

Damped system using QR algorithm

#### **VT**

Solve with the Variational Technology method of ANSYS DesignXplorer

### *NMODE*

The number of modes to extract. The value can depend on the value supplied for *Method*. Defaults to the number of master DOFs when *Method* = REDUC. For the other methods, *NMODE* has no default and

must be specified. If *Method* = LANB, LANPCG, or SNODE, the number of modes that can be extracted can equal the DOFs in the model after the application of all boundary conditions.

*Recommendation:*

When *Method* = REDUC, *NMODE* should be less than half of the number of master DOFs.

When *Method* = LANPCG, *NMODE* should be less than 100 to be computationally efficient.

When *Method* = SNODE, *NMODE* should be greater than 100 for 2-D plane or 3-D shell/beam models and greater than 250 for 3-D solid elements to be computationally efficient.

**FREQB**

The beginning, or lower end, of the frequency range of interest.

For *Method* = LANB, UNSYM, DAMP, and QRDAMP, *FREQB* also represents the first shift point for the eigenvalue iterations. If values for UNSYM or DAMP are zero or blank, the default value is -1.0. For the other methods, the default is internally computed. Eigenvalue extraction is most accurate near the shift point; multiple shift points are used internally in the LANB and QRDAMP methods. For LANB, LANPCG, UNSYM, DAMP, and QRDAMP methods with a positive *FREQB*, eigenvalues are output beginning at the shift point and increase in magnitude. For UNSYM and DAMP methods with a negative *FREQB* value, eigenvalues are output beginning at zero magnitude and increase.

Choosing higher *FREQB* values with the LANPCG and SNODE methods may lead to inefficient solution times because these methods will find all eigenvalues between zero and *FREQB* before finding the requested modes between *FREQB* and *FREQE*.

**FREQE**

The ending, or upper end, of the frequency range of interest (in Hz). The default for *Method* = SNODE is described below. The default for all other methods is to calculate all modes, regardless of their maximum frequency.

The default is 100 Hz for *Method* = SNODE. To maintain solution efficiency, you should not set the *FREQE* value too high; for example, not higher than 5000 Hz for an industrial problem. The higher the *FREQE* value used for the SNODE method, the more solution time it will take and the more eigenvalues it could produce. For example, if *FREQE* is set to 1e8, it will cause the underlying supernodal structures to find all the possible eigenvalues of each group of supernodes; hence, it will take an excessive amount of solution time.

**Cpxmod/PRMODE**

*CPXMOD* (Valid only when *Method* = QRDAMP).

**ON**

Calculate complex eigenmode shapes.

**OFF**

Do not calculate complex eigenmode shapes. This is the default.

*PRMODE*

The number of reduced modes to print. Valid only when *Method* = REDUC.

**Nrmkey**

Mode shape normalization key:

**OFF**

Normalize the mode shapes to the mass matrix (default).

**ON**

Normalize the mode shapes to unity instead of to the mass matrix. If a subsequent spectrum or mode superposition analysis is planned, the mode shapes should be normalized to the mass matrix (*Nrmkey* = OFF).

--

Unused field.

**BlockSize**

The block vector size to be used with the Block Lanczos eigensolver (used only when *Method* = LANB). *BlockSize* must be an integer value between 0 and 16. When *BlockSize* = zero or blank, the code decides the block size internally (normally, a value of 8 is used). Typically, higher *BlockSize* values are more efficient under each of the following conditions:

- When running in out-of-core mode and there is not enough physical memory to buffer all of the files written by the Block Lanczos eigensolver (and thus, the time spent doing I/O is considerable).
- Many modes are requested (>100).
- Higher-order solid elements dominate the model.

The memory usage only slightly increases as *BlockSize* is increased. It is recommended that you use a value divisible by 4 (4, 8, 12, or 16).

**Notes**

Specifies modal analysis (**ANTYPE**,MODAL) options. Additional options used only for the Supernode (SNODE) eigensolver are specified by the **SNOPTION** command. If *Method* = LANPCG, ANSYS automatically switches to the PCG solver internally for this modal analysis. You can further control the efficiency of the PCG solver with the **PCGOPT** and **EQSLV** commands.

The Block Lanczos method is strongly recommended for high-frequency magnetic eigenvalue problems. The initial frequency guess is not critical. The ratio of *FREQE* to *FREQB* can be up to 1e6. The PCG Lanczos method is not supported for high-frequency magnetic eigenvalue problems.

For models that involve a non-symmetric element stiffness matrix, as in the case of a contact element with frictional contact, the QR damp eigensolver (**MODOPT**, QRDAMP) extracts modes in the modal subspace formed by the eigenmodes from the symmetrized eigenproblem. The QR damp eigensolver symmetrizes the element stiffness matrix on the first pass of the eigensolution, and in the second pass, eigenmodes are extracted in the modal subspace of the first eigensolution pass. For such non-symmetric eigenproblems, you should verify the eigenvalue and eigenmode results using the non-symmetric matrix eigensolver (**MODOPT**, UNSYM ).

The UNSYM, DAMP, and QRDAMP options cannot be followed by a subsequent spectrum analysis.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** All extraction methods, except VT, are supported within Distributed ANSYS. However, PCG Lanczos, UNSYM, and DAMP are the only distributed eigensolvers that will run a fully distributed solution. The Block Lanczos and Supernode eigensolvers are not distributed eigensolvers; therefore, you will not see the full performance improvements with these methods that you would with a fully distributed solution. The REDUC and QRDAMP methods are supported, but do not use distributed memory parallelism within Distributed ANSYS.

## Product Restrictions

Command Option Method	Available Products
LANB	MP ME ST PR PRN DS DSS <> <> EH <> PP <> EME MFS [1]
LANPCG	MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS [1]
SNODE	MP ME ST PR PRN <> <> <> <> EH <> PP <> EME MFS
REDUC	MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS [1]
UNSYM	MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS
DAMP	MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS
QRDAMP	MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS
VT	<> <> <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

1. The ANSYS DesignSpace (DS) and ANSYS DesignSpace - Structural (DSS) products do not support distributed solutions (Distributed ANSYS).

## Menu Paths

**Main Menu>DesignXplorer>Solution>Solve**

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>Analysis Options**

## MONITOR, VAR, Node, Lab

**Controls contents of three variable fields in nonlinear solution monitor file.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

### VAR

One of three variable field numbers in the monitor file whose contents can be specified by the Lab field. Valid arguments are integers 1, 2, or 3. See Notes section for default values.

### Node

The node number for which information is monitored in the specified VAR field. In the GUI, if Node = P, graphical picking is enabled. If blank, the monitor file lists the maximum value of the specified quantity (Lab field) for the entire structure.

### Lab

The solution quantity to be monitored in the specified VAR field. Valid labels for solution quantities are UX, UY, and UZ (displacements); ROTX, ROTY, and ROTZ (rotations); and TEMP (temperature). Valid labels for reaction force are FX, FY, and FZ (structural force) and MX, MY, and MZ (structural moment). Valid label for heat flow rate is HEAT. For defaults see the Notes section.

## Notes

This command is only active when **SOLCONTROL,ON**.



The monitor file is an ASCII file which is automatically created and saved when **SOLCONTROL** is active (ON). The monitor file always has an extension of **.mntr**, and takes its file name from the specified `Jobname`. If no `Jobname` is specified, the file name defaults to `file`.

You must issue this command once for each solution quantity you want to monitor at a specified node at each load step. You cannot monitor a reaction force during a linear analysis. The variable field contents can be redefined at each load step by reissuing the command. The monitored quantities are appended to the file for each load step.

---

### Note

Reaction forces reported in the monitor file may be incorrect if the degree of freedom of the specified node is involved in externally defined coupling (**CP** command) or constraint equations (**CE** command), or if the program has applied constraint equations internally to the node.

The following example shows the format of a monitor file. Note that the file only records the solution substep history when a substep is convergent.

SOLUTION HISTORY INFORMATION FOR JOB: file.mntr

LOAD STEP	SUB-STEP	NO. ATMP	NO. ITER	TOTL ITER	INCREMENT TIME/LFACT	TOTAL TIME/LFACT	VARIAB 1 MONITOR	VARIAB 2 MONITOR	VARIAB 3 MONITOR
							UZ	MZ	MxP1
1	1	1	3	3	0.25000	0.25000	1.4145	0.19076E-06	0.78886E-30
1	2	1	2	5	0.25000	0.50000	2.8283	0.92989E-06	0.78886E-30
1	3	1	2	7	0.37500	0.87500	4.9467	0.33342E-05	0.78886E-30
1	4	1	2	9	0.12500	1.0000	5.6519	0.16826E-05	0.78886E-30
2	1	1	6	15	0.20000E-03	1.0002	4.2198	515.23	0.78886E-30
2	2	2	6	26	0.10000E-03	1.0003	4.4849	593.03	0.78886E-30
2	3	1	3	29	0.10000E-03	1.0004	4.7531	611.45	0.78886E-30
2	4	1	3	32	0.15000E-03	1.0006	5.0696	621.83	0.78886E-30
2	5	1	4	36	0.22500E-03	1.0008	5.4428	628.42	0.78886E-30
2	6	1	4	40	0.33750E-03	1.0011	5.8928	632.78	0.78886E-30
2	7	1	5	45	0.50625E-03	1.0016	6.4454	635.62	0.78886E-30
2	8	1	7	52	0.75938E-03	1.0024	7.1375	637.22	0.78886E-30
2	9	1	5	57	0.75938E-03	1.0031	7.7422	637.66	0.78886E-30
2	10	1	6	63	0.11391E-02	1.0043	8.5588	637.42	0.78886E-30
2	11	2	3	72	0.76887E-03	1.0050	9.0721	636.96	0.78886E-30
2	12	1	3	75	0.76887E-03	1.0058	9.5648	636.35	0.78886E-30
2	13	1	3	78	0.11533E-02	1.0070	10.277	635.25	0.78886E-30
2	14	1	4	82	0.17300E-02	1.0087	11.306	633.37	0.78886E-30
2	15	1	6	88	0.25949E-02	1.0113	12.802	630.21	0.78886E-30
2	16	1	5	93	0.25949E-02	1.0139	14.273	626.81	0.78886E-30
2	17	1	7	100	0.38924E-02	1.0178	16.477	621.42	0.78886E-30
2	18	1	6	106	0.38924E-02	1.0217	18.704	615.77	0.78886E-30
2	19	2	4	116	0.26274E-02	1.0243	20.229	611.83	0.78886E-30
2	20	1	4	120	0.26274E-02	1.0269	21.777	607.80	0.78886E-30

The following details the contents of the various fields in the monitor file:

#### LOAD STEP

The current load step number.

#### SUBSTEP

The current substep (time step) number.

#### NO. ATTEMPT

The number of attempts made in solving the current substep. This number is equal to the number of failed attempts (bisections) plus one (the successful attempt).

**NO. ITER**

The number of iterations used by the last successful attempt.

**TOTL. ITER**

Total cumulative number of iterations (including each iteration used by a bisection).

**INCREMENT**

**TIME/LFACT**

Time or load factor increments for the current substep.

**TOTAL TIME/LFACT**

Total time (or load factor) for the last successful attempt in the current substep.

**VARIAB 1**

Variable field 1. In this example, the field is reporting the UZ value. By default, this field lists the CPU time used up to (but not including) the current substep.

**VARIAB 2**

Variable field 2. In this example, the field is reporting the MZ value. By default, this field lists the maximum displacement in the entire structure.

**VARIAB 3**

Variable field 3. By default (and in the example), this field reports the maximum equivalent plastic strain in the entire structure.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Monitor**

**Main Menu>Solution>Load Step Opts>Nonlinear>Monitor**

---

**\*MOPER**, *ParR*, *Par1*, *Oper*, *Par2*, *Par3*, *kDim*, --, *kOut*, *LIMIT*

**Performs matrix operations on array parameter matrices.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

***ParR***

The name of the resulting array parameter matrix. See **\*SET** for name restrictions.

***Par1***

First array parameter matrix input to the operation. For *Oper* = MAP, this is an  $N \times 3$  array of coordinate locations at which to interpolate. *ParR* will then be an  $N(out) \times M$  array containing the interpolated values.

***Oper***

Matrix operations:

**INVERT --**

(**\*MOPER**, *ParR*, *Par1*, INVERT)

Square matrix invert: Inverts the  $n \times n$  matrix in *Par1* into *ParR*. The matrix must be well conditioned.

**Warning**

Non-independent or ill-conditioned equations can cause erroneous results.

For large matrices, use the APDL Math operation **\*LSFACTOR** for efficiency (see "APDL Math").

**MULT --**

(\*MOPER, *ParR*, *Par1*, MULT, *Par2*)

Matrix multiply: Multiplies *Par1* by *Par2*. The number of rows of *Par2* must equal the number of columns of *Par1* for the operation. If *Par2* is input with a number of rows greater than the number of columns of *Par1*, matrices are still multiplied. However, the operation only uses a number of rows of *Par2* equal to the number of columns of *Par1*.

**COVAR --**

(\*MOPER, *ParR*, *Par1*, COVAR, *Par2*)

Covariance: The measure of association between two columns of the input matrix (*Par1*). *Par1*, of size  $m$  runs (rows) by  $n$  data (columns) is first processed to produce a row vector containing the mean of each column which is transposed to a column vector (*Par2*) of  $n$  array elements. The *Par1* and *Par2* operation then produces a resulting  $n \times n$  matrix (*ParR*) of covariances (with the variances as the diagonal terms).

**CORR --**

(\*MOPER, *ParR*, *Par1*, CORR, *Par2*)

Correlation: The correlation coefficient between two variables. The input matrix (*Par1*), of size  $m$  runs (rows) by  $n$  data (columns), is first processed to produce a row vector containing the mean of each column which is then transposed to a column vector (*Par2*) of  $n$  array elements. The *Par1* and *Par2* operation then produces a resulting  $n \times n$  matrix (*ParR*) of correlation coefficients (with a value of 1.0 for the diagonal terms).

**SOLV --**

(\*MOPER, *ParR*, *Par1*, SOLV, *Par2*)

Solution of simultaneous equations: Solves the set of  $n$  equations of  $n$  terms of the form  $a_{n1}x_1 + a_{n2}x_2 + \dots + a_{nn}x_n = b_n$  where *Par1* contains the matrix of a-coefficients, *Par2* the vector(s) of b-values, and *ParR* the vector(s) of x-results. *Par1* must be a square matrix. The equations must be linear, independent, and well conditioned.

**Warning**

Non-independent or ill-conditioned equations can cause erroneous results.

For large matrices, use the APDL Math operation **\*LSFACTOR** for efficiency (see "APDL Math").

**SORT --**

(\*MOPER, *ParR*, *Par1*, SORT, *Par2*,  $n1$ ,  $n2$ ,  $n3$ )

Matrix sort: Sorts matrix *Par1* according to sort vector *Par2* and places the result back in *Par1*. Rows of *Par1* are moved to the corresponding positions indicated by the values of *Par2*. *Par2*

may be a column of *Par1* (in which case it will also be reordered). Alternatively, you may specify the column of *Par1* to sort using *n1* (leaving *Par2* blank). A secondary sort can be specified by column *n2*, and a third sort using *n3*. *ParR* is the vector of initial row positions (the permutation vector). Sorting *Par1* according to *ParR* should reproduce the initial ordering.

**NNEAR --**

(\*MOPER, *ParR*, *Par1*, NNEAR, Toler)

Nearest Node: Quickly determine all the nodes within a specified tolerance of a given array.

*ParR* is a vector of the nearest selected nodes, or 0 if no nodes are nearer than Toler. *Par1* is the  $n \times 3$  array of coordinate locations. Toler defaults to 1 and is limited to the maximum model size.

**ENEAR --**

(\*MOPER, *ParR*, *Par1*, ENEAR, Toler)

Nearest Element: Quickly determine the elements with centroids that are within a specified tolerance of the points in a given array.

*ParR* is a vector of the nearest selected elements, or 0 if no element centroids are nearer than Toler. *Par1* is the  $n \times 3$  array of coordinate locations.

**MAP --**

(\*MOPER, *ParR*, *Par1*, MAP, *Par2*, *Par3*, *kDim*, --, *kOut*, LIMIT)

Maps the results from another program onto your ANSYS finite element model. For example, you can map pressures from a CFD analysis onto your model for a structural analysis.

When you map results, the subsequent *Par2* and *Par3* arguments define your input values and their locations, and the arguments that follow determine the search area and interpolation schemes (see below).

For *Oper* = MAP, output points are incorrect if they are not within the boundaries (area or volume) set via the specified input points. Also, calculations for out-of-bound points require much more processing time than do points that are within bounds.

When mapping results from one analysis to another (*Oper* = MAP), *Par1* will be your final  $N(out) \times 3$  array of points. *Par2* will be an  $N(in) \times M$  array that corresponds to the points in *Par3*. For each point in the destination mesh, all possible triangles in the source mesh are searched to find the best triangle containing each point. It then does a linear interpolation inside this triangle. You should carefully specify your interpolation method and search criteria in order to provide faster and more accurate results (see LIMIT, below). Results mapping (*Oper* = MAP) is available from the command line only.

***Par2***

Second array parameter matrix input to the operation. For the COVAR and CORR operations, this parameter must exist as a dimensioned array vector without specified values since its values (means) will be calculated as part of the operations. For MAP, this will be an  $\{N(in) \times M\}$  array of values to be interpolated, where  $N(in)$  is the number of points to interpolate from, and  $M$  is the number of values at each point. For the ENEAR and NNEAR operations, this parameter specifies the tolerance for the search.

***Par3***

Third array parameter, used for *Oper* = MAP. This is an  $N \times 3$  array of coordinate locations corresponding to the values in *Par2*.

***kDim***

Interpolation criteria; used for *Oper* = MAP:

If *kDim* = 2 or 0, two dimensional interpolation is applied (interpolate on a surface).

If *kDim* = 3, three dimensional interpolation is applied (interpolate on a volume).

--

Unused field

***kOut***

Outside region results; used for *Oper* = MAP

If *kOut* = 0, use the value(s) of the nearest region point for points outside of the region.

If *kOut* = 1, set results extrapolated outside of the region to zero.

***LIMIT***

Number of nearby nodes considered for interpolation (*Oper* = MAP). Minimum = 5, default = 20. Lower values will reduce processing time, however, some distorted or irregular meshes will require a higher *LIMIT* value to encounter three nodes for triangulation.

## Notes

Each starting array element number must be defined for each array parameter matrix if it does not start at the first location. For example, **\*MOPER,A(2,3),B(1,4),MULT,C(1,5)** multiplies submatrix B (starting at element (1,4)) by submatrix C (starting at element (1,5)) and puts the result in matrix A (starting at element (2,3)).

The diagonal corner elements for each submatrix must be defined: the upper left corner by the array starting element (on this command), the lower right corner by the current values from the **\*VCOL** and **\*VLEN** commands. The default values are the (1,1) element and the last element in the matrix. No operations progress across matrix planes (in the 3rd dimension). Absolute values and scale factors may be applied to all parameters [**\*VABS**, **\*VFACT**]. Results may be cumulative [**\*VCUM**]. Array elements should not be skipped with the **\*VMASK** and the *NINC* value of the **\*VLEN** specifications. See the **\*VOPER** command for details.

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Array Operations>Matrix Operations

## MOPT, *Lab*, *Value*

Specifies meshing options.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Lab***

Meshing option to be specified (determines the meaning of *Value*):

**AORDER**

Mesh by ascending area size order. Set *Value* to ON to mesh smaller areas first. Using this results in finer meshes in critical areas for volume meshes; this can be used for cases where **SMRTSIZE** does not mesh as needed. Default is OFF.

**EXPND**

Area mesh expansion (or contraction) option. (This option is the same as **SMRTSIZE,,,EXPND**.) This option is used to size internal elements in an area based on the size of the elements on the area's boundaries.

*Value* is the expansion (or contraction) factor. For example, issuing **MOPT,EXPND,2** before meshing an area will allow a mesh with elements that are approximately twice as large in the interior of an area as they are on the boundary. If *Value* is less than 1, a mesh with smaller elements on the interior of the area will be allowed. *Value* for this option should be greater than 0.5 but less than 4.

*Value* defaults to 1, which does not allow expansion or contraction of internal element sizes (except when using **AESIZE** sizing). If *Value* = 0, the default value of 1 will be used. The actual size of the internal elements will also depend on the **TRANS** option (or upon **AESIZE** or **ESIZE** sizing, if used).

**TETEXPND**

Tet-mesh expansion (or contraction) option. This option is used to size internal elements in a volume based on the size of the elements on the volume's boundaries.

*Value* is the expansion (or contraction) factor. For example, issuing **MOPT,TETEXPND,2** before meshing a volume will allow a mesh with elements that are approximately twice as large in the interior of the volume as they are on the boundary. If *Value* is less than 1, a mesh with smaller elements on the interior of the volume will be allowed. *Value* for this option should be greater than 0.1 but less than 3.

*Value* defaults to 1, which does not allow expansion or contraction of internal element sizes. If *Value* = 0, the default value of 1 will be used. If *Value* is greater than 2, mesher robustness may be affected.

The **TETEXPND** option is supported for both the **VMESH** and **FVMESH** commands. Tet-mesh expansion is the only mesh control supported by **FVMESH**.

**TRANS**

Mesh-transition option. Controls how rapidly elements are permitted to change in size from the boundary to the interior of an area. (This option performs the same operation as **SMRTSIZE,,,,TRANS**.)

*Value* is the transitioning factor. *Value* defaults to 2.0, which permits elements to approximately double in size as they approach the interior of the area. (If *Value* = 0, the default value of 2 will be used.) *Value* must be greater than 1 and, for best results, should be less than 4. The actual size of the internal elements will also depend on the **EXPND** option (or upon **AESIZE** or **ESIZE** sizing, if used).

For a quad mesh with any element size, this option has no effect, as the program strictly respects any face size to ensure the most uniform quad mesh possible. To obtain a graded mesh using this option, apply **LESIZE** to the lines of the desired face.

**AMESH**

Triangle surface-meshing option. Valid inputs for *Value* are:

**DEFAULT**

Allows the program to choose which triangle mesher to use. In most cases, the program chooses the main triangle mesher, which is the Riemann space mesher. If the chosen mesher fails for any reason, the program invokes the alternate mesher and retries the meshing operation.

**MAIN**

The program uses the main triangle mesher (Riemann space mesher), and it does not invoke an alternate mesher if the main mesher fails. The Riemann space mesher is well suited for most surfaces.

**ALTERNATE**

The program uses the first alternate triangle mesher (3-D tri-mesher), and it does not invoke another mesher if this mesher fails. This option is not recommended due to speed considerations. However, for surfaces with degeneracies in parametric space, this mesher often provides the best results.

**ALT2**

The program uses the second alternate triangle mesher (2-D parametric space mesher), and it does not invoke another mesher if this mesher fails. This option is not recommended for use on surfaces with degeneracies (spheres, cones, etc.) or poorly parameterized surfaces because poor meshes may result.

**QMESH**

Quadrilateral surface meshing option. (Quadrilateral surface meshes will differ based on which triangle surface mesher is selected. This is true because all free quadrilateral meshing algorithms use a triangle mesh as a starting point.) Valid inputs for *Value* are:

**DEFAULT**

Let the program choose which quadrilateral mesher to use. In most cases, the program will choose the main quadrilateral mesher, which is the Q-Morph (quad-morphing) mesher. For very coarse meshes, the program may choose the alternate quadrilateral mesher instead. In most cases, the Q-Morph mesher results in higher quality elements. If either mesher fails for any reason, the program invokes the other mesher and retries the meshing operation. (Default.)

**MAIN**

The program uses the main quadrilateral mesher (Q-Morph mesher), and it does not invoke the alternate mesher if the main mesher fails.

**ALTERNATE**

The program uses the alternate quadrilateral mesher, and it does not invoke the Q-Morph mesher if the alternate mesher fails. To use the alternate quadrilateral mesher, you must also select **MOPT,AMESH,ALTERNATE** or **MOPT,AMESH,ALT2**.

**VMESH**

Tetrahedral element meshing option. Valid inputs for *Value* are:

**DEFAULT**

Let the program choose which tetrahedra mesher to use.

**MAIN**

Use the main tetrahedra mesher (Delaunay technique mesher). (GHS3D meshing technology by P. L. George, INRIA, France.) For most models, this mesher is significantly faster than the alternate mesher.

**ALTERNATE**

Use the alternate tetrahedra mesher (advancing front mesher). This mesher does not support the generation of a tetrahedral volume mesh from facets (**FVMESH**). If this mesher is selected and you issue the **FVMESH** command, the program uses the main tetrahedra mesher to create the mesh from facets and issues a warning message to notify you.

**SPLIT**

Quad splitting option for non-mapped meshing. If *Value* = 1, ON, or ERR, quadrilateral elements in violation of shape error limits are split into triangles (default). If *Value* = 2 or WARN, quadrilateral

elements in violation of either shape error or warning limits are split into triangles. If *Value* = OFF, splitting does not occur, regardless of element quality.

**LSMO**

Line smoothing option. *Value* can be ON or OFF. If *Value* = ON, smoothing of nodes on area boundaries is performed during smoothing step of meshing. During smoothing, node locations are adjusted to achieve a better mesh. If *Value* = OFF (default), no smoothing takes place at area boundaries.

**CLEAR**

This option affects the element and node numbering after clearing a mesh. If *Value* = ON (default), the starting node and element numbers will be the lowest available number after the nodes and elements are cleared. If *Value* = OFF, the starting node and element numbers are not reset after the clear operation.

**PYRA**

Transitional pyramid elements option. *Value* can be ON or OFF. If *Value* = ON (default), the program automatically creates transitional pyramid elements, when possible. Pyramids may be created at the interface of tetrahedral and hexahedral elements, or directly from quadrilateral elements. For pyramids to be created, you must also issue the command **MSHAPE,1,3D** (degenerate 3-D elements). If *Value* = OFF, the program does not create transitional pyramid elements.

**TIMP**

Identifies the level of tetrahedra improvement to be performed when the next free volume meshing operation is initiated (**VMESH, FVMESH**). (For levels 2-5, improvement occurs primarily through the use of face swapping and node smoothing techniques.) Valid inputs for *Value* are:

**0**

Turn off tetrahedra improvement. Although this value can lead to faster tetrahedral mesh creation, it is *not recommended* because it often leads to poorly shaped elements and mesh failures.

**1**

Do the minimal amount of tetrahedra improvement. (Default.) This option is supported by the main tetrahedra mesher only [**MOPT,VMESH,MAIN**]. If the alternate tetrahedra mesher (**MOPT,VMESH,ALTERNATE**) is invoked with this setting, the program automatically performs tetrahedra improvement at level 3 instead (**MOPT,TIMP,3**).

**2**

Perform the least amount of swapping/smoothing. No improvement occurs if all tetrahedral elements are within acceptable limits.

**3**

Perform an intermediate amount of swapping/smoothing. Some improvement is always done.

**4**

Perform the greatest amount of swapping/smoothing. Meshing takes longer with this level of improvement, but usually results in a better mesh.

**5**

Perform the greatest amount of swapping/smoothing, plus additional improvement techniques. This level of improvement usually produces results that are similar to those at level 4, except for very poor meshes.

**6**

For linear tetrahedral meshes, this value provides the same level of improvement as **MOPT,TIMP,5**. For quadratic tetrahedral meshes, this value provides an additional pass of cleanup. This value is supported for both the main (**MOPT,VMESH,MAIN**) and alternate (**MOPT,VMESH,ALTERNATE**) tetrahedra meshers.



**STAT**

Display status of **MOPT** settings. *Value* is ignored.

**DEFA**

Set all **MOPT** options to default values. *Value* is ignored.

*Value*

Value, as described for each different *Lab* above.

**Notes**

See the [Modeling and Meshing Guide](#) for more information on the **MOPT** command and its options.

This command is also valid for [rezoning](#).

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Mesher Opts**

**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Area Cntrls**

**Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Volu Cntrls**

**Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options**

**Utility Menu>List>Status>Preprocessor>Solid Model**

---

**MORPH**, *Option*, *--*, *Remeshopt*, *ElemSet*, *ARMAX*, *VOCH*, *ARCH*, *STEP*, *TIME*

**Specifies morphing and remeshing controls.**

PREP7: Morphing

MP ME <> <> <> <> <> <> <> <> <> PP <> EME MFS

*Option***OFF**

Turns off morphing for field elements (default).

**ON**

Turns on morphing for field elements.

--

Unused field

*Remeshopt***OFF**

Do not remesh (default).

**ON**

Remesh when element qualities fall below values specified by *ARMAX*, *VOCH*, or *ARCH* as explained below. Valid only when *Option* is ON.

*ElemSet***ALL**

Remesh all selected elements if the quality of the worst defined element falls below any quality requirement (default when *Remeshopt* = ON).

**CompName**

Specify a component name, up to 32 characters. All elements included in this component name are remeshed if the quality of the worst element falls below any quality requirement.

**ARMAX**

The maximum allowable element generalized aspect ratio. Defaults to 5. See [Element Qualities](#) in the *Fluids Analysis Guide* for the maximum generalized aspect ratio.

**VOCH**

The maximum allowable change of element size (area or volume). Defaults to 3. See [Element Qualities](#) in the *Fluids Analysis Guide* for maximum allowable change of element size.

**ARCH**

The maximum allowable element aspect ratio change. Defaults to 3. See [Element Qualities](#) in the *Fluids Analysis Guide* for maximum allowable element aspect ratio change.

**STEP**

The frequency of element quality checking, based on time steps. A quality check takes place at the intervals defined by *STEP*. Defaults to 1 (quality check at every step).

**TIME**

A quality check takes place at the time point specified. Defaults to -1 (a quality check at every time point).

## Notes

**MORPH** is applicable to any non-structural field analysis (not including fluid elements). It activates displacement degrees of freedom for non-structural elements so that boundary conditions may be placed on the field mesh to constrain the movement of the non-structural mesh during morphing. It morphs the non-structural mesh using displacements transferred at the surface interface between the structural field and the non-structural field. The displacements of non-structural elements are mesh displacements to avoid mesh distortion, but have no physical meaning except at the interface. **MORPH** does not support surface, link, or shell elements, or any element shape other than triangles, quads, tets, and bricks. Morphed fields must be in the global Cartesian system (**CSYS** = 0).

After each remesh, new databases and results files are written with the extensions `.rth0n` and `.db0n`, where *n* is the remesh file number (`FieldName.rth01`, `FieldName.rth02`, ... and `FieldName.db01`, `FieldName.db02`, etc.). The original database file is `FieldName.dbo`. The `FieldName.db01`, `FieldName.db02`, etc. files have elements that are detached from the solid model.

Remeshing has the following restrictions:

- Valid only for the electrostatic elements ([PLANE121](#), [SOLID122](#), and [SOLID123](#))
- Limited to triangle (2-D) and tetrahedral (3-D) options of these elements
- Valid only for the MFS solver
- No body loads allowed in the interior nodes of the remeshing domain
- Nodes on the boundary cannot be remeshed; remeshing will not work if morphing failed on the surface nodes
- Not suitable for extreme area or volume changes

This command is also valid in SOLUTION.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Element Morphing  
Main Menu>Solution>Load Step Opts>Other>Element Morphing

**MOVE**, *NODE*, *KC1*, *X1*, *Y1*, *Z1*, *KC2*, *X2*, *Y2*, *Z2*

**Calculates and moves a node to an intersection.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NODE*

Move this node. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

### *KC1*

First coordinate system number. Defaults to 0 (global Cartesian).

### *X1*, *Y1*, *Z1*

Input one or two values defining the location of the node in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Fields are R1,  $\theta$ 1, Z1 for cylindrical, or R1,  $\theta$ 1,  $\Phi$ 1 for spherical or toroidal.

### *KC2*

Second coordinate system number.

### *X2*, *Y2*, *Z2*

Input two or one value(s) defining the location of the node in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Fields are R2,  $\theta$ 2, Z2 for cylindrical, or R2,  $\theta$ 2,  $\Phi$ 2 for spherical or toroidal.

## Notes

Calculates and moves a node to an intersection location. The node may have been previously defined (at an approximate location) or left undefined (in which case it is internally defined at the **SOURCE** location). The actual location is calculated from the intersection of three surfaces (implied from three coordinate constants in two different coordinate systems). The three (of six) constants easiest to define should be used. The program will calculate the remaining three coordinate constants. All arguments, except *KC1*, must be input. Use the repeat command [**\*REPEAT**] after the **MOVE** command to define a line of intersection by repeating the move operation on all nodes of the line.

Surfaces of constant value are implied by some commands by specifying a single coordinate value. Implied surfaces are used with various commands [**MOVE**, **KMOVE**, **NSEL**, etc.]. Three surfaces are available with each of the four coordinate system types. Values of X, Y, or Z may be constant for the Cartesian coordinate system; values of R,  $\theta$ , or Z for the cylindrical system; and values of R,  $\theta$ ,  $\Phi$  for the spherical and toroidal systems. For example, an X value of 3 represents the Y-Z plane (or surface) at X=3. In addition, the parameters for the cylindrical and spherical coordinate systems may be adjusted [**CS**, **LOCAL**] to form elliptical surfaces. For surfaces in elliptical coordinate systems, a surface of "constant" radius is defined by the radius value at the X-axis. Surfaces of constant value may be located in local coordinate systems [**LOCAL**, **CLOCAL**, **CS**, or **CSKP**] to allow for any orientation.

The intersection calculation is based on an iterative procedure (250 iterations maximum) and a tolerance of 1.0E-4. The approximate location of a node should be sufficient to determine a unique intersection if more than one intersection point is possible. Tangent "intersections" should be avoided. If an intersection is not found, the node is placed at the last iteration location.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>To Intersect**

---

**MP**, *Lab, MAT, C0, C1, C2, C3, C4*

**Defines a linear material property as a constant or a function of temperature.**

[PREP7: Materials](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS  
[Product Restrictions](#)

### *Lab*

Valid material property label. Applicable labels are listed under "Material Properties" in the input table for each element type in the *Element Reference*. See [Linear Material Properties](#) of the *Element Reference* for more complete property label definitions:

#### **EX**

Elastic moduli (also EY, EZ).

#### **ALPX**

Secant coefficients of thermal expansion (also ALPY, ALPZ).

#### **CTEX**

Instantaneous coefficients of thermal expansion (also CTEY, CTEZ).

#### **THSX**

Thermal strain (also THSY, THSZ).

#### **REFT**

Reference temperature. Must be defined as a constant; C1 through C4 are ignored.

#### **PRXY**

Major Poisson's ratios (also PRYZ, PRXZ).

#### **NUXY**

Minor Poisson's ratios (also NUYZ, NUXZ).

#### **GXY**

Shear moduli (also GYZ, GXZ).

#### **DAMP**

K matrix multiplier for damping.

---

### **Note**

If used in an explicit dynamic analysis, the value corresponds to the percentage of damping in the high frequency domain. For example, 0.1 roughly corresponds to 10% damping in the high frequency domain.

**DMPR**

Constant material damping coefficient.

**MU**

Coefficient of friction.

**DENS**

Mass density.

**C**

Specific heat.

**ENTH**

Enthalpy.

**KXX**

Thermal conductivities (also KYY, KZZ).

**HF**

Convection or film coefficient.

**EMIS**

Emissivity.

**QRATE**

Heat generation rate.

**VISC**

Viscosity.

**SONC**

Sonic velocity.

**RSVX**

Electrical resistivities (also RSVY, RSVZ).

**PERX**

Electric relative permittivities (also PERY, PERZ).

---

**Note**

If you enter permittivity values less than 1 for [SOLID5](#), [PLANE13](#), or [SOLID98](#), the program interprets the values as absolute permittivity. Values input for [PLANE223](#), [SOLID226](#), or [SOLID227](#) are always interpreted as relative permittivity.

**MURX**

Magnetic relative permeabilities (also MURY, MURZ).

**MGXX**

Magnetic coercive forces (also MGY, MGZ).

**LSSM**

Magnetic loss tangent.

**LSST**

Electric loss tangent.

**SBKX**

Seebeck coefficients (also SBKY, SBKZ).

**MAT**

Material reference number to be associated with the elements (defaults to the current MAT setting [MAT]).

**C0**

Material property value, or if a property-versus-temperature polynomial is being defined, the constant term in the polynomial. *C0* can also be a table name (%*tablename*%); if *C0* is a table name, *C1* through *C4* are ignored.

**C1, C2, C3, C4**

Coefficients of the linear, quadratic, cubic, and quartic terms, respectively, in the property-versus-temperature polynomial. Leave blank (or set to zero) for a constant material property.

## Notes

**MP** defines a linear material property as a constant or in terms of a fourth order polynomial as a function of temperature. (See the **TB** command for nonlinear material property input.) Linear material properties typically require a single substep for solution, whereas nonlinear material properties require multiple substeps; see [Linear Material Properties](#) in the *Element Reference* for details.

If the constants *C1* - *C4* are input, the polynomial

$$\text{Property} = C0 + C1(T) + C2(T)^2 + C3(T)^3 + C4(T)^4$$

is evaluated at discrete temperature points with linear interpolation between points (that is, a piecewise linear representation) and a constant-valued extrapolation beyond the extreme points. First-order properties use two discrete points ( $\pm 9999^\circ$ ). The **MPTEMP** or **MPTGEN** commands must be used for second and higher order properties to define appropriate temperature steps. To ensure that the number of temperatures defined via the **MPTEMP** and **MPTGEN** commands is minimally sufficient for a reasonable representation of the curve, ANSYS generates an error message if the number is less than *N*, and a warning message if the number is less than  $2N$ . The value *N* represents the highest coefficient used; for example, if *C3* is nonzero and *C4* is zero, a cubic curve is being used which is defined using 4 coefficients so that  $N = 4$ .

A polynomial input is not valid in an explicit dynamic analysis or for *Lab* = DAMP. *C1*, *C2*, *C3*, and *C4* are ignored.

The use of tabular material properties (*C0* = %*tablename*%) is available only for **FLUID116**. Fluid properties can be evaluated as a function of pressure, temperature, velocity, time, and location (independent variables). Use the **\*DIM** command to create the table of property values as a function of the independent variables. Then refer to this table via the **MP** command for the property. If using temperature or pressure independent variables, you need to activate the appropriate pressure or temperature degrees of freedom on the element. Tabular material properties are calculated before the first iteration (that is, using initial values [IC]). For more information about using table-type array parameters, see the discussion on [applying tabular boundary conditions](#) in the *Basic Analysis Guide*.

When defining a reference temperature (**MP,REFT**), you can convert temperature-dependent secant coefficients of thermal expansion (SCTE) data from the definition temperature to the reference temperature. To do so, issue the **MPAMOD** command.

This command is also valid in SOLUTION.

## Product Restrictions

Command Option Lab	Available Products
ALPX	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
ALPY	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
ALPZ	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
C	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
CTEX	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
CTEY	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
CTEZ	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
DAMP	MP ME ST <> <> <> <> <> <> DY PP <> EME MFS
DENS	MP ME ST PR PRN DS DSS <> <> EH DY PP <> EME MFS
DMPR	MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS
EMIS	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
ENTH	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
EX	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
EY	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
EZ	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
GXY	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
GXZ	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
GYZ	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
HF	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
KXX	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
KYY	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
KZZ	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
LSSM	MP <> <> <> <> <> <> <> EM EH <> PP <> EME <>
LSST	MP ME <> <> <> <> <> <> EM EH <> PP <> EME MFS
MGXX	MP ME <> <> <> <> <> <> EM <> DY PP <> EME MFS
MGYY	MP ME <> <> <> <> <> <> EM <> DY PP <> EME MFS
MGZZ	MP ME <> <> <> <> <> <> EM <> DY PP <> EME MFS
MU	MP ME ST <> PRN <> <> <> <> <> DY PP <> EME MFS
MURX	MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS
MURY	MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS
MURZ	MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS
NUXY	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
NUXZ	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
NUYZ	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
PERX	MP ME <> <> <> <> <> FL EM EH DY PP <> EME MFS

PERY	MP ME <> <> <> <> <> FL EM EH DY PP <> EME MFS
PERZ	MP ME <> <> <> <> <> FL EM EH DY PP <> EME MFS
PRXY	MP ME ST PR PRN DS DSS FL <> <> DY PP <> EME MFS
PRXZ	MP ME ST PR PRN DS DSS FL <> <> DY PP <> EME MFS
PRYZ	MP ME ST PR PRN DS DSS FL <> <> DY PP <> EME MFS
QRAT	MP ME ST PR PRN DS DSS <> EM <> DY PP <> EME MFS
REFT	MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS
RSVX	MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS
RSVY	MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS
RSVZ	MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS
SBKX	MP ME <> <> <> <> <> <> EM <> <> PP <> EME MFS
SBKY	MP ME <> <> <> <> <> <> EM <> <> PP <> EME MFS
SBKZ	MP ME <> <> <> <> <> <> EM <> <> PP <> EME MFS
SONC	MP ME <> <> <> <> <> FL <> <> DY PP <> EME MFS
THSX	MP ME ST <> <> <> <> FL <> <> DY PP <> EME MFS
THSY	MP ME ST <> <> <> <> FL <> <> DY PP <> EME MFS
THSZ	MP ME ST <> <> <> <> FL <> <> DY PP <> EME MFS
VISC	MP ME ST <> <> <> <> FL <> <> DY PP <> EME MFS

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models**

**Main Menu>Preprocessor>Material Props>Material Models**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models**

## MPAMOD, MAT, DEFTEMP

**Modifies temperature-dependent secant coefficients of thermal expansion.**

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### **MAT**

Material number for which the secant coefficients of thermal expansion (SCTE's) are to be modified. Defaults to 1.

### **DEFTEMP**

Definition temperature at which the existing SCTE-versus-temperature tables were defined. Defaults to zero.

## Notes

This command converts temperature-dependent SCTE data (properties ALPX, ALPY, ALPZ) from the definition temperature (*DEFTEMP*) to the reference temperature defined by **MP,REFT** or **TREF**. If both the **MP,REFT**



and **TREF** commands have been issued, the reference temperature defined by the **MP,REFT** command will be used.

This command does not apply to the instantaneous coefficients of thermal expansion (properties CTEX, CTEY, CTEZ) or to the thermal strains (properties THSX, THSY, THSZ).

See [Linear Material Properties](#) of the *Element Reference* and the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more details.

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Convert ALPx**

**Main Menu>Preprocessor>Material Props>Convert ALPx**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Convert ALPx**

## MPCHG, MAT, ELEM

**Changes the material number attribute of an element.**

[PREP7: Materials](#)

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **MAT**

Assign this material number to the element. Material numbers are defined with the material property commands [**MP**].

### **ELEM**

Element for material change. If ALL, change materials for all selected elements [**ESEL**].

## Notes

Changes the material number of the specified element. Between load steps in SOLUTION, material properties cannot be changed from linear to nonlinear, or from one nonlinear option to another.

If you change from one MKIN model to another MKIN model, the different MKIN models need to have the same number of data points. This requirement also applies if you change from one KINH model to another KINH model, or from one CHABOCHE model to another CHABOCHE model.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Change Mat Num**

**Main Menu>Preprocessor>Material Props>Change Mat Num**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Change Mat Num**

**MPCOPY**, --, *MATF*, *MATT*

**Copies linear material model data from one material reference number to another.**

[PREP7: Materials](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

--

Unused field

**MATF**

Material reference number from where material property data will be copied.

**MATT**

Material reference number to where material property data will be copied.

**Notes**

The **MPCOPY** command copies linear material properties only, which are all properties defined through the **MP** command. If you copy a model that includes both linear and yield behavior constants (for example, a BKIN model), the **MPCOPY** and **TBCOPY**, ALL commands are used together to copy the entire model. All input data associated with the model is copied, that is, all data defined through the **MP** and **TB** commands.

Also, if you copy a material model using the [Material Model Interface \(Edit> Copy\)](#), both the commands **MPCOPY** and **TBCOPY**, ALL are issued, regardless of whether the model includes linear constants only, or if it includes a combination of linear and yield behavior constants.

This command is also valid in SOLUTION.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models**

**Main Menu>Preprocessor>Material Props>Material Models**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models**

**MPDATA**, *Lab*, *MAT*, *STLOC*, *C1*, *C2*, *C3*, *C4*, *C5*, *C6*

**Defines property data to be associated with the temperature table.**

[PREP7: Materials](#)

MP ME <> <> <> <> <> EM EH <> PP <> EME MFS

[Product Restrictions](#)

**Lab**

Valid property label. Applicable labels are listed under "Material Properties" in the input table for each element type in the [Element Reference](#). See [Linear Material Properties](#) of the [Element Reference](#) for more complete property label definitions:

**EX**

Elastic moduli (also EY, EZ).

**ALPX**

Secant coefficients of thermal expansion (also ALPY, ALPZ). (See also **MPAMOD** command for adjustment to reference temperature).

**CTEX**

Instantaneous coefficients of thermal expansion (also CTEY, CTEZ).

**THSX**

Thermal strain (also THSY, THSZ).

**REFT**

Reference temperature (may not be temperature dependent).

**PRXY**

Major Poisson's ratios (also PRYZ, PRXZ).

**NUXY**

Minor Poisson's ratios (also NUYZ, NUXZ).

**GXY**

Shear moduli (also GYZ, GXZ).

**DAMP**

K matrix multiplier for damping.

**DMPR**

Constant material damping coefficient.

**MU**

Coefficient of friction.

**DENS**

Mass density.

**C**

Specific heat.

**ENTH**

Enthalpy.

**KXX**

Thermal conductivities (also KYY, KZZ).

**HF**

Convection or film coefficient.

**EMIS**

Emissivity.

**QRATE**

Heat generation rate.

**VISC**

Viscosity.

**SONC**

Sonic velocity.

**RSVX**

Electrical resistivities (also RSVY, RSVZ).

**PERX**

Electric relative permittivities (also PERY, PERZ).

**MURX**

Magnetic relative permeabilities (also MURY, MURZ).

**MGXX**

Magnetic coercive forces (also MGY, MGZ).

**LSST**

Dielectric loss tangent.

**SBKX**

Seebeck coefficients (also SBK, SBKZ).

The **MPDATA** command may also be used to enter temperature dependent properties for fluids in a CFD analysis with FLOTRAN via [FLUID141](#) and [FLUID142](#). Valid **MPDATA** labels for a CFD analysis in a non-solid region are:

**DENS**

Density of fluid. This is the same as the label used to specify mass density with the **FLDATA** command.

**C**

Specific heat of fluid. This is equivalent to the SPHT label used to specify conductivity with the **FLDATA** command.

**KXXX**

Thermal conductivity of fluid.

**VISC**

Viscosity of fluid. This is the same as the label used to specify kinematic velocity with the **FLDATA** command.

**MAT**

Material reference number to be associated with the elements (defaults to 1 if you specify zero or no material number).

**STLOC**

Starting location in table for generating data. For example, if  $STLOC = 1$ , data input in the  $C1$  field is the first constant in the table. If  $STLOC = 7$ , data input in the  $C1$  field is the seventh constant in the table, etc. Defaults to the last location filled + 1.

**C1, C2, C3, ..., C6**

Property data values assigned to six locations starting with  $STLOC$ . If a value is already in this location, it is redefined. A blank (or zero) value for  $C1$  resets the previous value in  $STLOC$  to zero. A value of zero can only be assigned by  $C1$ . Blank (or zero) values for  $C2$  to  $C6$  leave the corresponding previous values unchanged.

## Notes

Defines a table of property data to be associated with the temperature table. Repeat **MPDATA** command for additional values (100 maximum). Temperatures must be defined first [**MPTEMP**]. Also stores assembled property function table (temperature and data) in virtual space.

This command is also valid in SOLUTION.

## Product Restrictions

In ANSYS without Emag enabled, the MUR\_ and MG\_\_ properties are not allowed. In ANSYS Professional, all structural and thermal properties are allowed except DAMP and MU. In ANSYS Emag, only the RSV\_, PER\_, MUR\_, and MG\_\_ properties are allowed. Only products that include ANSYS Emag can use the LSST property. The SBK\_ property is only available in ANSYS Multiphysics, ANSYS PrepPost, and ANSYS ED.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

**MPDELE**, *Lab*, *MAT1*, *MAT2*, *INC*, *--*, *LCHK*

Deletes linear material properties.

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lab*

Material property label (see **MP** command for valid labels). If ALL, delete properties for all applicable labels.

### *MAT1*, *MAT2*, *INC*

Delete materials from *MAT1* to *MAT2* (defaults to *MAT1*) in steps of *INC* (defaults to 1). If *MAT1* = ALL, *MAT2* and *INC* are ignored and the properties for all materials are deleted.

--

Unused field.

### *LCHK*

Specifies the level of element-associativity checking:

#### **NOCHECK**

No element-associativity check occurs. This option is the default.

#### **WARN**

When a section, material, or real constant is associated with an element, ANSYS issues a message warning that the necessary entity has been deleted.

#### **CHECK**

The command terminates, and no section, material, or real constant is deleted if it is associated with an element.

## Notes

This command is also valid in SOLUTION.

The *LCHK* argument is valid only when *Lab* = ALL.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

**MPDRES**, *LabF*, *MATF*, *LabT*, *MATT*

Reassembles existing material data with the temperature table.

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***LabF***

Material property label associated with *MATF*.

***MATF***

Material reference number of property to restore from virtual space.

***LabT***

Material property label associated with *MATT* (defaults to label associated with *MATF*).

***MATT***

Material reference number assigned to generated property (defaults to *MATF*).

## Notes

Restores into the database (from virtual space) a data table previously defined [MP] for a particular property, assembles data with *current* database temperature table, and stores back in virtual space as a new property.

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Other>Change Mat Props>Modify Temps**

**Main Menu>Preprocessor>Material Props>Modify Temps**

**Main Menu>Solution>Other>Change Mat Props>Modify Temps**

---

**/MPLIB**, *R-W\_opt*, *PATH*

Sets the default material library read and write paths.

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***R-W\_opt***

Determines what path is being set. Possible values are:

**READ**

Set the read path.

**WRITE**

Set the write path.

**STAT**

Report what read and write paths are currently in use.

***PATH***

The directory path to be used for material library files.

## Notes

The **/MPLIB** command sets two path strings used in conjunction with the material library feature and the **MPREAD** and **MPWRITE** commands.

For **MPREAD**, when you use the *LIB* option and no directory path is given in the file name, the command searches for the file in these locations: the current working directory, the user's home directory, the user-specified material library directory (as defined by the **/MPLIB,READ,PATH** command), and */ansys\_dir/mat-lib*.

For **MPWRITE**, when you use the *LIB* option and the directory portion of the specification for the material library file is blank, the command writes the material library file to the directory specified by the **/MPLIB,WRITE,PATH** command (if that path has been set). If the path has not been set, the default is to write the file to the current working directory.

The Material Library files supplied with the distribution disks are meant for demonstration purposes only. These files are not intended for use in customer applications.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Lib Path Status**

**Main Menu>Preprocessor>Material Props>Material Library>Lib Path Status**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Lib Path Status**

**MPLIST**, *MAT1*, *MAT2*, *INC*, *Lab*, *TEVL*

**Lists linear material properties.**

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**MAT1**, **MAT2**, **INC**

List materials from *MAT1* to *MAT2* (defaults to *MAT1*) in steps of *INC* (defaults to 1). If *MAT1*= ALL (default), *MAT2* and *INC* are ignored and properties for all material numbers are listed.

**Lab**

Material property label (see the **MP** command for labels). If ALL (or blank), list properties for all labels. If EVLT, list properties for all labels evaluated at TEVL.

**TEVL**

Evaluation temperature for *Lab* = EVLT listing (defaults to **BFUNIF**).

## Notes

For *Lab* = EVLT, when the property is from tables, the **MP PLOT** command will not be valid because the property could be a function of more than temperature.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Properties>All Materials  
Utility Menu>List>Properties>All Matls, All Temps  
Utility Menu>List>Properties>All Matls, Specified Temp  
Utility Menu>List>Properties>Specified Matl, All Temps

---

**MPPLOT**, *Lab, MAT, TMIN, TMAX, PMIN, PMAX*

Plots linear material properties as a function of temperature.

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lab*

Linear material property label (EX, EY, etc.) [**MP**].

### *MAT*

Material reference number. Defaults to 1.

### *TMIN*

Minimum abscissa value to be displayed.

### *TMAX*

Maximum abscissa value.

### *PMIN*

Minimum property (ordinate) value to be displayed.

### *PMAX*

Maximum property value.

## Notes

When the property is from tables, the **MPPLOT** command will not be valid because the property could be a function of more than temperature.

This command is valid in any processor.

## Menu Paths

Utility Menu>Plot>Materials

---

**MPREAD**, *Fname, Ext, --, LIB*

Reads a file containing material properties.

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including directory). If you do not specify the **LIB** option, the default directory is the current working directory. If you specify the **LIB** option, the



default is the following search path: the current working directory, the user's home directory, MPLIB\_DIR (as specified by the **/MPLIB,READ,PATH** command) and */ansys\_dir/matlib* (as defined by installation). If you use the default for your directory, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

#### **Ext**

Filename extension (8 character maximum).

If you omit the default extension is MP. extension is units\_MPL, where units is the system of units currently in use. (See the description of the **/UNITS** command.) For example, if **/UNITS** is set to SI, the extension defaults to SI\_MPL.

--

Unused field.

#### **LIB**

Reads material library files previously written with the **MPWRITE** command. (See the description of the LIB option for the **MPWRITE** command.) The only allowed value for LIB is LIB.

The LIB field indicates that the specified file was written by **MPWRITE** using the LIB option, and that the file is consistent with the material library file format. When the **MPREAD** command executes, the ANSYS program reads material properties defined in the specified file into the current ANSYS working database. The currently selected material, as defined by the **MAT** command (**MAT,MAT**), determines the material number used when reading the material properties. The LIB option for **MPREAD** and **MPWRITE** supports storing and retrieving both linear and nonlinear properties.

## Notes

Material properties written to a file without the LIB option do not support nonlinear properties. Also, properties written to a file without the LIB option are restored in the same material number as originally defined. To avoid errors, use **MPREAD** with the LIB option only when reading files written using **MPWRITE** with the LIB option.

If you omit the LIB option for **MPREAD**, this command supports only linear properties.

Material numbers are hardcoded. If you write a material file without specifying the LIB option, then read that file in using the **MPREAD** command with the LIB option, the ANSYS program will not write the file to a new material number. Instead, it will write the file to the "old" material number (the number specified on the **MPWRITE** command that created the file.)

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Export Library**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Import Library**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Select Units**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Read from File**

**Main Menu>Preprocessor>Material Props>Material Library>Export Library**

**Main Menu>Preprocessor>Material Props>Material Library>Import Library**

**Main Menu>Preprocessor>Material Props>Material Library>Select Units**  
**Main Menu>Preprocessor>Material Props>Read from File**  
**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Export Library**  
**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Import Library**  
**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Select Units**  
**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Read from File**

---

## MPRINT, *KEY*

Specifies that radiation matrices are to be printed.

[AUX12: Radiation Matrix Method](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *KEY*

Print key:

- 0**  
Do not print matrices.
- 1**  
Print matrices.

## Command Default

Matrices are not printed.

## Notes

Specifies that the element and node radiation matrices are to be printed when the **WRITE** command is issued. If *KEY* = 1, form factor information for each element will also be printed.

## Menu Paths

**Main Menu>Radiation Opt>Matrix Method>Write Matrix**

---

## MPTEMP, *STLOC, T1, T2, T3, T4, T5, T6*

Defines a temperature table for material properties.

[PREP7: Materials](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

### *STLOC*

Starting location in table for entering temperatures. For example, if *STLOC* = 1, data input in the *T1* field applies to the first constant in the table. If *STLOC* = 7, data input in the *T1* field applies to the seventh constant in the table, etc. Defaults to the last location filled + 1.

### *T1, T2, T3, . . . , T6*

Temperatures assigned to six locations starting with *STLOC*. If a value is already in this location, it will be redefined. A blank (or zero) value for *T1* resets the previous value in *STLOC* to zero. A value of zero

can only be assigned by  $T1$ . Blank (or zero) values for  $T2$  to  $T6$  leave the corresponding previous values unchanged.

## Command Default

No temperature table defined (i.e., properties must be defined as a constant or linear function of temperature with the **MP** command).

## Notes

Defines a temperature table to be associated with the property data table [**MPDATA**]. These temperatures are also used for polynomial property evaluation, if defined [**MP**]. Temperatures must be defined in non-descending order. Issue **MATER \$ STAT** to list the current temperature table. Repeat **MPTEMP** command for additional temperatures (100 maximum). If all arguments are blank, the temperature table is erased.

For clear definition, the temperature range you define with the **MPTEMP** command should include the entire range you'll use in subsequently defined materials. To assist the user in this, the first (and only the first) excursion out of the temperature range defined by the **MPTEMP** commands is flagged with a warning message. Similarly, the reference temperature (**TREF** or **MP,ref** commands) should also fall in this same temperature range. If not and **MP,alpx** was used, a note will be output. If not, and **MP,ctex** or **MP,thsx** was used, an error message will be output.

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models**

**Main Menu>Preprocessor>Material Props>Material Models**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models**

---

## MPTGEN, *STLOC*, *NUM*, *TSTRT*, *TINC*

**Adds temperatures to the temperature table by generation.**

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL <> <> DY PP <> EME MFS

### *STLOC*

Starting location in table for generating temperatures. Defaults to last location filled + 1.

### *NUM*

Number of temperatures to be generated (1-100).

### *TSTRT*

Temperature assigned to *STLOC* location.

### *TINC*

Increment previous temperature by *TINC* and assign to next location until all *NUM* locations are filled.

## Notes

Adds temperatures to the temperature table by generation. May be used in combination (or in place of) the **MPTEMP** command.

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Other>Change Mat Props>Generate Temp**

**Main Menu>Preprocessor>Material Props>Generate Temp**

**Main Menu>Solution>Other>Change Mat Props>Generate Temp**

## MPTRES, *Lab*, *MAT*

Restores a temperature table previously defined.

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL <> <> DY PP <> EME MFS

### *Lab*

Material property label [[MP](#)].

### *MAT*

Material reference number.

## Notes

Restores into the database (from virtual space) a temperature table previously defined [[MP](#)] for a particular property. The existing temperature table in the database is erased before this operation.

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Other>Change Mat Props>Restore Temps**

**Main Menu>Preprocessor>Material Props>Restore Temps**

**Main Menu>Solution>Other>Change Mat Props>Restore Temps**

## MPWRITE, *Fname*, *Ext*, *--*, *LIB*, *MAT*

**Writes linear material properties in the database to a file (if the LIB option is not specified) or writes both linear and nonlinear material properties (if LIB is specified) from the database to a file.**

[PREP7: Materials](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including directory). If you do not specify the `LIB` option, the default directory is the current working directory. If you specify `LIB` and you have specified a material library directory (via the [/MPLIB](#) command), that directory is the default. Otherwise, the default is the current working directory. If you use the default for your directory, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

**Ext**

Filename extension (8 character maximum).

If you omit the `LIB` option, the default extension is `MP`. If you specify the `LIB` option, the default extension is `units_MPL`, where `units` is the system of units currently in use. (See the description of the `/UNITS` command.) For example, if `/UNITS` is set to `BIN`, the extension defaults to `BIN_MPL`.

--

Unused field.

**LIB**

The only value allowed for this field is the string "LIB."

The `LIB` option indicates that you wish to have properties associated with the material (`MAT`) written to the specified material library file using the material library file format. The material library file format is ASCII-text-based ANSYS command input. Certain commands associated with this format have been modified to interpret the string "`_MATL`" to mean the currently selected material. This feature makes the material library file independent of the material number in effect when the file was written; this enables you to restore the properties into the ANSYS database using the material number of your choice. The `LIB` option also enables you to save both linear and nonlinear properties. If you omit the `LIB` option, you can save linear properties only.

**MAT**

Specifies the material to be written to the named material library file. *There is no default; you must either specify a material or omit the `MAT` argument.* Even if you specify a `MAT` value, the ANSYS program ignores it if the `LIB` argument is not specified.

## Notes

Writes linear material properties currently in the database to a file. The file is rewound before and after writing.

This command is also valid in `SOLUTION`.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Export Library**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Import Library**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Select Units**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Write to File**

**Main Menu>Preprocessor>Material Props>Material Library>Export Library**

**Main Menu>Preprocessor>Material Props>Material Library>Import Library**

**Main Menu>Preprocessor>Material Props>Material Library>Select Units**

**Main Menu>Preprocessor>Material Props>Write to File**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Export Library**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Import Library**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Select Units**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Write to File**

**/MREP**, NAME, ARG1, ARG2, ARG3, . . . , ARG4, ARG5, ARG6, ARG7, ARG8, ARG9, ARG10, ARG11, ARG12, ARG13, ARG14, ARG15, ARG16, ARG17, ARG18

Enables you to reissue the graphics command macro "name" during a replot or zoom operation.

GRAPHICS: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### NAME

The name identifying the macro file or macro block on a macro library file. The name can contain up to eight characters maximum and must begin with a letter.

#### ARG1, ARG2, ARG3, . . . , ARG18

Values to be passed into the file or block.

## Notes

This command reissues the graphics command macro "name" during a replot operation [**/RELOT**] or a zoom [**/ZOOM**] operation. The ANSYS program passes the command macro arguments to the replot and zoom feature for use by the graphics macro. You should place the **s-MREP** command at the end of the graphics command macro, following the last graphics command within the macro, to enable the replot or zoom feature.

## Menu Paths

This command cannot be accessed from a menu.

## MSADV, SPNUM, MTHA

Specifies the approach to discretize the advection term in a species transport equation.

PREP7: FLOTTRAN Multiple Species

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

#### SPNUM

Species number, from 1 to 6. Must be specified.

#### MTHA

Choice of approach to discretize the advection term:

##### MSU

Monotone Streamline Upwind (MSU) approach.

##### SUPG

Streamline Upwind / Petrov-Galerkin (SUPG) approach (default).

##### COLG

Collocated Galerkin (COLG) approach.

## Notes

This command is valid for the multiple species transport option in a FLOTTRAN analysis. See [Advection Term](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on the advection term.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species

Main Menu>Solution>FLOTRAN Set Up>Multiple Species

## MSAVE, Key

Sets the solver memory saving option. This option only applies to the PCG solver.

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

### Key

Activation key:

#### 0 or OFF

Use global assembly for the stiffness matrix (and mass matrix, when using **MODOPT,LANPCG**) of the entire model.

#### 1 or ON

Use an element-by-element approach when possible to save memory during the solution. In this case, the global stiffness (and mass) matrix is not assembled; element stiffness (mass) is regenerated during PCG or **MODOPT,LANPCG** iterations.

## Notes

**MSAVE,ON** only applies to and is the default for parts of the model using the following element types with linear material properties that meet the conditions listed below.

- [SOLID186](#) (Structural Solid only)
- [SOLID187](#)

The following conditions must also be true:

- The PCG solver has been specified.
- The analysis is a static analysis. Topological optimization is not supported.
- Small strains are assumed (**NLGEOM,OFF**).
- No prestress effects (**PSTRES**) are included.
- No stress stiffening effects (**SSTIF**) are included when small strains are assumed (**NLGEOM,OFF**).
- All nodes on the supported element types must be defined (i.e., the midside nodes cannot be removed using the **EMID** command).
- For elements with thermally dependent material properties, **MSAVE,ON** applies only to elements with uniform temperatures prescribed.
- The default element coordinate system must be used.

If you manually force **MSAVE,ON** by including it in the input file, the model can include the following additional conditions:

- The analysis can be a modal analysis using the PCG Lanczos method (**MODOPT**,LANPCG).
- Large deflection effects (**NLGEOM**,ON) are included.
- **SOLID185** (brick shapes and KEYOPT(2) = 3 only) elements can be included.

All other element types or other parts of the model that don't meet the above criteria will be solved using global assembly (**MSAVE**,OFF). This command can result in memory savings of up to 70 percent over the global assembly approach for the part of the model that meets the criteria. Depending on the processor speed and the computer manufacturer, the solution time may increase or decrease when this feature is used.

This memory-saving feature runs in parallel when multiple processors are used with the **/CONFIG** command or with Distributed ANSYS. The gain in performance with using multiple processors with this feature turned on should be similar to the default case when this feature is turned off. Performance also improves when using the uniform reduced integration option for **SOLID186** elements.

This command does not support the *layered* option of the **SOLID185** and **SOLID186** elements.

When using **MSAVE**,ON with the **PCGOPT** command, note the following restrictions: .

- For static and modal analyses, **MSAVE**,ON is not valid when using a *Lev\_Diff* value of 5 on the **PCGOPT** command; *Lev\_Diff* will automatically be reset to 2.
- For modal analyses, **MSAVE**,ON is not valid with the *StrmCk* option of the **PCGOPT** command; *Strmck* will be set to OFF.

When using **MSAVE**,ON for modal analyses, no *.FULL* file will be created. The *.FULL* file may be necessary for subsequent analyses (e.g., harmonic, transient mode superposition, or spectrum analyses). To generate the *.FULL* file, rerun the modal analysis using the **WRFULL** command.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>Analysis Options**

## MSCAP, SPNUM, Capkey, UPPER, LOWER

Activates and controls mass fraction capping for a species.

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### SPNUM

Species number, from 1 to 6. Must be specified.

### Capkey

Key to activate mass fraction capping:

#### OFF

Capping not enforced (default).

#### ON

Capping will be enforced.

### UPPER, LOWER

Upper and lower bounds on mass fraction if capping is activated. Default to 1.0 and 0.0 respectively.



## Command Default

No mass fraction capping.

## Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**

**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

## MSDATA, ALGEB, UGAS

**Defines multiple species data applicable to all species.**

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### *ALGEB*

The algebraic species number whose mass fraction is calculated by subtracting the sum of the mass fractions of all other species from 1.0. This ensures that the sum of the mass fractions of all the species is 1.0. Defaults to 2.

### *UGAS*

The universal gas constant. Defaults to 8314.3 (SI units).

## Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**

**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

**\*MSG**, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*

**Writes an output message via the ANSYS message subroutine.**

APDL: Macro Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Lab*

Label for output and termination control:

#### **INFO --**

Writes the message with no heading (default).

#### **NOTE --**

Writes the message with a "NOTE" heading.

#### **WARN --**

Writes the message with a "WARNING" heading. Also writes the message to the errors file, *Jobname*.ERR.

#### **ERROR --**

Writes the message with a "ERROR" heading and causes run termination (if batch) at earliest "clean exit" point. Also writes the message to the errors file, *Jobname*.ERR.

#### **FATAL --**

Writes the message with a "FATAL ERROR" heading and causes run termination immediately. Also writes the message to the errors file, *Jobname*.ERR.

#### **UI --**

Writes the message with a "NOTE" heading and displays it in the message dialog box. This option is most useful in GUI mode.

### *VAL1*, *VAL2*, *VAL3*, . . . , *VAL8*

Numeric or alphanumeric character values to be included in message. Values may be the results of parameter evaluations. All numeric values are assumed to be double precision. The FORTRAN nearest integer (NINT) function is used to form integers for the %I specifier.

## Notes

Allows writing an output message via the ANSYS message subroutine. Also allows run termination control. This command is used only when contained in a prepared file read into the ANSYS program (i.e., **\*USE/INPUT**, etc.). A message format must immediately follow the **\*MSG** command (on a separate line, without parentheses, as described below).

The message format may be up to 80 characters long, consisting of text strings and predefined "data descriptors" between the strings where numeric or alphanumeric character data are to be inserted. The normal descriptors are %I for integer data, %G for double precision data, %C for alphanumeric character data, and %/ for a line break. The corresponding FORTRAN data descriptors are I9, 1PG16.9 and A8, respectively. Each descriptor must be preceded by a blank. There must be one data descriptor for each specified value (8 maximum) in the order of the specified values.

Enhanced descriptions may also be used:

%w.pE            w is field width

%w.pG	p is precision
%w.pF	
%%	a single percent sign
%wC;%wS	character string
%-wC;%-wS	left justify string
%wX	w blank characters
%wl	integer format
%0wl	pad integer with leading zeros rather than blanks
%0w.pl	w is field width; p is number of characters filled

Do not begin **\*MSG** format lines with **\*IF**, **\*ELSE**, **\*ELSEIF**, or **\*ENDIF**. If the last nonblank character of the message format is an ampersand (&), a second line will also be read as a continuation of the format. Up to nine continuations (ten total lines) may be read. If normal descriptions are used, then consecutive blanks are condensed into one blank upon output, and a period is appended. Up to ten lines of output of 72 characters each may be produced (using the %/ descriptor). Two examples follow.

Here is an example of the **\*MSG** command and a format to print a message with two integer values and one real value:

```
*MSG, INFO, 'Inner',25,1.2,148
Radius ( %C) = %I, Thick = %G, Length = %I
```

The output line is:

```
Radius (Inner) = 25, Thick = 1.2, Length = 148.
```

Here is an example illustrating multiline displays in GUI message windows:

```
*MSG,UI,Vcoilrms,THTAv,Icoilrms,THTAi,Papprnt,Pelec,PF,indctnc
Coil RMS voltage, RMS current, apparent pwr, actual pwr, pwr factor: %/&
Vcoil = %G V (electrical angle = %G DEG) %/&
Icoil = %G A (electrical angle = %G DEG) %/&
APPARENT POWER = %G W %/&
ACTUAL POWER = %G W %/&
Power factor: %G %/&
Inductance = %G %/&
VALUES ARE FOR ENTIRE COIL (NOT JUST THE MODELED SECTOR)
```

---

## Note

The **/UIS,MSGPOP** command controls which messages are displayed in the message dialog box when the GUI is active. All messages produced by the **\*MSG** command are subject to the **/UIS** specification, with one exception, if *Lab* = UI, the message will be displayed in the dialog box regardless of the **/UIS** specification.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

**MSHAPE**, *KEY*, *Dimension*

For elements that support multiple shapes, specifies the element shape to be used for meshing.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KEY**

Key indicating the element shape to be used:

**0**

Mesh with quadrilateral-shaped elements when *Dimension* = 2-D mesh with hexahedral-shaped elements when *Dimension* = 3-D.

**1**

Mesh with triangle-shaped elements when *Dimension* = 2-D mesh with tetrahedral-shaped elements when *Dimension* = 3-D.

**Dimension**

Specifies the dimension of the model to be meshed:

**2D**

2-D model (area mesh).

**3D**

3-D model (volume mesh).

**Command Default**

Since specification of element shape [**MSHAPE**] and meshing type [**MSHKEY**] are so closely related, the element shape that ANSYS meshes with depends on the combination of the values that are set for the two commands. The table below explains what happens when you fail to specify values for these settings.

Your action...	How it affects the mesh...
You issue the <b>MSHAPE</b> command with no arguments.	ANSYS uses quadrilateral-shaped or hexahedral-shaped elements to mesh the model, depending on whether you are meshing an area or a volume.
You do not specify an element shape, but you do specify the type of meshing to be used [ <b>MSHKEY</b> ].	ANSYS uses the default shape of the element to mesh the model. It uses the type of meshing that you specified.
You specify neither an element shape nor the type of meshing to be used.	ANSYS uses the default shape of the element to mesh the model. It uses whichever type of meshing is the default for that shape.

**Notes**

If no value is specified for *Dimension* the value of *KEY* determines the element shape that will be used for both 2-D and 3-D meshing. In other words, if you specify **MSHAPE,0**, quadrilateral-shaped and hexahedral-shaped elements will be used. If you specify **MSHAPE,1**, triangle-shaped and tetrahedral-shaped elements will be used.

The **MSHAPE**, **MSHKEY**, and **MSHMD** commands replace the functionality that was provided by the **ESHAPE** command in ANSYS 5.3 and earlier releases.

This command is also valid for [rezoning](#).

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided**

**Main Menu>Preprocessor>Meshing>Mesher Opts**

**Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options**

**MSHCOPY**, *KEYLA, LAPTRN, LACOPY, KCN, DX, DY, DZ, TOL, LOW, HIGH*

**Simplifies the generation of meshes that have matching node element patterns on two different line groups (in 2-D) or area groups (3-D).**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **KEYLA**

Copy line mesh (default) if LINE, 0 or 1. Copy area mesh if AREA, or 2.

### **LAPTRN**

Meshed line/area to be copied, or a component name containing a list. If *LAPTRN* = P, graphical picking is enabled (valid only in the GUI).

### **LACOPY**

Unmeshed line/area to get copied mesh, or a component name containing a list. If *LACOPY* = P, graphical picking is enabled (valid only in the GUI).

### **KCN**

In coordinate system *KCN*, *LAPTRN* + *DX DY DZ* = *LACOPY*.

### **DX, DY, DZ**

Node location increments in the active coordinate system (DR, D $\theta$ , DZ for cylindrical, DR, D $\theta$ , D $\Phi$  for spherical or toroidal).

### **TOL**

Tolerance. Defaults to 1.e--4.

### **LOW**

Name of low node component to be defined (optional).

### **HIGH**

Name of high node component to be defined (optional).

## Notes

Matching meshes are used for rotational (cyclic) symmetry, or for contact analysis using coupling or node-to-node gap elements. See [Using CPCYC and MSHCOPY Commands](#) in the *Modeling and Meshing Guide* for more information.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Copy>Area Mesh**

**Main Menu>Preprocessor>Modeling>Copy>Line Mesh**

**MSHKEY**, *KEY*

Specifies whether free meshing or mapped meshing should be used to mesh a model.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KEY**

Key indicating the type of meshing to be used:

- 0**  
Use free meshing (the default).
- 1**  
Use mapped meshing.
- 2**  
Use mapped meshing if possible; otherwise, use free meshing. If you specify **MSHKEY,2**, SmartSizing will be inactive even while free meshing non-map-meshable areas.

**Command Default**

As stated above, free meshing is the default. However, since the **MSHKEY** and **MSHAPE** settings are closely related, you should refer to the table in the **MSHAPE** command description for more information about defaults.

**Notes**

The **MSHKEY**, **MSHAPE**, and **MSHMID** commands replace the functionality that was provided by the **ESHAPE** command in ANSYS 5.3 and earlier releases.

This command is also valid for [rezoning](#).

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>3 or 4 sided**

**Main Menu>Preprocessor>Meshing>Mesh>Areas>Target Surf**

**Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided**

**Main Menu>Preprocessor>Meshing>Mesher Opts**

**Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options**

---

**MSHMID**, *KEY*

Specifies placement of midside nodes.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KEY**

Key indicating placement of midside nodes:

- 0** Midside nodes (if any) of elements on a region boundary follow the curvature of the boundary line or area (the default).
- 1** Place midside nodes of all elements so that element edges are straight. Allows coarse mesh along curves.
- 2** Do not create midside nodes (elements will have removed midside nodes).

## Notes

The **MSHMID**, **MSHAPE**, and **MSHKEY** commands replace the functionality that was provided by the **ESHAPE** command in ANSYS 5.3 and earlier releases.

This command is also valid for [rezoning](#).

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesher Opts**

**Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options**

---

## MSHPATTERN, *KEY*

Specifies pattern to be used for mapped triangle meshing.

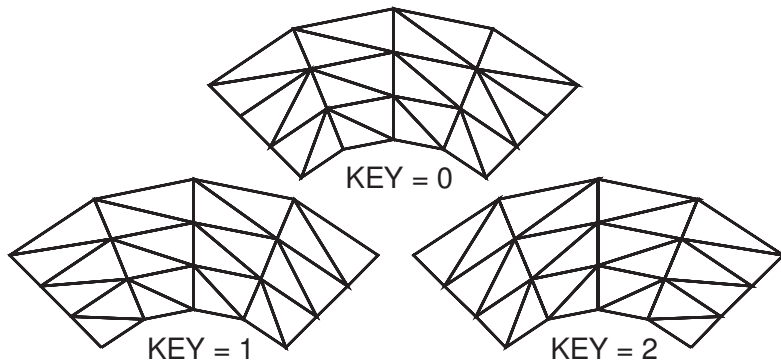
[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *KEY*

Key indicating triangle pattern to be used (the figures below illustrate the pattern that will be used for each value of *KEY*):

- 0** Let ANSYS choose the pattern (the default). ANSYS maximizes the minimum angle of the triangular-shaped elements that are created.
- 1** Unidirectional split at node I.
- 2** Unidirectional split at node J.



## Notes

"Mapped triangle meshing" refers to the ANSYS program's ability to take a map-meshable area and mesh it with triangular elements, based on the value of **MSHPATTERN**,*KEY*. This type of meshing is particularly useful for analyses that involve the meshing of rigid contact elements.

The **MSHPATTERN** command is valid only when you have specified that ANSYS use triangle-shaped elements [**MSHAPE**,1,2D] (or you are meshing with an element that supports only triangles), and you have also specified mapped meshing [**MSHKEY**,1] to mesh an area.

For details about mapped meshing with triangles, see the [Modeling and Meshing Guide](#).

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesher Opts**

**Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options**

## MSMASS, SPNUM, Value

Specifies the mass type for a transient species analysis.

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### SPNUM

Species number, from 1 to 6. Must be specified. If ALL, all numbers are used.

### Value

Mass type for transient species analysis:

#### LUMP

Lumped mass matrix (default).

#### CONS

Consistent mass matrix.

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.



## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species  
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

## MSMETH, *SPNUM*, *KEY*

Specifies the method of solution of the species transport equations.

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### *SPNUM*

Species number, from 1 to 6. Must be specified.

### *KEY*

Key defining the method of solution for the specified species number:

- 0** No solution of equations for species *SPNUM*
- 1** Tri-Diagonal Matrix Algorithm (TDMA).
- 2** Conjugate residual method.
- 3** Preconditioned Conjugate Residual method.
- 4** Preconditioned Generalized Minimum Residual (PGMR) solution method (default).
- 5** Sparse Direct method.
- 6** Preconditioned BiCGStab method (PBCGM).

## Command Default

The Preconditioned Generalized Minimum Residual (PGMR) method is used for all species.

## Notes

The Tri-Diagonal Matrix Algorithm (TDMA) method is a special version of the standard Gauss-Seidel iterative method for the solution of sets of algebraic equations. The number of iterations (sweeps) to be performed is specified with the **MSSOLU** command. No convergence criterion is required for the TDMA method.

Three methods are semi-direct solution methods based on search directions. They are conjugate direction iterative techniques which develop a solution as a linear combination of search directions. The Conjugate Residual method requires the least memory, but stalls when solving ill-conditioned problems. The Preconditioned Conjugate Residual method requires much more memory but performs better for ill-conditioned matrix problems. The PGMR method is memory-intensive; by necessity, it incorporates a tight convergence criterion. The Preconditioned BiCGStab method (PBCGM) requires less memory than the PGMR method and

sometimes performs better than the PGMR method. The number of search vectors and the convergence criterion are specified with the **MSSOLU** command.

The Sparse Direct method is based on Gaussian elimination to factorize the matrix. This method is memory intensive and creates temporary files on the hard disk. It is robust and can be used for symmetric as well as non-symmetric equation systems.

See the *Fluids Analysis Guide* for additional information on the FLOTRAN Solvers.

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**

**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

---

**MSMIR**, *SPNUM*, *Value*

**Sets modified inertial relaxation factors for multiple species.**

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**SPNUM**

Species number, from 1 to 6. Must be specified.

**Value**

Modified inertial relaxation factor. *Value* defaults to 0 (modified inertial relaxation off).

## Notes

*Value* must be a positive real number. A *Value* between 0.1 and 1.0 is recommended. A larger *Value* provides a more robust scheme, but it may yield a slower convergence.

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**

**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

---

**MSNOMF**, *SPNUM*, *FRACTION*

**Specifies the initial value of nominal mass fraction for a species.**

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

***SPNUM***

Species number, from 1 to 6. Must be specified.

***FRACTION***

The initial mass fraction of the entire problem domain for this species. Defaults to 0.0. The sum of the mass fractions for all species should equal 1.0.

**Notes**

This command is valid for the multiple species transport option in a FLOTRAN analysis and is required if the CMIX option has been activated for a property [**FLDATA7**,*PROT*,*Label*,*CMIX*].

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**

**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

**MSPROP**, *SPNUM*, *Label*, *Type*, *NOMINAL*, *COF1*, *COF2*, *COF3*

**Defines the fluid properties of a species.**

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

***SPNUM***

Species number, from 1 to 6. Must be specified.

***Label***

Label identifying the property being defined:

**DENS**

Density.

**VISC**

Viscosity.

**COND**

Thermal conductivity.

**MDIF**

Mass diffusion coefficient.

**SPHT**

Specific heat.

***Type***

Type of property:

**CONSTANT**

Constant property (default). The property does not vary with temperature.

**LIQUID**

Liquid property. Density varies according to a second order polynomial relationship, and all other properties follow Sutherland's law for liquids.

**GAS**

Gas property. Density varies according to the ideal gas law, and all other properties follow Sutherland's law for gases.

**NOMINAL**

Nominal value of the property being defined. For CONSTANT fluid types, the property remains at this value and does not vary. For GAS and LIQUID property types, this is the value of the property corresponding to the temperature defined by *COF1*.

**COF1**

Temperature corresponding to the *NOMINAL* value of the property (for GAS and LIQUID property types only; see Notes section). Not required for *Label* = SPHT.

**COF2, COF3**

Second and third coefficients for temperature variation of the property. Not required for *Label* = SPHT.

**Notes**

If the property type is CONSTANT, the equation used is as follows:

$$Lab = NOMINAL$$

If the property type is LIQUID, a second order polynomial relationship is used for density, and Sutherland's law for liquids is used for the other properties:

$$DENS = NOMINAL + COF2*(T-COF1) + COF3*(T-COF1)^2$$

$$\text{Property} = NOMINAL * \text{EXP}[COF2*(1/T-1/COF1) + COF3*(1/T-1/COF1)^2]$$

where T is the temperature of the node where the property is being calculated.

If the property type is GAS, the ideal gas law is used for density, and Sutherland's law for gases is used for other properties:

$$DENS = NOMINAL * (P/COF2 / (T/COF1))$$

$$\text{Property} = NOMINAL * (T/COF1)^{1.5} * (COF1 + COF2)/(T + COF2)]$$

where P and T are the pressure and temperature of the node where the property is being calculated.

Specific heat is always a CONSTANT. Also, property types (*Type*) such as TABLE, USER, POWL, BIN, etc. are not available for individual species. They are valid only for the bulk fluid.

If a property type is entered incorrectly (e.g. a misspelling), a CONSTANT property type is assumed and the property is assigned the NOMINAL value.

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species

Main Menu>Solution>FLOTRAN Set Up>Multiple Species

## MSQUAD, QDIF, QSRC

Specifies the quadrature order for multiple species elements.

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### QDIF

Quadrature order for diffusion term integration:

**0**

One-point quadrature (default).

**1**

Same as 0, except a distributed value of temperature is used to evaluate temperature-dependent properties.

**2**

Two-point quadrature (default for axisymmetric models).

### QSRC

Quadrature order for source term integration:

**0**

One-point quadrature (default).

**1**

Same as 0, except a distributed value of temperature is used to evaluate temperature-dependent properties.

**2**

Two-point quadrature (default for axisymmetric models).

## Command Default

As described above.

## Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species

Main Menu>Solution>FLOTRAN Set Up>Multiple Species

**MSRELAX**, *SPNUM*, *CONC*, *MDIF*, *EMDI*, *STAB*

**Specifies relaxation factors for a multiple species transport analysis.**

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

***SPNUM***

Species number, from 1 to 6. Must be specified.

***CONC***

Species concentration relaxation factor. Defaults to 0.5.

***MDIF***

Mass diffusion coefficient relaxation factor. Defaults to 0.5.

***EMDI***

Effective mass diffusion coefficient relaxation factor (used for turbulent flow). Defaults to 0.5.

***STAB***

Inertial relaxation factor for solution of the transport equation. Defaults to  $1.0 \times 10^{+20}$ .

**Command Default**

As described above for each relaxation factor.

**Notes**

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**

**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

**MSSOLU**, *SPNUM*, *NSWEEP*, *MAXI*, *NSRCH*, *CONV*, *DELMAX*

**Specifies solution options for multiple species transport.**

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

***SPNUM***

Species number, from 1 to 6. Must be specified.

***NSWEEP***

Number of Tri-Diagonal Matrix Algorithm (TDMA) sweeps. Valid only for the TDMA method [**MSMETH**]. Defaults to 100.

***MAXI***

Maximum number of iterations allowed for the semi-direct methods (conjugate residual and preconditioned conjugate residual methods, chosen with the **MSMETH** command). Defaults to 1000.

***NSRCH***

Number of search vectors used for the semi-direct methods. Defaults to 2. If you are using the Preconditioned BiCGStab Method (PBCGM) solver, the number of search directions is 1 to 8, with 2 as the default. New search vectors are made orthogonal to *NSRCH* previous vectors in the solution of the unsymmetric matrix systems.

***CONV***

Convergence criterion for the semi-direct methods. It represents the factor by which the inner product of the residual vector is reduced during the solution of the equations at any global iteration. Defaults to  $1.0 \times 10^{-12}$ . If the convergence criterion has not been achieved, the algebraic solver issues a warning message, and the execution of FLOTRAN continues normally.

***DELMAX***

Minimum normalized rate of change which will permit the semi-direct solution methods to continue. Used to terminate the semi-direct solvers in the event that stall occurs. Defaults to  $1.0 \times 10^{-9}$ . If the methods stall, the solver increments the solution only a very small amount despite the fact that the correct solution has been not been achieved (or perhaps even approached). The maximum nodal difference between the solutions, normalized to the value of the variable, is compared to *DELMAX*, and the solution is terminated if the value is less than *DELMAX*. Termination of the algebraic solver due to the small rate of change is considered a normal function, and no warning message is printed. Execution of FLOTRAN continues normally.

## Command Default

As described above.

## Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**  
**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

## MSSPEC, *SPNUM*, *Name*, *MOLWT*, *SCHMIDT*

**Specifies the name, molecular weight, and Schmidt number of a species.**

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

***SPNUM***

Species number, from 1 to 6. Must be specified.

***Name***

Name to be assigned to the species, up to 4 characters long. Defaults to SP01 for species 1, SP02 for species 2, ..., SP06 for species 6. This name can be used in place of the species number when specifying mass fraction boundary conditions and in postprocessing.

## Note

The GUI always shows the default names, not the user-defined names.) *Name* should not be the same as an existing degree of freedom label.

### *MOLWT*

Molecular weight for the species. Required only for gases (determined by the property type on **MSPROP** command). Defaults to 29.0.

### *SCHMIDT*

Schmidt number (diffusion term divisor) for the species. Required only for gases (determined by the property type on **MSPROP** command). Defaults to 1.0.

## Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**  
**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

---

## **/MSTART**, *Label*, *KEY*

**Controls the initial GUI components.**

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Label*

Label identifying the GUI component:

#### **ZOOM**

Pan, Zoom, Rotate dialog box, off by default.

#### **WORK**

Offset Working Plane dialog box, off by default.

#### **WPSET**

Working Plane Settings dialog box, off by default.

#### **ABBR**

Edit Toolbar/Abbreviations dialog box, off by default.

#### **PARM**

Scalar Parameters dialog box, off by default.

#### **SELE**

Select Entities dialog box, off by default.

#### **ANNO**

Annotation dialog box, off by default.



**HARD**

Hard Copy dialog box, off by default.

**UTIL**

Turns on the pre-ANSYS 6.1 (UIDL) GUI, off by default.

**KEY**

Switch value:

**OFF or 0**

Component does not appear when GUI is initialized.

**ON or 1**

Component appears when GUI is initialized.

**Command Default**

Same as *Label* defaults.

**Notes**

Controls which components appear when the Graphical User Interface (GUI) is initially brought up. This command is valid only *before* the GUI is brought up [**/MENU,ON**] and is intended to be used in the start130.ans file. It only affects how the GUI is *initialized*; you can always bring up or close any component once you are in the GUI.

This command is valid only at the Begin Level.

**Menu Paths**

**This command cannot be accessed from a menu.**

**MSTERM, SPNUM, STER, TTER**

**Sets the convergence monitors for species.**

[PREP7: FLOTRAN Multiple Species](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**SPNUM**

Species number, from 1 to 6. Must be specified.

**STER**

Termination criteria for steady-state analysis. Defaults to  $1 \times 10^{-8}$ .

**TTER**

Termination criteria for transient analysis. Defaults to  $1 \times 10^{-6}$ .

**Notes**

Repeat command to set each species number as required.

All specified criteria must be met before the case is terminated.

If a termination criterion for a specific species number is set negative, the termination check is ignored for that particular species.

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**  
**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

---

**MSTOLE**, *METHOD*, *Namesurf*, *Namefluid*

Adds two extra nodes from **FLUID116** elements to **SURF151** or **SURF152** elements for convection analyses.

*PREP7: Special Purpose*

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *METHOD*

Mapping method:

**0**

Hybrid method (default).

**1**

Projection method.

**2**

Minimum centroid distance method.

### *Namesurf*

Component name for a group of **SURF151** or **SURF152** elements. The name must be enclosed in single quotes (e.g., 'COM152') when the **MSTOLE** command is manually typed in.

### *Namefluid*

Component name for a group of **FLUID116** elements. The name must be enclosed in single quotes (e.g., 'COM116') when the **MSTOLE** command is manually typed in.

## Notes

For convection analyses, the **MSTOLE** command adds two extra nodes from **FLUID116** elements to **SURF151** or **SURF152** elements by employing the specified mapping method. In the hybrid method, the projection method is tried first and if it fails the centroid distance method is used. The **SURF151** or **SURF152** elements and the **FLUID116** elements must be grouped into components and named using the **CM** command.

The **SURF151** or **SURF152** extra node option must be set for two extra nodes (KEYOPT(5) = 2).

For more information, see [Using the Surface Effect Elements](#) in the *Thermal Analysis Guide*.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**MSVARY**, *SPNUM*, *Lab*, *Key*

**Allows species properties to vary between global iterations.**

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

**SPNUM**

Species number, from 1 to 6. Must be specified.

**Lab**

Label identifying the species property:

**DENS**

Density.

**VISC**

Viscosity.

**COND**

Thermal conductivity.

**MDIF**

Mass diffusion coefficient.

**Key**

Key to allow property variation between global iterations:

**OFF**

Variation not allowed (default).

**ON**

Variation allowed.

**Command Default**

No property is allowed to vary between global iterations within a load step.

**Notes**

This command is valid for the multiple species transport option in a FLOTRAN analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species**

**Main Menu>Solution>FLOTRAN Set Up>Multiple Species**

**\*MULT**, *M1*, *T1*, *M2*, *T2*, *M3*

**Performs the matrix multiplication  $M3 = M1(T1)*M2(T2)$ .**

APDL: Matrix Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *M1*

Name of matrix *M1*. Must have been previously specified by a **\*DMAT** or **\*SMAT** command.

### *T1*

Transpose key. Set *T1* = TRANS to use the transpose of *M1*. If blank, transpose will not be used.

### *M2*

Name of matrix *M2*. Must have been previously specified by a **\*DMAT** command.

### *T2*

Transpose key. Set *T2* = TRANS to use the transpose of *M2*. If blank, transpose will not be used.

### *M3*

Name of resulting matrix, *M3*. Must be specified.

## Notes

The matrices must be dimensionally consistent such that the number of columns of *M1* (or the transposed matrix, if requested) is equal to the number of rows of *M2* (or the transposed matrix, if requested).

You cannot multiply two sparse matrices with this command (that is, *M1* and *M2* cannot both be sparse). The resulting matrix, *M3*, will always be a dense matrix, no matter what combination of input matrices is used (dense\*sparse, sparse\*dense, or dense\*dense).

## Menu Paths

**This command cannot be accessed from a menu.**

---

**\*MWRITE**, *ParR*, *Fname*, *Ext*, --, *Label*, *n1*, *n2*, *n3*

**Writes a matrix to a file in a formatted sequence.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *ParR*

The name of the array parameter. See **\*SET** for name restrictions.

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

If the file name fields are left blank, the default file is the current output file.

**Ext**

Filename extension (8 character maximum).

--

Unused field.

**Label**

Can use a value of IJK, IKJ, JIK, JKI, KIJ, KJI, or blank (JIK).

**n1, n2, n3**

Write as  $((ParR(i,j,k), k = 1, n1), i = 1, n2), j = 1, n3)$  for *Label* = KIJ. *n1*, *n2*, and *n3* default to the corresponding dimensions of the array parameter ParR.

**Notes**

Writes a matrix or vector to a specified file in a formatted sequence. You can also use the **\*VWRITE** command to write data to a specified file. Both commands contain format descriptors on the line immediately following the command. The format descriptors can be in either Fortran or C format.

Fortran format descriptors are enclosed in parentheses. They must immediately follow the **\*MWRITE** command on a separate line of the same input file. The word FORMAT should not be included. The format must specify the number of fields to be written per line, the field width, the placement of the decimal point, etc. There should be one field descriptor for each data item written. The write operation uses the available system FORTRAN FORMAT conventions (see your system FORTRAN manual). Any standard FORTRAN *real* format (such as (4F6.0), (E10.3,2X,D8.2), etc.) and character format (A) may be used. Integer (I) and list-directed (\*) descriptors may *not* be used. Text may be included in the format as a quoted string. *The FORTRAN descriptor must be enclosed in parentheses* and the format must not exceed 80 characters (including parentheses).

The "C" format descriptors are used if the first character of the format descriptor line is not a left parenthesis. "C" format descriptors may be up to 80 characters long, consisting of text strings and predefined "data descriptors" between the strings where numeric or alphanumeric character data are to be inserted. The normal descriptors are %I for integer data, %G for double precision data, %C for alphanumeric character data, and %/ for a line break. There must be one data descriptor for each specified value in the order of the specified values. The enhanced formats described in **\*MSG** may also be used.

The starting array element number must be defined. Looping continues in the directions indicated by the Label argument. The number of loops and loop skipping may also be controlled with the **\*VLEN** and **\*VMASK** commands, which work in the *n2* direction (by row on the output file), and by the **\*VCOL** command, which works in the *n1* direction (by column in the output file). The vector specifications **\*VABS** and **\*VFACT** apply to this command, while **\*VCUM** does not apply to this command. See the **\*VOPER** command for details. If you are in the GUI, the **\*MWRITE** command must be contained in an externally prepared file and read into ANSYS (i.e., **\*USE**, **/INPUT**, etc.).

This command is valid in any processor.

**Menu Paths**

**Utility Menu>Parameters>Array Parameters>Write to File**

**MPXAND**, *NMODE*, *FREQB*, *FREQE*, *Elcalc*, *SIGNIF*, *MSUPkey*

**Specifies the number of modes to expand and write for a modal or buckling analysis.**

SOLUTION: Dynamic Options  
 SOLUTION: Nonlinear Options

MP ME ST PR PRN DS DSS <> EM EH <> PP <> EME MFS

***NMODE***

Number of modes to expand and write. If blank or ALL, expand and write all modes within the frequency range specified. If -1, do not expand and do not write modes to the results file during the analysis.

***FREQB***

Beginning, or lower end, of frequency range of interest. If *FREQB* and *FREQE* are both blank, expand and write the number of modes specified without regard to the frequency range. Defaults to the entire range.

***FREQE***

Ending, or upper end, of frequency range of interest.

***Elcalc***

Element calculation key:

**NO**

Do not calculate element results, reaction forces, and energies (default).

**YES**

Calculate element results, reaction forces, energies, and the nodal degree of freedom solution.

***SIGNIF***

Expand only those modes whose significance level exceeds the *SIGNIF* threshold. The significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not expanded. The higher the *SIGNIF* threshold, the fewer the number of modes expanded. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. *SIGNIF* value is only used for single-point or DDAM response (**SPOPT**,SPRS or DDAM) analyses.

***MSUPkey***

Element result superposition key:

**NO**

Do not write element results to the mode file Jobname.MODE.

**YES**

Write element result to the mode file for use in the expansion pass of a subsequent mode superposition, PSD, transient, or harmonic analysis (default if *Elcalc* = YES).

**Command Default**

Mode shapes (displacements) are expanded and output to the results file (so that if you need to inspect the mode shapes, it is not necessary to run the analysis again). Element stresses are not expanded.

**Notes**

Specifies the number of modes to expand and write over a frequency range for a modal (**ANTYPE**,MODAL) or buckling (**ANTYPE**,BUCKLE) analysis. For reduced analyses, an expansion is required. If used in SOLUTION, this command is valid only within the first load step.

There is no limit on the number of expanded modes (*NMODE*). However, there *is* a limit on the maximum number of modes used via the **\*GET,,MODE** command, [mode combinations](#), and the **MDAMP** command.

With *MSUPkey* = YES, the computed element results (*Elcalc* = YES) are written to *Jobname.MODE* for subsequent use in downstream mode superposition analyses, including harmonic, transient and PSD analyses. This reduces computation time significantly for the combination or expansion passes.

In a Distributed ANSYS analysis, you must issue **MXPAND** to specify the number of modes to expand when computing the modes and mode shapes. In a Distributed ANSYS run, **MXPAND** cannot be issued in an expansion pass (**EXPASS**).

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Expand Modes**

**Main Menu>Solution>Analysis Type>Analysis Options**

**Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Expand Modes**

---





---

## N Commands

**N**, *NODE*, *X*, *Y*, *Z*, *THXY*, *THYZ*, *THZX*

**Defines a node.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NODE**

Node number to be assigned. A previously defined node of the same number will be redefined. Defaults to the maximum node number used +1.

**X, Y, Z**

Node location in the active coordinate system ( $R, \theta, Z$  for cylindrical,  $R, \theta, \Phi$  for spherical or toroidal). If  $X = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**THXY**

First rotation about nodal Z (positive X toward Y).

**THYZ**

Second rotation about nodal X (positive Y toward Z).

**THZX**

Third rotation about nodal Y (positive Z toward X).

### Notes

Defines a node in the active coordinate system [**CSYS**]. The nodal coordinate system is parallel to the global Cartesian system unless rotated. Rotation angles are in degrees and redefine any previous rotation angles. See the **NMODIF**, **NANG**, **NROTAT**, and **NORA** commands for other rotation options.

### Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Nodes>In Active CS**

**Main Menu>Preprocessor>Modeling>Create>Nodes>On Working Plane**

---

**NANG**, *NODE*, *X1*, *X2*, *X3*, *Y1*, *Y2*, *Y3*, *Z1*, *Z2*, *Z3*

**Rotates a nodal coordinate system by direction cosines.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

**NODE**

Rotate coordinate system of this node.

**X1, X2, X3**

Global X, Y, Z components of a unit vector in new nodal X direction.

**Y1, Y2, Y3**

Global X, Y, Z components of a unit vector in new nodal Y direction.

**Z1, Z2, Z3**

Global X, Y, Z components of a unit vector in new nodal Z direction.

**Notes**

Rotates a nodal coordinate system to the orientation specified by the X, Y and Z direction cosines. Existing rotation specifications on the node are redefined. If only two of the three unit vectors are specified, the third is defined according to the right hand rule. It is the responsibility of the user to ensure that input direction cosines are orthogonal in a right-handed system.

See the **NMODIF**, **NROTAT**, and **NORA** commands for other rotation options.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Nodes>RotateNode>By Vectors**

**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>By Vectors**

**NAXIS**, *Action*, *Val*

**Generates nodes for general axisymmetric element sections.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

**Action**

Specifies one of the following command behaviors:

**GEN**

Generates nodes around the axis of an axisymmetric section (default).

**CLEAR**

Clears all nodes around the axis of an axisymmetric section.

**EFACET**

Specifies the number of facets per edge between nodal planes and [integration planes](#) in the circumferential direction to display using PowerGraphics. This option is only valid with [/ESHAPE,1](#) and [RSYS,SOLU](#) commands.

**Val**

Tolerance value *or* number of facets per edge:

**TOLER**

When *Action* = GEN, the tolerance to use for merging the generated nodes around the axis.

**NUM**

When *Action* = EFACET, the number of facets per element edge for element plots:

**AUTO**

Use program-chosen facets per edge (default).

**1**

Use 1 facet per edge (default for elements with 9, 10, 11, or 12 nodal planes). Shows nodal and integration planes only.

- 2**  
Use 2 facets per edge (default for elements with 5, 6, 7, or 8 nodal planes, and maximum for elements with 9, 10, 11, or 12 nodal planes).
- 3**  
Use 3 facets per edge (default for elements with 3 or 4 nodal planes, and maximum for elements with 6, 7, or 8 nodal planes).
- 4**  
Use 4 facets per edge (maximum for elements with 5 nodal planes).
- 5**  
Use 5 facets per edge (maximum for elements with 4 nodal planes).
- 6**  
Use 6 facets per edge (maximum for elements with 3 nodal planes).

## Notes

The **NAXIS** command generates or clears the nodes for general axisymmetric element sections. The command applies to elements [SURF159](#), [SOLID272](#), and [SOLID273](#).

The generate option (*Action* = GEN) operates automatically on any current-technology axisymmetric element. Any nodes within the tolerance value (*TOLER*) of the axis are merged into a single node. The default tolerance is 1.0e-4.

If you want to change the number of nodes, use the clear option (*Action* = CLEAR) before regenerating the nodes.

To cause the 3-D element plot to appear more like the actual 3-D model, use **NAXIS,EFACET,NUM**, where *NUM* > 1. In this case, the coordinate system specified for displaying element and nodal results (*RSYS*) must be solution (**RSYS,SOLU**); otherwise, ANSYS resets *NUM* to 1.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Nodes>About AXIS Section**

**NCNV**, *KSTOP*, *DLIM*, *ITLIM*, *ETLIM*, *CPLIM*

**Sets the key to terminate an analysis.**

[SOLUTION: Nonlinear Options](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### **KSTOP**

Program behavior upon nonconvergence:

- 0**  
Do not terminate the analysis if the solution fails to converge.
- 1**  
Terminate the analysis and the program execution if the solution fails to converge (default).
- 2**  
Terminate the analysis, but not the program execution, if the solution fails to converge.

**DLIM**

Terminates program execution if the largest nodal DOF solution value (displacement, temperature, etc.) exceeds this limit. Defaults to 1.0E6 for all DOF except MAG and A and FLOTRAN DOF. Defaults to 1.0E10 for MAG and A. Defaults to 1.0E20 for the FLOTRAN VX, VY, VZ, PRES, and TEMP DOF. Does not apply to the FLOTRAN ENKE and ENDS DOF.

**ITLIM**

Terminates program execution if the cumulative iteration number exceeds this limit (defaults to infinity).

**ETLIM**

Terminates program execution if the elapsed time (seconds) exceeds this limit (defaults to infinity).

**CPLIM**

Terminates program execution if the CPU time (seconds) exceeds this limit (defaults to infinity).

## Command Default

As described for each argument above.

## Notes

Sets the key to terminate an analysis if not converged, or if any of the following limits are exceeded for nonlinear and full transient analyses: DOF (displacement), cumulative iteration, elapsed time, or CPU time limit. Applies only to static and transient analyses (**ANTYPE**,STATIC and **ANTYPE**,TRANS), and FLOTRAN analyses. Time limit checks are made at the end of each equilibrium iteration. This command is ignored during an optimization run.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Criteria to Stop**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL**

**Main Menu>Solution>Load Step Opts>Nonlinear>Criteria to Stop**

---

## NDELE, NODE1, NODE2, NINC

**Deletes nodes.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NODE1, NODE2, NINC**

Delete nodes from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and all selected nodes [**NSSEL**] are deleted. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*.

## Notes

Deletes selected nodes that are not connected to elements. Nodes may also be redefined instead of deleted, if desired. Boundary conditions (displacements, forces, etc.) as well as any coupling or constraint equations containing the deleted nodes are also deleted.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements**  
**Main Menu>Preprocessor>Modeling>Delete>Nodes**

## NDIST, *ND1*, *ND2*

**Calculates and lists the distance between two nodes.**

[PREP7: Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *ND1*

First node in distance calculation. If *ND1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### *ND2*

Second node in distance calculation.

## Notes

**NDIST** lists the distance between nodes *ND1* and *ND2*, as well as the current coordinate system offsets from *ND1* to *ND2*, where the X, Y, and Z locations of *ND1* are subtracted from the X, Y, and Z locations of *ND2* (respectively) to determine the offsets. **NDIST** is valid in any coordinate system except toroidal [**CSYS**,3].

**NDIST** returns a variable, called "*\_RETURN*," which contains the distance value. You can use this value for various purposes, such as the calculation of distributed loads. In interactive mode, you can access this command by using the Model Query Picker (**Utility Menu> List> Picked Entities**), where you can also access automatic annotation functions and display the value on your model.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Check Geom>ND distances**

**NDSURF**, *Snode*, *Telem*, *DIMN*

**Generates surface elements overlaid on the edge of existing elements and assigns the extra node as the closest fluid element node.**

[PREP7: Elements](#)

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

***Snode***

Component name for the surface nodes of the solid elements.

***Telem***

Component name for the target fluid elements.

***DIMN***

Model dimensionality:

**2**

2-D model.

**3**

3-D model.

## Notes

This command macro is used to generate surface effect elements ([SURF151](#) or [SURF152](#)) overlaid on the surface of existing plane or solid elements and, based on proximity, to determine and assign the extra node ([FLUID116](#)) for each surface element. The active element type must be [SURF151](#) or [SURF152](#) with appropriate settings for [KEYOPT\(4\)](#), [KEYOPT\(5\)](#), [KEYOPT\(6\)](#), and [KEYOPT\(8\)](#).

The surface nodes of the plane or solid elements must be grouped into a node component and the fluid elements must be grouped into an element component and named using the **CM** command. The names must be enclosed in single quotes (e.g., 'NOD') when the **NDSURF** command is manually typed in.

When using the GUI method, node and element components are created through the picking dialog boxes associated with this command.

The macro is applicable for the [SURF151](#), [SURF152](#), and [FLUID116](#) element types.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Area to Fluid**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Line to Fluid**

**Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Node to Fluid**

**NEQIT**, *NEQIT*, *FORCEkey*

**Specifies the maximum number of equilibrium iterations for nonlinear analyses.**

[SOLUTION: Nonlinear Options](#)

MP ME ST PR PRN <> <> <> EM <> DY PP <> EME MFS

**NEQIT**

Maximum number of equilibrium iterations allowed each substep.

**FORCEkey**

One iteration forcing key (used only when *NEQIT* = 1 and [SOLCONTROL,ON](#)):

**FORCE**

Forces one iteration per substep.

**(blank)**

[SOLCONTROL](#) controls the minimum number of iterations (internally set to two iterations per substep if [SOLCONTROL,ON](#)).

**Note**

Using one iteration per substep may result in unconverged solutions for nonlinear analysis, and the program may not indicate divergence in this case. This option is intended primarily for use by the ANSYS Workbench interface. Keep in mind that forcing one iteration per substep is only recommended under very specific conditions; for example, nonlinearity in bonded penalty type contact. Under these conditions the solution typically converges in one iteration.

**Command Default**

When [SOLCONTROL,ON](#), *NEQIT* is set between 15 and 26 depending on the physics of the problem. When [SOLCONTROL,OFF](#), *NEQIT* defaults to 25 for all cases.

**Notes**

See the description of [SOLCONTROL](#) for a complete listing of the defaults set by [SOLCONTROL,ON](#) and [SOLCONTROL,OFF](#).

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Equilibrium Iter**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear**

**Main Menu>Solution>Load Step Opts>Nonlinear>Equilibrium Iter**

**/NERR**, *NMERR*, *NMABT*, *--*, *IFKEY*, *NUM*

### Limits the number of warning and error messages displayed.

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ***NMERR***

Maximum number of warning and error messages displayed per command. Defaults to 5 for interactive runs with the GUI turned on, 20 for interactive runs with the GUI turned off, 200 for batch runs. If *NMERR* is negative, the absolute value of *NMERR* is used as the maximum number of warning and error messages written to the error file (`file.ERR`) per command, as well as the maximum number of messages displayed per command.

#### ***NMABT***

Maximum number of warning and error messages allowed per command before run aborts (must be greater than zero). Maximum value is 99,999,999. Defaults to 10,000.

***--***

Unused field.

#### ***IFKEY***

Specifies whether or not to abort if an error occurs during a **/INPUT** operation:

##### **0 or OFF**

Do not abort. This option is the default.

##### **1 or ON**

Abort.

#### ***NUM***

The number of invalid command warnings before a stop warning will be issued:

##### **0**

Disables the stop warning/error function.

##### ***n***

An integer value representing the number of warnings that will be encountered before prompting the user to stop (default = 5). The first error encountered will ALWAYS result in a prompt.

---

#### **Note**

Invalid command warnings and error tracking are mutually exclusive.

## **Command Default**

As described above.

## **Notes**

Limits the number of warning and error messages displayed for any one command in an interactive run.

Warning and error messages continue to be written to `Jobname.ERR` regardless of these limits (unless *NMERR* is negative).



Issue this command with  $NUM = n$  to specify the number of "invalid command" warnings to be encountered before the user is prompted to stop. You can then continue or abort the run. If you choose to abort the run, the log file can be saved so that any of the processing up to that point can be appended to an input that rectifies the condition. A batch run always aborts on the first error. Issue **/NERR,STAT** to list current settings.

Issue **/NERR,DEFA** to reset values to initial defaults.

An *IFKEY* value of 1 or ON causes the ANSYS program to abort immediately upon encountering an error during a file **/INPUT** operation. However, use of this option may cause the following conditions to occur:

- The **/INPUT** command may abort if issued for a log file (*jobname.log*).
- Some macros may abort.
- A CAD connection may fail after reading only a small portion of a CAD model.

The command is valid in any processor.

## Menu Paths

Utility Menu>MenuCtrls>Message Controls

## NFORCE, ITEM

Sums the nodal forces and moments of elements attached to nodes.

POST1: Special Purpose

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### ITEM

Specifies the selected set of nodes for summing forces and moments for contact elements.

#### (blank)

Sums the nodal forces of elements for all selected nodes and excludes contact elements (elements 169-177).

#### CONT

Sums the nodal forces of elements for contact nodes only.

#### BOTH

Sums the nodal forces of elements for all selected nodes, including contact elements.

## Notes

Sums and prints, in each component direction for each selected node, the nodal force and moment contributions of the selected elements attached to the node. If all elements are selected, the sums are usually zero except where constraints or loads are applied. The nodal forces and moments may be displayed [**/PBC,FORC** and **/PBC,MOME**]. Use **PRESOL** to print nodal forces and moments on an element-by-element basis. You can use the **FORCE** command to specify which component (static, damping, inertia, or total) of the nodal load is to be used. Nodal forces associated with surface loads are not included.

This vector sum is printed in the global Cartesian system. Moment summations are about the global origin unless another point is specified with the **SPOINT** command. The summations for each node are printed in the global Cartesian system unless transformed [**RSYS**]. This command is generally not applicable to

axisymmetric models because moment information from the **NFORCE** command is not correct for axisymmetric elements.

Selecting a subset of elements [**ESEL**] and then issuing this command will give the forces and moments required to maintain equilibrium of that set of elements. The effects of nodal coupling and constraint equations are ignored. The option *ITEM* = **CONT** provides the forces and moments for the contact elements (**CONTA171**, **CONTA172**, **CONTA173**, **CONTA174**, **CONTA175**, **CONTA176**, and **CONTA177**). Setting *ITEM* = **BOTH** provides the forces and moments for all selected nodes, including contact elements.

This command also includes the **FSUM** command function which vectorially sums and prints, in each component direction for the total selected node set, the nodal force and moment contributions of the selected elements attached to the selected node set.

## Using NFORCE in a Spectrum or PSD Analysis (**ANTYPE**, **SPECTR**)

When using **NFORCE** in a spectrum analysis after the combination file has been input (**/INPUT**,,**MCOM**), or in a PSD analysis when postprocessing 1-sigma results (loadstep 3, 4, or 5), the following message will display in the printout header:

```
(Spectrum analysis summation is used)
```

This message means that the summation of the element nodal forces is performed prior to the combination of those forces. In this case, **RSYS** does not apply. The forces are in the nodal coordinate systems, and the vector sum is always printed in the global coordinate system.

Because modal displacements cannot be used to calculate contact element nodal forces, *ITEM* does not apply to spectrum and PSD analyses.

## Menu Paths

**Main Menu>General Postproc>Nodal Calcs>Sum @ Each Node**

**NGEN**, *ITIME*, *INC*, *NODE1*, *NODE2*, *NINC*, *DX*, *DY*, *DZ*, *SPACE*

**Generates additional nodes from a pattern of nodes.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***ITIME*, *INC***

Do this generation operation a total of *ITIME* times, incrementing all nodes in the given pattern by *INC* each time after the first. *ITIME* must be > 1 for generation to occur.

### ***NODE1*, *NODE2*, *NINC***

Generate nodes from the pattern of nodes beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = **ALL**, *NODE2* and *NINC* are ignored and the pattern is all selected nodes [**NSEL**]. If *NODE1* = **P**, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

### ***DX*, *DY*, *DZ***

Node location increments in the active coordinate system (*DR*, *Dθ*, *DZ* for cylindrical, *DR*, *Dθ*, *DΦ* for spherical or toroidal).

**SPACE**

Spacing ratio. Ratio of last division size to first division size. If > 1.0, divisions increase. If < 1.0, divisions decrease. Ratio defaults to 1.0 (uniform spacing).

**Note**

The *average* spacing ratio remains 1.0, such that the location of the last generated set will be the same regardless of *SPACE*. *SPACE* only serves to skew the position of the nodes between the pattern set and the last set.

**Notes**

Generates additional nodes from a given node pattern. Generation is done in the active coordinate system. Nodes in the pattern may have been generated in any coordinate system.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Copy>Nodes>Copy**

**NKPT**, *NODE*, *NPT*

**Defines a node at an existing keypoint location.**

[PREP7: Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NODE**

Arbitrary reference number for node. If zero or blank, defaults to the highest node number +1 [**NUMSTR**].

**NPT**

Keypoint number defining global X, Y, Z location. If *NPT* = All, then a node will be placed at each selected keypoint. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Nodes>On Keypoint**

**NLDIAG**, *Label*, *Key*

**Sets nonlinear diagnostics functionality.**

[SOLUTION: Nonlinear Options](#)

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

**Label**

Specifies a diagnostic function:

**NRRE**

Store the Newton-Raphson residuals information.

**EFLG**

Identify or display elements or nodes that violate the criteria.

**MAXF**

Set a maximum number of files (*MaxFile*) to create. Information is written to *Jobname.ndxxx* or *Jobname.nrxxx*, where *xxx* iterates from 001 through *MaxFile*. When the maximum number of files is reached, the counter resets to 001 and earlier files are overwritten.

**CONT**

Write contact information to a single *Jobname.cnd* text file during solution.

**Key**

Sets the characteristics of the diagnostic function:

**OFF, or 0**

Suppresses writing of diagnostic information. This value is the default.

**ON, or 1**

Writes diagnostic information to the *Jobname.ndxxx*, *Jobname.nrxxx*, or *Jobname.cnd* file. (If *Label* = CONT, this option is the same as the SUBS option described below.)

**ITER**

Writes contact diagnostic information at each iteration. Valid only when *Label* = CONT.

**SUBS**

Writes contact diagnostic information at each substep. Valid only when *Label* = CONT.

**LSTP**

Writes contact diagnostic information at each load step. Valid only when *Label* = CONT.

**STAT**

Lists information about the diagnostic files in the current working directory.

**DEL**

Deletes all diagnostic files in the current working directory.

**MaxFile**

Sets the maximum number of files to create (*Jobname.ndxxx* or *Jobname.nrxxx*). Valid values are 1 through 999. The default is 4. (If you change the default *MaxFile* value, that value is retained until you change it again.) Valid only when *Label* = MAXF.

## Command Default

No nonlinear diagnostic files are written.

## Notes

The **NLDIAG** command is a [nonlinear diagnostics](#) tool valid for nonlinear structural analyses. It is a debugging tool for use when you must restart after an unconverged solution. The command creates *Jobname.ndxxx*, *Jobname.nrxxx*, or *Jobname.cnd* files in the working directory to store the information you specify.

**Newton-Raphson residual** Issue the **NLDIAG,NRRE,ON** command to create *Jobname.nrxxx* files which store the relevant Newton-Raphson residual information for the last *MaxFile* equilibrium iterations. Issue a **NLDPOST,NRRE,STAT** command to list the load step, substep, time, and equilibrium iteration corresponding to each of the *Jobname.nrxxx* files in the working directory, then issue a **PLNSOL,NRRES,,,FileID** command to point to the file from which you want to create a contour plot of your Newton-Raphson residuals.

For **NLDIAG,NRRE,ON**, any `Jobname.nrxxx` files in the current (working) directory are overwritten when you issue a new **SOLVE** command (or restart).

**Element components that violate criteria** Issue a **NLDIAG,EFLG,ON** command to create `Jobname.ndxxx` files which store IDs for elements violating the following criteria:

- Too large a distortion
- Elements contain nodes that have near zero pivots for nonlinear analyses
- Too large a plastic/creep strain increment (**CUTCONTROL**)
- Elements for which mixed u-P constraints are not satisfied (mixed U-P option of 18x solid elements only)

For **NLDIAG,EFLG,ON**, all `Jobname.ndxxx` diagnostic files in the current (working) directory are deleted when you issue a new **SOLVE** command (or restart).

In the solution processor (**/SOLU**), use the **STAT** option to list the active status of this command. In the postprocessor (**/POST1**), issue a **NLDPOST,EFLG,STAT** command to list the load step, substep, time, and equilibrium iteration corresponding to each of the `Jobname.ndxxx` files in the working directory, then issue a **NLDPOST,EFLG,CM,FileID** command to create element components that violate the criteria.

**Contact diagnostic** Issue the **NLDIAG,CONT,ON** command to create a `Jobname.cnd` file which stores contact information for all defined contact pairs at all substeps. Alternatively, you may issue one of the following commands to store contact information at a specific frequency:

- **NLDIAG,CONT,ITER** to write at each iteration
- **NLDIAG,CONT,SUBS** to write at each substep (default)
- **NLDIAG,CONT,LSTP** to write at each load step

File `Jobname.cnd` is written during solution and lists, on a pair-base, the following contact information:

- Number of contact elements in contact<sup>[1]</sup>
- Number of contact elements in "sticking" contact status
- Maximum chattering level
- Maximum contact penetration/Minimum gap<sup>[2]</sup>
- Maximum closed gap
- Maximum normal contact stiffness
- Minimum normal contact stiffness
- Maximum resulting pinball
- Maximum elastic slip distance
- Maximum tangential contact stiffness
- Minimum tangential contact stiffness
- Maximum sliding distance
- Maximum contact pressure
- Maximum friction stress

- Average contact depth
  - Maximum closed penetration
  - Number of contact points having too much penetration
  - Contacting area
1. If >0, this quantity indicates the number of contact elements in contact. Other values are interpreted as follows:
    - 0 indicates that the contact pair is in near-field contact status.
    - 1 indicates that the contact pair is in far-field contact status.
    - 2 indicates that the contact pair is inactive ([symmetric to asymmetric contact](#))
  2. A positive value indicates penetration and a negative value indicates a gap. If the contact pair has a far-field contact status, penetration and gap are not available and the value stored is the current pinball radius.

In the solution processor (**/SOLU**), use the **NLDIAG,CONT,STAT** command to list the active status of the contact information. If you subsequently issue a new **SOLVE** command (or restart), the `Jobname.cnd` file in the current (working) directory is not deleted; information is appended to it. Delete the existing file (**NLDIAG,CONT,DEL** command) if you do not want to retain diagnostic information from previous solutions.

For more information, see [nonlinear diagnostics](#).

**Distributed ANSYS Restriction** The diagnostic function EFLG (identify or display elements or nodes that violate the criteria) is not supported in Distributed ANSYS. All other diagnostic functions are supported.

## Menu Paths

**Main Menu>Solution>Diagnostics>Nonlinear Diagnostics**

---

### **NLDPOST**, *Label, Key, FileID, Prefix*

**Gets element component information from nonlinear diagnostic files.**

[POST1: Special Purpose](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### **Label**

Specifies the type of command operation:

#### **EFLG**

Element flag for nonlinear diagnostics.

#### **NRRE**

Newton-Raphson residuals.

#### **Key**

Specifies the command action:

#### **STAT**

List information about the diagnostic files (`Jobname.ndxxx` or `Jobname.nrxxx`) in the current directory.

For *Label* = EFLG, the listing gives a summary that associates the loadstep, substep, time, equilibrium iteration number, cumulative iteration number, and the number of elements that fail each criteria with a specific file ID (*Jobname.ndxxx*). Use the list to create element components (via the CM option) based on the cumulative iteration number.

For *Label* = NRRE, the listing provides a summary that associates the loadstep, substep, time, equilibrium iteration number, and cumulative iteration number with a specific file ID (*Jobname.nrxxx*). Use the list to identify the respective file ID for creating Newton-Raphson residual contour plots (**PLNSOL**,NRRE,...,*FileID*).

**DEL**

Delete *Jobname.ndxxx* or *Jobname.nrxxx* files in the working directory, if any exist.

**CM**

Create components for elements that violate criteria. This value is valid only when *Label* = EFLG.

**FileID**

Valid only when *Label* = EFLG and *Key* = CM, this value specifies file IDs:

**IDnum**

The file ID number. Creates the element components from the diagnostic files corresponding to the specified file ID number in the working directory.

**ALL**

Creates element components from all available diagnostic files residing in the working directory. This value is the default if you do not specify an *IDnum* value.

**Prefix**

Sets the prefix name for components. Specify up to 21 alphanumeric characters.

## Command Default

The **NLDPOST** command returns no nonlinear diagnostic information.

## Notes

Based on the [nonlinear diagnostic](#) results (created via the **NLDIAG**,EFLG command), the **NLDPOST** command creates element components with predefined names.

The following table lists the diagnostic criteria and component names (with specified prefix and without). Here *xxx* corresponds to the file ID (*FileID*) of *Jobname.ndxxx* or *Jobname.nrxxx*.

Elements with:	If prefix is specified:	Without prefix specified:
Too large a plastic strain increment	<i>prefix_xxx_eppl</i>	ndxxx_eppl
Too large a creep strain increment	<i>prefix_xxx_epcr</i>	ndxxx_epcr
Unsatisfied mixed u-P constraints	<i>prefix_xxx_mxup</i>	ndxxx_mxup
Too much distortion	<i>prefix_xxx_hdst</i>	ndxxx_hdst
Negative/small pivots	<i>prefix_xxx_pivt</i>	ndxxx_pivt

For more information, see [Performing Nonlinear Diagnostics](#).

## Menu Paths

Main Menu>General Postproc>Nonlinear Diagnostics

### NLGEOM, Key

Includes large-deflection effects in a static or full transient analysis.

SOLUTION: Nonlinear Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions

#### Key

Large-deflection key:

#### OFF

Ignores large-deflection effects (that is, a small-deflection analysis is specified). This option is the default.

#### ON

Includes large-deflection (large rotation) effects or large strain effects, according to the element type.

## Command Default

Large-deflection effects are ignored.

## Notes

Large-deflection effects are categorized as either large deflection (or large rotation) or large strain, depending on the element type. These are listed (if available) under Special Features in the input data table for each element in the *Element Reference*. If used during the solution (**/SOLU**), this command is valid only within the first load step.

In a large-deflection analysis, pressure loads behave differently than other load types. For more information, see [Load Direction in a Large-Deflection Analysis](#).

The gyroscopic matrix (that occurs due to rotational angular velocity) does not support large-deflection effects. The theoretical formulations for the gyroscopic matrix support small deflection (linear formulation) only.

This command is also valid in PREP7.

## Product Restrictions

In ANSYS Professional NLT, large deflection effects should not be turned on if 2-D solid (PLANE $n$ ) or 3-D solid (SOLID $n$ ) elements are defined. ANSYS Professional NLS supports NLGEOM,ON for plane and solid elements.

## Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic

Main Menu>Solution>Analysis Type>Analysis Options



---

**Main Menu>Solution>Analysis Type>Sol'n Controls>Basic**


---

**NLHIST**, *Key, Name, Item, Comp, NODE, ELEM, SHELL, LAYER*

**Specify result items to track during solution.**

SOLUTION: Nonlinear Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**Key**

Specifies the command operation:

**NSOL**

Nodal solution data.

**ESOL**

Element nodal data.

**PAIR**

Contact data (pair-based).

**STAT**

Displays a list of items to track.

**OFF or 0**

Deactivates tracking of all variables. This value is the default.

**ON or 1**

Activates tracking of all variables. Tracking also activates whenever any specification changes.

**DEL**

Removes the specified variable from the set of result items to track. If *Name* = ALL (default), all specifications are removed.

**Name**

The 32-character user-specified name.

**Item, Comp**

Predetermined output item and component label for valid elements. See the [Element Reference](#) for more information.

**NODE**

Valid node number (if *Key* = NSOL or ESOL), or valid real constant set number (if *Key* = PAIR) identifying a contact pair. Required input for NSOL, ESOL, and PAIR items.

**ELEM**

Valid element number for element results. Used for ESOL items. If *ELEM* is specified, then a node number that belongs to the element must also be specified in the *NODE* field.

**SHELL**

Valid labels are TOP, MID or BOT. This field can specify the location on shell elements for which to retrieve data. Used only for element nodal data (ESOL).

**LAYER**

Layer number (for layered elements only). Used only for element nodal data (ESOL).

## Notes

The **NLHIST** command is a [nonlinear diagnostics](#) tool allowing you to monitor diagnostics results of interest in real time during solution.

You can track a maximum of 50 variables during solution. The specified result quantities are written to the file `Jobname.nlh`. Nodal results and contact results are written for every converged substep (irrespective of the **OUTRES** command setting) while element results are written only at time points specified via the **OUTRES** command. For time points where element results data is not available, a very small number is written instead. If the conditions for contact to be established are not satisfied, 0.0 will be written for contact results.

Results tracking is available only for a nonlinear structural analysis (static or transient), a nonlinear steady-state thermal analysis, or a transient thermal analysis (linear or nonlinear). All results are tracked in the Solution Coordinate System (that is, nodal results are in the nodal coordinate system and element results are in the element coordinate system). Results tracking is not available for FLOTRAN analyses.

The `Jobname.nlh` file is an ASCII file that lists each time point at which a converged solution occurs along with the values of the relevant result quantities.

The GUI option **Solution > Results tracking** provides an interface to define the result items to be tracked. The GUI also allows you to graph one or more variables against time or against other variables during solution. You can use the interface to graph or list variables from any `.nlh` file generated by the ANSYS program.

You can also track results during batch runs. Either access the [ANSYS Launcher](#) and select **File Tracking** from the **Tools** menu, or type `nlhist130` at the command line. Use the supplied file browser to navigate to your `Jobname.nlh` file, and click on it to invoke the tracking utility. You can use this utility to read the file at any time, even after the solution is complete (the data in the file must be formatted correctly).

Specifications that you set via the **NLHIST** command are not saved in the database (`.db` file).

**Table 225 NLHIST - Valid NSOL Item and Component Labels**

Item	Comp	Description
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
F	X, Y, Z	X, Y, or Z structural reaction force.
M	X, Y, Z	X, Y, or Z structural reaction moment.
TEMP [1]	-	Temperature.
TEMP	MAX, MIN	Maximum or minimum temperature in the model.
HEAT [2]	-	Reaction heat flow.

1. For [SHELL131](#) and [SHELL132](#) elements with `KEYOPT(3) = 0` or `1`, use the labels `TBOT`, `TE2`, `TE3`, . . . , `TTOP` instead of `TEMP`.

2. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels HBOT, HE2, HE3, . . . , HTOP instead of HEAT.

**Table 226 NLHIST - ValidESOL Item and Component Labels**

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.

ETABLE items are not supported for **ESOL** items.

PAIR solution quantities are output on a "per contact pair" basis. As a consequence, the corresponding values listed in the `Jobname.nlh` file represent a minimum or a maximum over the associated contact pair, as detailed in the table below.

**Table 227 NLHIST - Valid PAIR Item and Component Labels**

Item	Comp	Description
CONT	ELCN	If >0, number of contact elements in contact. Other values are interpreted as follows:  0 indicates the contact pair is in near-field contact status. -1 indicates the contact pair is in far-field contact status. -2 indicates that the contact pair is inactive ( <a href="#">symmetric to asymmetric contact</a> ).
"	ELST	Number of contact elements in sticking contact status.
"	CNOS	Maximum chattering level.
"	PENE	Maximum penetration (or minimum gap). [1]
"	CLGP	Maximum closed (geometrical) gap.
"	SLID	Maximum total sliding distance.
"	ESLI	Maximum elastic slip distance.
"	KNMX	Maximum normal contact stiffness.
"	KTMX	Maximum tangential contact stiffness.
"	KNMN	Minimum normal contact stiffness.
"	KTMN	Minimum tangential contact stiffness.
"	PINB	Maximum pinball radius.
"	PRES	Maximum contact pressure.
"	SFRI	Maximum frictional stress.
"	CNDP	Average contact depth.
"	CLPE	Maximum closed (geometrical) penetration.
"	LGPE	Number of contact points having too much penetration.
"	CAREA	Contacting area.

1. For PENE, a positive value indicates a penetration, and a negative value indicates a gap. If the contact pair has a far-field contact status, penetration and gap are not available, and the value stored for PENE is the current pinball radius.

## Menu Paths

**Main Menu>Solution>Results Tracking**

**NLIST**, *NODE1*, *NODE2*, *NINC*, *Lcoord*, *SORT1*, *SORT2*, *SORT3*, *KINTERNAL*

**Lists nodes.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NODE1*, *NODE2*, *NINC***

List nodes from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL (default), *NODE2* and *NINC* are ignored and all selected nodes [**NSEL**] are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

***Lcoord***

Coordinate listing key:

**(blank)**

List all nodal information

**COORD**

Suppress all but the XYZ coordinates (shown to a higher degree of accuracy than when displayed with all information).

***SORT1***

First item on which to sort. Valid item names are NODE, X, Y, Z, THXY, THYZ, THXZ

***SORT2*, *SORT3***

Second and third items on which to sort. Valid item names are the same as for *SORT1*.

***KINTERNAL***

Internal nodes listing key:

**(blank)**

List only external nodes.

**INTERNAL**

List all nodes, including internal nodes.

## Notes

Lists nodes in the active display coordinate system [**DSYS**]. Nodal coordinate rotation angles are also listed (relative to the global Cartesian coordinate system).

Node listing can be in a sorted order (ascending). *SORT2*, for example, will be carried out on nodes having equal values of *SORT1*.

This command is valid in any processor.

## Menu Paths

**Main Menu>General Postproc>List Results>Sorted Listing>Sort Nodes**

**Utility Menu>List>Nodes**

**NLOG**, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*, *FACTB*

**Forms the natural log of a variable.**

[POST26: Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to NV [\[NUMVAR\]](#)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

***IA***

Reference number of the variable to be operated on.

--, --

Unused fields.

***Name***

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --

Unused fields.

***FACTA***

Scaling factor applied to variable *IA* (defaults to 1.0).

***FACTB***

Scaling factor (positive or negative) applied to the operation (defaults to 1.0).

## Notes

Forms the natural log of a variable according to the operation:

$$IR = FACTB * LN(FACTA \times IA)$$

## Menu Paths

**Main Menu>TimeHist Postpro>Math Operations>Natural Log**

---

## NLOPT

**Specifies "Nonlinear analysis options" as the subsequent status topic.**

[SOLUTION: Status](#)

MP ME ST PR PRN <> <> FL EM <> DY PP <> EME MFS

## Notes

This is a status [\[STAT\]](#) topic command. Status topic commands are generated by the GUI and will appear in the log file (*Jobname.LOG*) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>Solution>Nonlinear Options**

**NMODIF**, *NODE*, *X*, *Y*, *Z*, *THXY*, *THYZ*, *THZX*

**Modifies an existing node.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **NODE**

Modify coordinates of this node. If ALL, modify coordinates of all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

### **X, Y, Z**

Replace the previous coordinate values assigned to this node with these corresponding coordinate values. Values are interpreted in the active coordinate system (R,  $\theta$ , Z for cylindrical; R,  $\theta$ ,  $\Phi$  for spherical or toroidal). Leaving any of these fields blank retains the previous value(s).

### **THXY**

First rotation of nodal coordinate system about nodal Z (positive X toward Y). Leaving this field blank retains the previous value.

### **THYZ**

Second rotation of nodal coordinate system about nodal X (positive Y toward Z). Leaving this field blank retains the previous value.

### **THZX**

Third rotation of nodal coordinate system about nodal Y (positive Z toward X). Leaving this field blank retains the previous value.

## Notes

Modifies an existing node. Nodal coordinate system rotation angles are in degrees and redefine any existing rotation angles. Nodes can also be redefined with the **N** command.

See the **NROTAT**, **NANG**, and **NORA** commands for other rotation options.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Nodes>RotateNode>By Angles**

**Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>Set of Nodes**

**Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>Single Node**

**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>By Angles**

**NOCOLOR**, *KEY*

Removes color from graphics displays.

[DISPLAY: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**KEY**

Color key:

**0**

Color the displays.

**1**

Do not color the displays.

**2**

Do not shade the displays.

**Command Default**

Color graphics displays (device dependent).

**Menu Paths**

It is part of the DISPLAY program.

---

**NODES**

Specifies "Nodes" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

**Menu Paths**

**Utility Menu>List>Status>Preprocessor>Nodes**

---



## /NOERASE

Prevents the screen erase between displays.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Command Default

Issue the command with no arguments to prevent automatic screen erase before creating the next display.

### Notes

Preventing the normal screen erase between requested displays allows you to overlay multiple views.

Clearing the screen with the **ERASE** command (**Utility Menu**> **PlotCtrls**> **Erase Options**> **Erase screen**) active simply clears the display area. Subsequent replots will provide the cumulative plots previously generated by the **/NOERASE** command.

For 3-D devices, you can issue **/DV3D,DELS** to suppress repeated screen overlays and generate clear contour plots.

Use the **/ERASE** command to reactivate automatic screen erase.

For 3-D devices (**/SHOW,3D**), the model in all active windows will be the same, even if you issue a different display command (**NPLOT**, **EPLLOT**, etc.) for each active window. Use the Multi-Plot command (**GPLOT**) to display different entities, in different windows, on 3-D devices.

This command is valid in any processor.

### Menu Paths

**Utility Menu**>**PlotCtrls**>**Erase Options**>**Erase between Plots**

---

## /NOLIST

Suppresses the data input listing.

[SESSION: List Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Command Default

Issue the command with no arguments to suppress the data input listing.

### Notes

Printout is suppressed until a **/GOLIST** command is read or the end of the listing is encountered.

This command is valid in any processor, but only within a batch run [**/BATCH**].

## Menu Paths

This command cannot be accessed from a menu.

---

### **NOOFFSET**, *Label*

Prevents the **CDREAD** command from offsetting specified data items

[PREP7: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Label*

Specifies items not to be offset.

#### **NODE**

Node numbers

#### **ELEM**

Element numbers

#### **KP**

Keypoint numbers

#### **LINE**

Line numbers

#### **AREA**

Area numbers

#### **VOLU**

Volume numbers

#### **MAT**

Material numbers

#### **TYPE**

Element type numbers

#### **REAL**

Real constant numbers

#### **CSYS**

Coordinate system numbers

#### **SECN**

Section numbers

#### **CP**

Coupled set numbers

#### **CE**

Constraint equation numbers

#### **CLEAR**

All items will be offset

#### **STATUS**

Shows which items are specified *notto* be offset.

## Notes

The **NOOFFSET** command specifies data items not to be offset by a set of data read from a **CDREAD** command.

## Menu Paths

This command cannot be accessed from a menu.

## NOORDER, *Lab*

Re-establishes the original element ordering.

[PREP7: Element Reordering](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *Lab*

Turns element reordering on or off.

#### **ON (or blank)**

Re-establishes original element ordering (default).

#### **OFF**

Original ordering is not used and program establishes its own ordering at the beginning of the solution phase.

## Notes

If *Lab* = ON, the original element ordering is re-established and no automatic reordering occurs at the beginning of the solution phase. Use *Lab* = OFF only to remove the effect of a previous **NOORDER** command. This command affects only those elements that were defined up to the point that this command is issued. See the **WSORT** and **WAVES** commands for reordering.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Numbering Ctrl>Element Reorder>Reset Elem Order

## /NOPR

Suppresses the expanded interpreted input data listing.

[SESSION: List Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Command Default

Issuing this command with no arguments suppresses the interpreted data input print out.

## Notes

Suppresses printout of interpreted input data, including information labeled as "Notes." When this printout is not suppressed, the data input to the analysis is echoed to the output file in an expanded format. Printout is suppressed until a **/GOPR** or **/GO** command is read.

Use of **/NOPR** is not recommended when the graphical user interface (GUI) is active. The GUI sometimes issues "hidden" **/NOPR** and **/GOPR** command sequences, which will countermand user-issued **/NOPR** commands, thus making the use of **/NOPR** in the GUI environment unpredictable.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Coupling / Ceqn>Rigid Region**

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Smooth Data**

**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Smooth Data**

**Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts**

**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Smooth Data**

**Main Menu>Solution>Loading Options>Smooth Data**

**Main Menu>Solution>Time Controls>Time Step Prediction**

**Main Menu>TimeHist Postpro>Smooth Data**

---

## NORA, AREA, NDIR

### Rotates nodal coordinate systems to surface normal

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### AREA

The area number containing the nodes to be rotated to their normals. If ALL, applies to all selected areas (see the **ASEL** command). If AREA = P, graphical picking is enabled.

#### NDIR

Direction of the normal. If NDIR = -1, the nodal coordinate system is rotated in the opposite direction of the surface normal. The default is the same direction as the surface normal.

## Notes

The NORA command rotates the X-axis of the nodal coordinate system to the surface normal. The rotated nodal coordinate systems may be displayed through the **/PSYMB** command. In case multiple areas are selected, there could be conflicts at the boundaries. If a node belongs to two areas that have a different normal, its nodal coordinate system will be rotated to the area normal with the lowest number. You can use the **AREVERSE** and **ANORM** commands to rotate the surface normals in the appropriate direction. Keep the following in mind when using the NORA command:

- If the nodal coordinate system is parallel to the global Cartesian system, it is not displayed through the **/PSYMB** command.
- Previously specified rotation on the selected nodes are overridden.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Areas**

---

**NORL**, *LINE*, *AREA*, *NDIR*

**Rotates nodal coordinate systems perpendicular to line normal**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **LINE**

Line number containing the nodes to be rotated. If ALL, applies to all selected lines (see the **LSEL** command). If *LINE* = P, graphical picking is enabled.

### **AREA**

The area number containing the selected lines. The normal of the line(s) selected is supposed to lie on this area. Defaults to the lowest numbered selected area containing the line number.

### **NDIR**

Direction of the normal. If *NDIR* = -1, the nodal coordinate system is rotated in the opposite direction of the line normal. The default is the same direction as the surface normal.

## Notes

The NORL command rotates the X-axis of the nodal coordinate perpendicular to the line normal. The rotated nodal coordinate systems may be displayed through the **/PSYMB** command. In case multiple lines are selected, there could be conflicts at the boundaries. If a node belongs to two lines that have a different normal, its nodal coordinate system will be rotated to the line normal with the lowest number. Keep the following in mind when using the NORL command:

- If the nodal coordinate system is parallel to the global Cartesian system, it is not displayed through the **/PSYMB** command.
- Previously specified rotation on the selected nodes are overridden.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Lines**

**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>with Area**

---

**/NORMAL**, *WN*, *KEY*

**Allows displaying area elements by top or bottom faces.**

[GRAPHICS:Style](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **WN**

Window number (or ALL) to which command applies (defaults to 1).

### **KEY**

Display key:

- 0**  
No face distinction.
- 1**  
Show only area elements having their positive normals directed toward the viewing point.
- 1**  
Show only area elements having their positive normals directed away from the viewing point.

## Command Default

Do not distinguish between top face and bottom face elements.

## Notes

**/NORMAL** allows you to select area elements and area plots by the top or bottom faces. It is useful for checking the normal directions on shell elements. The positive normal (element Z direction) is defined by the right-hand rule following the node I, J, K, L input direction. This command is available only with raster or hidden-line displays, for WIN32 or X11 2-D displays only.

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Style>Shell Normals**

---

## NPLOT, *KNUM*

**Displays nodes.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *KNUM*

Node number key:

- 0**  
No node numbers on display.
- 1**  
Include node numbers on display. See also **/PNUM** command.

## Notes

Produces a node display. Only selected nodes [**NSEL**] are displayed. Elements need not be defined. See the **DSYS** command for display coordinate system.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Areas**  
**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Lines**

**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>with Area  
Utility Menu>Plot>Nodes**

---

## **NPRINT, *N***

**Defines which time points stored are to be listed.**

[POST26: Listing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*N*

List data associated with every *N* time (or frequency) point(s), beginning with the first point stored (defaults to 1).

## **Command Default**

List all stored points.

## **Notes**

Defines which time (or frequency) points within the range stored are to be listed.

## **Menu Paths**

**Main Menu>TimeHist Postpro>Settings>List**

---

## **NREAD, *Fname, Ext, --***

**Reads nodes from a file.**

[PREP7: Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

*Ext*

Filename extension (8 character maximum).

The extension defaults to `NODE` if *Fname* is blank.

--

Unused field.

## **Notes**

The read operation is not necessary in a standard ANSYS run but is provided as a convenience to users wanting to read a coded node file, such as from another mesh generator or from a CAD/CAM program. Data

should be formatted as produced with the **NWRITE** command. Only nodes that are within the node range specified with the **NRRANG** command are read from the file. Duplicate nodes already in the database will be overwritten. The file is rewound before and after reading. Reading continues until the end of the file.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Nodes>Read Node File**

---

**NREFINE**, *NN1*, *NN2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

**Refines the mesh around specified nodes.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NN1*, *NN2*, *NINC***

Nodes (*NN1* to *NN2* in increments of *NINC*) around which the mesh is to be refined. *NN2* defaults to *NN1*, and *NINC* defaults to 1. If *NN1* = ALL, *NN2* and *NINC* are ignored and all selected nodes are used for refinement. If *NN1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NN1* (*NN2* and *NINC* are ignored).

### ***LEVEL***

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

### ***DEPTH***

Depth of mesh refinement in terms of number of elements outward from the indicated nodes (defaults to 1).

### ***POST***

Type of postprocessing to be done after element splitting, in order to improve element quality:

#### **OFF**

No postprocessing will be done.

#### **SMOOTH**

Smoothing will be done. Node locations may change.

#### **CLEAN**

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

### ***RETAIN***

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

#### **ON**

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

#### **OFF**

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.



## Notes

**NREFINE** performs local mesh refinement around the specified nodes. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

**NREFINE** refines all area elements and tetrahedral volume elements that are adjacent to the specified nodes. Any volume elements that are adjacent to the specified nodes, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. For additional restrictions on mesh refinement, see [Revising Your Model](#) in the *Modeling and Meshing Guide*.

This command is also valid for [rezoning](#).

## Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Nodes

## NRLSUM, *SIGNIF*, *Label*

Specifies the Naval Research Laboratory (NRL) sum mode combination method.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions

### **SIGNIF**

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multipoint, or DDAM response (**SPOPT**,SPRS, MPRS or DDAM), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**,PSD.)

### **Label**

Label identifying the combined mode solution output.

### **DISP**

Displacement solution (default). Displacements, stresses, forces, etc., are available.

### **VELO**

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

### **ACEL**

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

## Notes

This command is also valid in PREP7. This mode combination method is usually used for **SPOPT**,DDAM.

## Product Restrictions

NRLSUM is not allowed in ANSYS Professional.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine  
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine  
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine  
Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine  
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine  
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine

---

**\*NRM**, *Name*, *NormType*, *ParR*

Computes the norm of the specified matrix or vector.

[APDL: Matrix Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Name*

Matrix or vector for which the norm will be computed. This can be a dense matrix (created by the **\*DMAT** command), a sparse matrix (created by the **\*SMAT** command) or a vector (created by the **\*VEC** command)

### *NormType*

Mathematical norm to use:

#### **NRM2** --

L2 (Euclidian or SRSS) norm (default).

#### **NRM1** --

L1 (absolute sum) norm (vectors only).

#### **NRMINF** --

Maximum norm.

### *ParR*

Parameter name that contains the result.

## Notes

The NRM2 option corresponds to the Euclidian or L2 norm and is applicable to either vectors or matrices. The NRM1 option corresponds to the L1 norm and is applicable to vectors only. The NRMINF option is the maximum norm and is applicable to either vectors or matrices.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**NROPT**, *Option1*, *Option2*, *Optval*

**Specifies the Newton-Raphson options in a static or full transient analysis.**

[SOLUTION: Nonlinear Options](#)

MP ME ST <> PRN <> <> <> EM <> <> PP <> EME MFS

*Option1*

Option key:

**AUTO**

Let the program choose the option (default).

**FULL**

Use full Newton-Raphson.

**MODI**

Use modified Newton-Raphson.

**INIT**

Use the previously computed matrix (initial-stiffness).

**UNSYM**

Use full Newton-Raphson with unsymmetric matrices of elements where the unsymmetric option exists.

*Option2*

Option key:

**CRPL**

When applicable in a static creep analysis, activates modified Newton-Raphson with a creep-ratio limit. Valid only when *Option1* = AUTO.

*Optval*

Adaptive descent key:

**ON**

Use adaptive descent (default if frictional contact exists). Explicit ON is valid only if *Option* = FULL.

**OFF**

Do not use adaptive descent (default in all other cases).

**CRLIMIT**

The creep-ratio limit for use with the modified Newton-Raphson procedure. Typically, this value should not exceed 0.15. Valid only when *Option1* = AUTO (default) and *Option2* = CRPL.

## Notes

The **NROPT** command specifies the [Newton-Raphson option](#) used to solve the nonlinear equations in a static or full transient analysis.

If used within the solution processor, this command is valid only within the first load step.

The automatic modified Newton-Raphson procedure with creep-ratio limit control (**NROPT,AUTO,CRPL,CRLIMIT**) applies to static creep analysis only. When the creep ratio is smaller than value of the creep ratio limit specified, the modified Newton-Raphson procedure is used. If convergence difficulty occurs during solution, use the full Newton-Raphson procedure.

The default values given for this command assume **SOLCONTROL,ON** (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL,ON** and **SOLCONTROL,OFF**.

See [Newton-Raphson Option](#) in the *Structural Analysis Guide* for more information.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>Analysis Options**

---

## NROTAT, NODE1, NODE2, NINC

**Rotates nodal coordinate systems into the active system.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### NODE1, NODE2, NINC

Rotate nodes from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and all selected nodes [**NSEL**] are rotated. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

## Command Default

No automatic nodal rotation.

## Notes

Rotates nodal coordinate systems into the active coordinate system. Nodal coordinate systems may be automatically rotated into the active (global or local) coordinate system as follows: Rotations in Cartesian systems will have nodal x directions rotated parallel to the Cartesian X direction. Rotations in cylindrical, spherical or toroidal systems will have the nodal x directions rotated parallel to the R direction. Nodes at (or near) a zero radius location should not be rotated. Nodal coordinate directions may be displayed [**/PSYMB**]. Nodal forces and constraints will also appear rotated when displayed if the nodal coordinate system is rotated. For FLOTRAN analyses, nodal coordinate systems should only be rotated parallel to the global Cartesian system.

ANSYS LS-DYNA (explicit dynamics) does not support the **NROTAT** command. If you have rotated nodes in the implicit phase of an implicit-to-explicit sequential solution, you must rotate the nodes back to the global Cartesian direction before switching from implicit to explicit elements (**ETCHG,ITE**). Use the **EDNROT** command in the explicit run to maintain the same displacement constraints as were used on rotated nodes in the implicit run.

---

### Note

When the nodal coordinate systems are defined, they remain parallel to the global Cartesian system unless subsequently rotated.

Previously specified rotations on the specified nodes are overridden.

See the **NMODIF**, **NANG**, and **NORA** commands for other rotation options.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Nodes>RotateNode>To Active CS**

**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Active CS**

## NRRANG, NMIN, NMAX, NINC

**Specifies the range of nodes to be read from the node file.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### NMIN, NMAX, NINC

Node range is defined from *NMIN* (defaults to 1) to *NMAX* (defaults to 99999999) in steps of *NINC* (defaults to 1).

## Notes

Defines the range of nodes to be read [**NREAD**] from the node file. Also implies an element range since only elements fully attached to these nodes will be read from the element file.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Nodes>Read Node File**

## NSCALE, INC, NODE1, NODE2, NINC, RX, RY, RZ

**Generates a scaled set of nodes from a pattern of nodes.**

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### INC

Do this scaling operation one time, incrementing all nodes in the given pattern by *INC*. If *INC* = 0, nodes will be redefined at the scaled locations.

### NODE1, NODE2, NINC

Scale nodes from pattern of nodes beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and pattern is all selected nodes [**NSEL**]. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

### RX, RY, RZ

Scale factor ratios. Scaling is relative to the origin of the active coordinate system (RR, R $\theta$ , RZ for cylindrical, RR, R $\theta$ , R $\Phi$  for spherical or toroidal). If absolute value of ratio > 1.0, pattern is enlarged. If < 1.0, pattern is reduced. Ratios default to 1.0 (each).

## Notes

Generates a scaled pattern of nodes from a given node pattern. Scaling is done in the active coordinate system. Nodes in the pattern may have been generated in any coordinate system.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Copy>Nodes>Scale & Copy**

**Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>Scale & Move**

**Main Menu>Preprocessor>Modeling>Operate>Scale>Nodes>Scale & Copy**

**Main Menu>Preprocessor>Modeling>Operate>Scale>Nodes>Scale & Move**

**NSEL**, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

**Selects a subset of nodes.**

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of select:

#### **S**

Select a new set (default).

#### **R**

Reselect a set from the current set.

#### **A**

Additionally select a set and extend the current set.

#### **U**

Unselect a set from the current set.

#### **ALL**

Restore the full set.

#### **NONE**

Unselect the full set.

#### **INVE**

Invert the current set (selected becomes unselected and vice versa).

#### **STAT**

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

### *Item*

Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to NODE.

### *Comp*

Component of the item (if required). Valid component labels are shown in the table below.

**VMIN**

Minimum value of item range. Ranges are node numbers, set numbers, coordinate values, load values, or result values as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored).

**VMAX**

Maximum value of item range. *VMAX* defaults to *VMIN* for input values. For result values, *VMAX* defaults to infinity if *VMIN* is positive, or to zero if *VMIN* is negative.

**VINC**

Value increment within range. Used only with integer ranges (such as for node and set numbers). Defaults to 1. *VINC* cannot be negative.

**KABS**

Absolute value key:

**0**

Check sign of value during selection.

**1**

Use absolute value during selection (sign ignored).

## Command Default

All nodes are selected.

## Notes

Selects a subset of nodes. For example, to select a new set of nodes based on node numbers 1 through 7, use **NSEL,S,NODE,,1,7**. The subset is used when the ALL label is entered (or implied) on other commands, such as **NLIST,ALL**. Only data identified by node number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

When selecting nodes by results, the full graphics value is used, regardless of whether PowerGraphics is on.

Solution result data consists of two types, 1) nodal degree of freedom--results initially calculated at the nodes (such as displacement, temperature, pressure, etc.), and 2) element--results initially calculated elsewhere (such as at an element integration point or thickness location) and then recalculated at the nodes (such as stresses, strains, etc.). Various element results also depend upon the recalculation method and the selected results location [**AVPRIN**, **RSYS**, **FORCE**, **LAYER** and **SHELL**].

You must have all the nodes (corner and midside nodes) on the external face of the element selected to use *Item* = EXT.

This command is valid in any processor.

For Selects based on non-integer numbers (coordinates, results, etc.), items that are within the range  $VMIN - Toler$  and  $VMAX + Toler$  are selected. The default tolerance *Toler* is based on the relative values of *VMIN* and *VMAX* as follows:

- If  $VMIN = VMAX$ ,  $Toler = 0.005 \times VMIN$ .
- If  $VMIN = VMAX = 0.0$ ,  $Toler = 1.0E-6$ .
- If  $VMAX \neq VMIN$ ,  $Toler = 1.0E-8 \times (VMAX - VMIN)$ .

Use the **SELTOL** command to override this default and specify *Toler* explicitly.

**Table 228 NSEL - Valid Item and Component Labels**

**Valid Item and Component Labels NSEL, Type, Item, Comp, VMIN, VMAX, VINC, KABS**

**Valid item and component labels for input values are:**

Item	Comp	Description
NODE		Node number.
EXT		Nodes on exterior of selected elements (ignore remaining fields).
LOC	X, Y, Z	X, Y, or Z location in the active coordinate system.
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle.
M		Master node number.
CP		Coupled set number.
CE		Constraint equation set number.
D	U	Any of X, Y, or Z structural displacements. Amplitude only, if complex.
"	UX, UY, UZ	X, Y, or Z structural displacement. Amplitude only, if complex.
"	ROT	Any of X, Y, or Z structural rotations. Amplitude only, if complex.
"	ROTX, ROTY, ROTZ	X, Y, or Z structural rotation. Amplitude only, if complex.
"	TEMP, TBOT, TE2, TE3, . . ., TTOP	Temperature.
"	PRES	Pressure.
"	VOLT	Electric potential.
"	MAG	Magnetic scalar potential.
"	V	Any of X, Y, or Z fluid velocities.
"	VX, VY, VZ	X, Y, or Z fluid velocity.
"	A	Any of X, Y, or Z magnetic vector potentials. Amplitude only, if complex.
"	AX, AY, AZ	X, Y, or Z magnetic vector potential. Amplitude only, if complex.
"	CURR	Current.
"	EMF	Electromotive force drop.
"	ENKE, ENDS	Turbulent kinetic energy or energy dissipation (FLOTRAN).
F	F	Any of X, Y, or Z structural forces. Amplitude only, if complex.
"	FX, FY, FZ	X, Y, or Z structural force. Amplitude only, if complex.
"	M	Any of X, Y, or Z structural moments. Amplitude only, if complex
"	MX, MY, MZ	X, Y, or Z structural moment. Amplitude only, if complex.



**Valid Item and Component Labels NSEL, Type, Item, Comp, VMIN, VMAX, VINC, KABS****Valid item and component labels for input values are:**

Item	Comp	Description
"	HEAT, HBOT, HE2, HE3, . . . , HTOP	Heat flow.
"	FLOW	Fluid flow.
"	AMPS	Current flow.
"	FLUX	Magnetic flux.
"	CSG	Any of X, Y, or Z magnetic current segment components. Amplitude only, if complex.
"	CSGX, CSGY, CSGZY,	X, Y, or Z magnetic current segment component. Amp- litude only, if complex. if complex.
"	CHRG	Electric charge.
"	CHRGD	Electric charge density.
BF	TEMP	Nodal temperature.
"	FLUE	Nodal fluence.
"	HGEN	Nodal heat generation rate.
"	JS	Any of X, Y, or Z current densities. Amplitude only, if complex.
"	JSX, JSY, JSZ	X, Y, or Z current density. Amplitude only, if complex.
"	MVDI	Magnetic virtual displacements flag.

**Table 229 NSEL - Valid Item and Component Labels for Nodal DOF Result Values**

Item	Comp	Description
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy (FLOTRAN).

Item	Comp	Description
ENDS		Turbulent energy dissipation (FLOTRAN).

**Table 230 NSEL - Valid Item and Component Labels for Element Result Values**

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1,2,3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1,2,3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1,2,3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPth	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
CONT	STAT1	Contact status.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.

Item	Comp	Description
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component electromagnetic forces or vector sum.
TOPO		Densities used for topological optimization.

- For more information on the meaning of contact status and its possible values, see [Reviewing Results in POST1](#) in the *Contact Technology Guide*.

**Table 231 NSEL - Valid Item and Component Labels for FLOTRAN Nodal Result Values**

Item	Comp	Description
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements**

**Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts**

**Utility Menu>Select>Entities**

**NSLA**, *Type*, *NKEY*

Selects those nodes associated with the selected areas.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of node select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

**NKEY**

Specifies whether only interior area nodes are to be selected:

- 0**  
Select only nodes interior to selected areas.
- 1**  
Select all nodes (interior to area, interior to lines, and at keypoints) associated with the selected areas.

**Notes**

Valid only if the nodes were generated by an area meshing operation [[AMESH](#), [VMESH](#)] on a solid model that contains the selected areas.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>Select>Entities**

**NSLE**, *Type*, *NodeType*, *Num*

Selects those nodes attached to the selected elements.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of node select:

- S**  
Select a new set (default).

**R**

Reselect a set from the current set.

**A**

Additionally select a set and extend the current set.

**U**

Unselect a set from the current set.

**NodeType**

Label identifying type of nodes to consider when selecting:

**ALL**

Select all nodes of the selected elements (default).

**ACTIVE**

Select only the active nodes. An active node is a node that contributes DOFs to the model.

**INACTIVE**

Select only inactive nodes (such as orientation or radiation).

**CORNER**

Select only corner nodes.

**MID**

Select only midside nodes.

**POS**

Select nodes in position *Num*.

**FACE**

Select nodes on face *Num*.

**Num**

Position or face number for *NodeType* = POS or FACE.

**Notes**

**NSLE** selects *NodeType* nodes attached to the currently-selected set of elements. Only nodes on elements in the currently-selected element set can be selected.

**Note**

When using degenerate hexahedral elements, **NSLE**, *U*, *CORNER* and **NSLE**, *S*, *MID* will not select the same set of nodes because some nodes appear as both corner and midside nodes.

This command is valid in any processor.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking**  
**Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts**  
**Utility Menu>Select>Entities**

**NSLK**, *Type*

Selects those nodes associated with the selected keypoints.

DATABASE: [Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of node select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

**Notes**

Valid only if the nodes were generated by a keypoint meshing operation [[KMESH](#), [LMESH](#), [AMESH](#), [VMESH](#)] on a solid model that contains the selected keypoints.

This command is valid in any processor.

**Menu Paths**

Utility Menu>Select>Entities

**NSLL**, *Type*, *NKEY*

Selects those nodes associated with the selected lines.

DATABASE: [Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of node select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

**NKEY**

Specifies whether only interior line nodes are to be selected:

- 0**  
Select only nodes interior to selected lines.
- 1**  
Select all nodes (interior to line and at keypoints) associated with the selected lines.

## Notes

Valid only if the nodes were generated by a line meshing operation [**LMESH**, **AMESH**, **VMESH**] on a solid model that contains the associated lines.

This command is valid in any processor.

## Menu Paths

Utility Menu>Select>Entities

## NSLV, *Type*, *NKEY*

Selects those nodes associated with the selected volumes.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying the type of node select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

### *NKEY*

Specifies whether only interior volume nodes are to be selected:

- 0**  
Select only nodes interior to selected volumes.
- 1**  
Select all nodes (interior to volume, interior to areas, interior to lines, and at keypoints) associated with the selected volumes.

## Notes

Valid only if the nodes were generated by a volume meshing operation [**VMESH**] on a solid model that contains the selected volumes.

This command is valid in any processor.

## Menu Paths

Utility Menu>Select>Entities

### NSMOOTH, *NPASS*

Smooths selected nodes among selected elements.

[PREP7:Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *NPASS*

Number of smoothing passes. Defaults to 3.

## Notes

Repositions each selected node at the average position of its immediate neighbors on the selected elements. The node positions converge after some number of smoothing passes. For some initial conditions, *NPASS* may need to be much larger than 3. If the boundary of a mesh is to be undisturbed (usually desirable), the boundary nodes should be unselected before issuing **NSMOOTH**.

## Menu Paths

This command cannot be accessed from a menu.

### NSOL, *NVAR*, *NODE*, *Item*, *Comp*, *Name*, *SECTOR*

Specifies nodal data to be stored from the results file.

[POST26:Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *NVAR*

Arbitrary reference number or name assigned to this variable. Variable numbers can be 2 to *NV* (**NUMVAR**) while the name can be an eight byte character string. Overwrites any existing results for this variable.

#### *NODE*

Node for which data are to be stored.

#### *Item*

Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label.

#### *Comp*

Component of the item (if required). Valid component labels are shown in the table below.

#### *Name*

Thirty-two character name identifying the item on printouts and displays. Defaults to a label formed by concatenating the first four characters of the *Item* and *Comp* labels.

#### *SECTOR*

For a cyclic symmetry solution, the sector number for which the results from *NODE* are to be stored.



## Notes

Stores nodal degree of freedom and solution results in a variable. For more information, see [Data Interpreted in the Nodal Coordinate System](#) in the *Modeling and Meshing Guide*.

**Table 232 NSOL - Valid Item and Component Labels**

**Valid Item and Component Labels NSOL, NVAR, NODE, Item, Comp, Name**

**Valid item and component labels for nodal degree of freedom results are:**

Item	Comp	Description
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity in a fluid analysis.
A	X, Y, Z	X, Y, or Z magnetic vector potential in an electromagnetic analysis.
VEL	X, Y, Z	X, Y, or Z velocity in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
ACC	X, Y, Z	X, Y, or Z acceleration in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
OMG	X, Y, Z	X, Y, or Z rotational velocity in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
DMG	X, Y, Z	X, Y, or Z rotational acceleration in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.

**Table 233 NSOL - Valid Item and Component Labels for FLOTRAN Nodal Results**

Item	Comp	Description
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)

Item	Comp	Description
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
RDFL		Radiation heat flux.

**Table 234 NSOL - Valid Item and Component Labels for ANSYS LS-DYNA Nodal Results**

Item	Comp	Description
U	X, Y, Z	X, Y, or Z nodal displacements.
ROT	X, Y, Z	X, Y, or Z nodal rotations.
V	X, Y, Z	X, Y, or Z nodal velocities.
A	X, Y, Z	X, Y, or Z nodal accelerations.

- For [SHELL131](#) and [SHELL132](#) elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . . , TTOP instead of TEMP.

## Menu Paths

**Main Menu>Drop Test>Time History>Graph Variables**

**Main Menu>Drop Test>Time History>List Variables**

**Main Menu>TimeHist Postpro>Define Variables**

**Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables**

**NSORT**, *Item*, *Comp*, *ORDER*, *KABS*, *NUMB*, *SEL*

**Sorts nodal data.**

[POST1: Results](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Item*

Label identifying the item to be sorted on. Valid item labels are shown in the table below. Some items also require a component label.

### *Comp*

Component of the item (if required). Valid component labels are shown in the table below.

### *ORDER*

Order of sort operation:

**0**

Sort into descending order.

**1**

Sort into ascending order.

**KABS**

Absolute value key:

**0**

Sort according to real value.

**1**

Sort according to absolute value.

**NUMB**

Number of nodal data records to be sorted in ascending or descending order (*ORDER*) before sort is stopped (remainder will be in unsorted sequence) (defaults to all nodes).

**SEL**

Allows selection of nodes in the sorted field.

**(blank)**

No selection (default).

**SELECT**

Select the nodes in the sorted list.

## Command Default

Use ascending node-number order.

## Notes

Values are in the active coordinate system [**CSYS** for input data or **RSYS** for results data]. Various element results also depend upon the recalculation method and the selected results location [**AVPRIN**, **RSYS**, **SHELL**, **ESEL**, and **NSEL**]. If simultaneous load cases are stored, the last sorted sequence formed from any load case applies to all load cases. Use **NUSORT** to restore the original order. This command is not valid with Power-Graphics.

**Table 235 NSORT - Valid Item and Component Labels**

**Valid Item and Component Labels NSORT, *Item*, *Comp*, *ORDER*, *KABS*, *NUMB*, *SEL***

**Valid item and component labels for input values are:**

Item	Comp	Description
LOC	X, Y, Z	X, Y, or Z location.
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle.

**Table 236 NSORT - Valid Item and Component Labels for Nodal DOF Result Values**

Item	Comp	Description
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP		Temperature (includes TEMP, TBOT, TE2, TE3, . . . , TTOP values).
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.

Item	Comp	Description
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy (FLOTRAN).
ENDS		Turbulent energy dissipation (FLOTRAN).

**Table 237 NSORT - Valid Item and Component Labels for Element Result Values**

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2,3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
FAIL	MAX	Maximum of all active failure criteria defined at the current location. (See the <b>FCTYP</b> command for details.) [1]
"	EMAX	Maximum Strain Failure Criterion [1]
"	SMAX	Maximum Stress Failure Criterion [1]

Item	Comp	Description
"	TWSI	Tsai-Wu Strength Index Failure Criterion [1]
"	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion [1]
"	HFIB	Hashin Fiber Failure Criterion. [1][2]
"	HMAT	Hashin Matrix Failure Criterion. [1][2]
"	PFIB	Puck Fiber Failure Criterion. [1][2]
"	PMAT	Puck Matrix Failure Criterion. [1][2]
"	USR1, USR2, ..., USR9	User-defined failure criteria [1][2][3]
CONT	STAT [4]	Contact status.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component electromagnetic forces or vector sum.
TOPO		Densities used for topological optimization.

1. Works only if failure criteria information is provided. (For more information, see the documentation for the **FC** and **TB** commands.)
2. Must be added via the **FCTYP** command first.
3. Works only if user failure criteria routine is provided.
4. For more information about contact status and its possible values, see [Reviewing Results in POST1](#) in the [Contact Technology Guide](#).

**Table 238 NSORT - Valid Item and Component Labels for FLOTRAN Nodal Result Values**

Item	Comp	Description
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.

Item	Comp	Description
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.

## Menu Paths

**Main Menu>General Postproc>List Results>Sorted Listing>Sort Nodes**  
**Utility Menu>Parameters>Get Scalar Data**

---

## NSTORE, *TINC*

Defines which time points are to be stored.

[POST26: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *TINC*

Store data associated with every *TINC* time (or frequency) point(s), within the previously defined range of *TMIN* to *TMAX* [**TIMERANGE**]. (Defaults to 1)

## Command Default

Store every point.

## Notes

Defines which time (or frequency) points within the range are to be stored.

## Menu Paths

**Main Menu>TimeHist Postpro>Settings>Data**

---

**NSUBST**, *NSBSTP*, *NSBMX*, *NSBMN*, *Carry*

**Specifies the number of substeps to be taken this load step.**

**SOLUTION:** Load Step Options

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

***NSBSTP***

Number of substeps to be used for this load step (i.e., the time step size or frequency increment). If automatic time stepping is used [**AUTOTS**], *NSBSTP* defines the size of the first substep. If **SOLCONTROL**,ON and contact elements **TARGE169**, **TARGE170**, **CONTA171**, **CONTA172**, **CONTA173**, **CONTA174**, **CONTA175**, **CONTA176**, or **CONTA177** are used, defaults to 1 or 20 substeps, depending on the physics of the problem. If **SOLCONTROL**,ON and none of these contact elements are used, defaults to 1 substep. If **SOLCONTROL**,OFF, defaults to the previously specified value (or 1, if there is no previously specified value).

***NSBMX***

Maximum number of substeps to be taken (i.e., the minimum time step size) if automatic time stepping is used. If **SOLCONTROL**,ON, ANSYS determines the default depending on the physics of the problem. If **SOLCONTROL**,OFF, defaults to the previously specified value (or *NSBSTP*, if there is no previously specified value).

***NSBMN***

Minimum number of substeps to be taken (i.e., the maximum time step size) if automatic time stepping is used. If **SOLCONTROL**,ON, ANSYS determines the default depending on the physics of the problem. If **SOLCONTROL**,OFF, defaults to the previously specified value (or 1, if there is no previously specified value).

***Carry***

Time step carryover key (ANSYS determines the default depending on the physics of the problem):

**OFF**

Use *NSBSTP* to define time step at start of each load step.

**ON**

Use final time step from previous load step as the starting time step (if automatic time stepping is used).

If **SOLCONTROL**,ON, ANSYS determines the default depending on the physics of the problem. If **SOLCONTROL**,OFF, defaults to OFF.

**Notes**

See **DELTIM** for an alternative input. It is recommended that all fields of this command be specified for solution efficiency and robustness.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substps**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Basic**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substps**

**NSVR**, *ITYPE*, *NSTV*

Defines the number of variables for user-programmable element options.

[PREP7: Element Type](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**ITYPE**

Element type number as defined on the **ET** command.

**NSTV**

Number of extra state variables to save (must be no more than 840).

**Command Default**

No extra variables are saved.

**Notes**

Defines the number of extra variables that need to be saved for user-programmable (system-dependent) element options, e.g., material laws through user subroutine USERPL. *ITYPE* must first be defined with the **ET** command.

**Menu Paths**

**This command cannot be accessed from a menu.**

**NSYM**, *Ncomp*, *INC*, *NODE1*, *NODE2*, *NINC*

Generates a reflected set of nodes.

[PREP7: Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Ncomp**

Symmetry key:

**X**

X (or R) symmetry (default).

**Y**

Y (or  $\theta$ ) symmetry.

**Z**

Z (or  $\Phi$ ) symmetry.

**INC**

Increment all nodes in the given pattern by *INC* to form the reflected node pattern.

**NODE1**, **NODE2**, **NINC**

Reflect nodes from pattern beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and pattern is all selected nodes [**NSSEL**]. If



*NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

## Notes

Generates nodes from a given node pattern by a symmetry reflection. Reflection is done in the active coordinate system by changing a particular coordinate sign. Nodes in the pattern may have been generated in any coordinate system. Nodal rotation angles are not reflected.

Symmetry reflection may be used with any node pattern, in any coordinate system, as many times as desired. Reflection is accomplished by a coordinate sign change (in the active coordinate system). For example, an X-reflection in a Cartesian coordinate system generates additional nodes from a given pattern, with a node increment added to each node number, and an X coordinate sign change. An R-reflection in a cylindrical coordinate system gives a reflected "radial" location by changing the "equivalent" Cartesian (i.e., the Cartesian system with the same origin as the active cylindrical system) X and Y coordinate signs. An R-reflection in a spherical coordinate system gives a reflected "radial" location by changing the equivalent Cartesian X, Y, and Z coordinate location signs. Nodal coordinate system rotation angles are not reflected.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Reflect>Nodes**

## /NUMBER, NKEY

**Specifies whether numbers, colors, or both are used for displays.**

[GRAPHICS:Labeling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NKEY*

Numbering style:

**0**

Color (terminal dependent) the numbered items and show numbers.

**1**

Color the numbered items. Do not show the numbers.

**2**

Show the numbers. Do not color the items.

**-1**

Do not color the items or show the numbers. For contour plots, the resulting display will vary (see below).

## Notes

Specifies whether numbers, colors, or both are used for numbering displays [/PNUM] of nodes, elements, keypoints, lines, areas, and volumes.

Shading is also available for terminals configured with more than 4 color planes [/SHOW]. Color automatically appears for certain items and may be manually controlled (off or on) for other items.

When you suppress color (*NKEY* = -1) your contour plots will produce different results, depending on your graphics equipment. For non-3-D devices (X11, Win32, etc.) your contour plot will be white (no color). For 3-D devices, such as OpenGL, the resulting display will be in color.

The following items are automatically given discrete colors: Boundary condition symbols [**/PBC**], curves on graph displays, and distorted geometry on postprocessing displays. Contour lines in postprocessing displays are automatically colored based upon a continuous, rather than a discrete, spectrum so that red is associated with the highest contour value. On terminals with raster capability [**/SHOW**], the area between contour lines is filled with the color of the higher contour.

Explicit entity colors or the discrete color mapping may be changed with the **/COLOR** command.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Numbering

---

### NUMCMP, *Label*

Compresses the numbering of defined items.

[PREP7: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Label*

Items to be compressed:

#### **NODE**

Node numbers

#### **ELEM**

Element numbers

#### **KP**

Keypoint numbers

#### **LINE**

Line numbers

#### **AREA**

Area numbers

#### **VOLU**

Volume numbers

#### **MAT**

Material numbers

#### **TYPE**

Element type numbers

#### **REAL**

Real constant numbers

#### **CP**

Coupled set numbers

**SECN**

Section numbers

**CE**

Constraint equation numbers

**ALL**

All item numbers

## Notes

The **NUMCMP** command effectively compresses out unused item numbers by renumbering all the items, beginning with one and continuing throughout the model. The renumbering order follows the initial item numbering order (that is, compression lowers the maximum number by "sliding" numbers down to take advantage of unused or skipped numbers). All defined items are renumbered, regardless of whether or not they are actually used or selected. Applicable related items are also checked for renumbering as described for the merge operation (**NUMMRG**).

Compressing material numbers (**NUMCMP,ALL** or **NUMCMP,MAT**) does *not* update the material number referenced by either of the following:

- A temperature-dependent convection or surface-to-surface radiation load (**SF, SFE, SFL, SFA**)
- Real constants for multi-material elements (such as **SOLID65**)

Compression is usually not required unless memory space is limited and there are large gaps in the numbering sequence.

## Menu Paths

**Main Menu>Preprocessor>Numbering Ctrl>Compress Numbers**

## NUMEXP, NUM, BEGRNG, ENDRNG, Elcalc

**Specifies solutions to be expanded from reduced analyses.**

**SOLUTION: Load Step Options**

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

**NUM**

The number of solutions to expand. This value is required.

**Num**

Number of solutions to expand.

**ALL**

Expand all substeps between *BEGRNG* and *ENDRNG* (provided that *ENDRNG* > 0). If *BEGRNG* and *ENDRNG* have no specified values, this option expands *all* substeps of *all* load steps.

**BEGRNG, ENDRNG**

Beginning and ending time (or frequency) range for expanded solutions. The default is 0 for both values.

**Elcalc**

The element-calculation key:

**YES**

Calculate element results, nodal loads, and reaction loads. This value is the default.

**NO**

Do not calculate these items.

## Command Defaults

Issuing this command with no arguments is invalid. You must specify the number of solutions, or all solutions, to expand (*NUM*). The default value for both the beginning (*BEGRNG*) and ending (*ENDRNG*) time or frequency is 0. The default behavior of the command is to calculate element results, nodal loads, and reaction loads (*Elcalc* = YES).

## Notes

Specifies a range of solutions to be expanded from analyses that use reduced or mode superposition methods (**ANTYPE**, HARMIC, TRANS, or SUBSTR).

For **ANTYPE**, TRANS, *NUM*, evenly spaced solutions are expanded between time *BEGRNG* and time *ENDRNG*.

For **ANTYPE**, HARMIC, *NUM*, evenly spaced solutions are expanded between frequency *BEGRNG* and frequency *ENDRNG*.

The first expansion in all cases is done at the first point beyond *BEGRNG* (that is, at  $BEGRNG + (ENDRNG - BEGRNG) / NUM$ ).

For a single expansion of a solution, or for multiple expansions when the solutions are not evenly spaced (such as in a mode superposition harmonic analysis with the cluster option), ANSYS recommends issuing one or more **EXPSOL** commands.

The **NUMEXP** command is invalid in these cases:

- In a substructuring analysis (**ANTYPE**, *SUBST*) when a factorized matrix file (the .LN22 file generated by the sparse solver) does not exist, causing ANSYS to employ the full-resolve method.
- If the full-resolve option is selected using the **SEOPT** command.

In both situations, use the **EXPSOL** command to perform a single expansion for each solution desired.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's**  
**Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's**

---

**NUMMRG**, *Label*, *TOLER*, *GTOLER*, *Action*, *Switch*

**Merges coincident or equivalently defined items.**

PREP7: Database

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Label***

Items to be merged:

**NODE**

Nodes

**ELEM**

Elements

**KP**

Keypoints (will also merge lines, areas, and volumes)

**MAT**

Materials

**TYPE**

Element types

**REAL**

Real constants

**CP**

Coupled sets

**CE**

Constraint equations

**ALL**

All items

***TOLER***

Range of coincidence. For *Label* = NODE and KP, defaults to 1.0E-4 (based on maximum Cartesian coordinate difference between nodes or keypoints). For *Label* = MAT, REAL, and CE, defaults to 1.0E-7 (based on difference of the values normalized by the values). Only items within range are merged. (For keypoints attached to lines, further restrictions apply. See the *GTOLER* field and *Merging Solid Model Entities* below.)

***GTOLER***

Global solid model tolerance -- used only when merging keypoints attached to lines. If specified, *GTOLER* will override the internal relative solid model tolerance. See *Merging Solid Model Entities* below.

***Action***

Specifies whether to merge or select coincident items.

**SELE**

Select coincident items but do not merge. *Action* = SELE is only valid for *Label* = NODE.

**(Blank)**

Merge the coincident items (default).

***Switch***

Specifies whether the lowest or highest numbered coincident item is retained after the merging operation. This option does not apply to keypoints; i.e., for *Label* = KP, the lowest numbered keypoint is retained regardless of the *Switch* setting.

**LOW**

Retain the lowest numbered coincident item after the merging operation (default).

**HIGH**

Retain the highest numbered coincident item after the merging operation.

**Notes**

After issuing the command, the area and volume sizes (**ASUM** and **VSUM**) may give slightly different results. In order to obtain the same results as before, use **/FACET**, **/NORMAL**, and **ASUM / VSUM**.

The merge operation is useful for tying separate, but coincident, parts of a model together. If not all items are to be checked for merging, use the select commands (**NSEL**, **ESEL**, etc.) to select items. Only selected items are included in the merge operation for nodes, keypoints, and elements.

By default, the merge operation retains the lowest numbered coincident item. Higher numbered coincident items are deleted. Set *Switch* to HIGH to retain the highest numbered coincident item after the merging operation. Applicable related items are also checked for deleted item numbers and if found, are replaced with the retained item number. For example, if nodes are merged, element connectivities (except superelements), mesh item range associativity, coupled degrees of freedom, constraint equations, master degrees of freedom, gap conditions, degree of freedom constraints, nodal force loads, nodal surface loads, and nodal body force loads are checked. Merging material numbers [**NUMMRG,ALL** or **NUMMRG,MAT**] does *not* update the material number referenced:

- By temperature-dependent film coefficients as part of convection load or a temperature-dependent emissivity as part of a surface-to-surface radiation load [**SF**, **SFE**, **SFL**, **SFA**]
- By real constants for multi-material elements (such as **SOLID65**)

If a unique load is defined among merged nodes, the value is kept and applied to the retained node. If loads are not unique (not recommended), only the value on the lowest node (or highest if *Switch* = HIGH) will be kept, except for "force" loads for which the values will be summed if they are not defined using tabular boundary conditions.

---

**Note**

The unused nodes (not recommended) in elements, couplings, constraint equations, etc. may become active after the merge operation.

The *Action* field provides the option of visualizing the coincident items before the merging operation.

---

## Caution

When merging entities in a model that has already been meshed, the order in which you issue multiple **NUMMRG** commands is significant. If you want to merge two adjacent meshed regions that have coincident nodes and keypoints, always merge nodes [**NUMMRG,NODE**] before merging keypoints [**NUMMRG,KP**]. Merging keypoints before nodes can result in some of the nodes becoming "orphaned"; that is, the nodes lose their association with the solid model. Orphaned nodes can cause certain operations (such as boundary condition transfers, surface load transfers, and so on) to fail. However, using **NUMMRG** should be avoided if at all possible, as the procedure outlined above may even cause meshing failure, especially after multiple merging and meshing operations.

After a **NUMMRG,NODE**, is issued, some nodes may be attached to more than one solid entity. As a result, subsequent attempts to transfer solid model loads to the elements may not be successful. Issue **NUMMRG,KP** to correct this problem. Do *NOT* issue **VCLEAR** before issuing **NUMMRG,KP**.

For **NUMMRG,ELEM**, elements must be identical in all aspects, including the direction of the element coordinate system.

For certain solid and shell elements (181, 185, 190, etc) ANSYS will interpret coincident faces as internal and eliminate them. To prevent this from occurring, shrink the entities by a very small factor to delineate coincident items (**/SHRINK**, 0.0001) and no internal nodes, lines, areas or elements will be eliminated.

When working with solid models, you may have better success with the gluing operations (**AGLUE**, **LGLUE**, **VGLUE**). Please read the following information when attempting to merge solid model entities.

### *Gluing Operations vs. Merging Operations*

Adjacent, touching regions can be joined by gluing them (**AGLUE**, **LGLUE**, **VGLUE**) or by merging coincident keypoints (**NUMMRG,KP**, which also causes merging of identical lines, areas, and volumes). In many situations, either approach will work just fine. Some factors, however, may lead to a preference for one method over the other.

## Geometric Configuration

Gluing is possible regardless of the initial alignment or offset of the input entities. Keypoint merging is possible only if each keypoint on one side of the face to be joined is matched by a coincident keypoint on the other side. This is commonly the case after a symmetry reflection (**ARSYM** or **VSYMM**) or a copy (**AGEN** or **VGEN**), especially for a model built entirely in ANSYS rather than imported from a CAD system. When the geometry is extremely precise, and the configuration is correct for keypoint merging, **NUMMRG** is more efficient and robust than **AGLUE** or **VGLUE**.

## Model Accuracy

Gluing, like all ANSYS boolean operations, requires that the input entities meet the current boolean tolerance (BTOL). Otherwise, **AGLUE** or **VGLUE** may fail. In such cases, relaxing the tolerance may allow the glue to complete. An advantage of gluing is that it is unlikely to degrade the accuracy of a geometric model. Keypoint merging can operate on almost any combination of entities (although you may have to override the default tolerances on **NUMMRG**). However, it can also introduce or increase accuracy flaws, making later boolean operations less likely to succeed. If the input tolerances are too large, **NUMMRG** can collapse out small lines, areas, or volumes you intended to keep, possibly rendering the model unusable.

## Mesh Status

Gluing, like all ANSYS boolean operations, requires that the input entities be unmeshed. Keypoint merging is effective for meshed models under the right conditions. More information on keypoint merging follows.

### *Merging Solid Model Entities:*

When merging solid model entities (*Label* = KP or ALL), keypoint locations are used as the basis for merging. Once keypoints are merged, any higher order solid model entities (lines, areas, and volumes), regardless of their select status or attachment to the merged keypoints, are considered for merging.

Keypoints that are attached to lines will be merged only if:

- $\Delta X$ ,  $\Delta Y$ , and  $\Delta Z$  are each less than *TOLER*

where,

$\Delta X$  is the X component of the distance between keypoints,  
 $\Delta Y$  is the Y component of the distance between keypoints, and  
 $\Delta Z$  is the Z component of the distance between keypoints;

and

- $\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$  is less than 1E-5 times the length of the longest line attached to those keypoints (internal relative solid model tolerance), or  $\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$  is less than *GTOLER* (global solid model tolerance) if specified.

The *TOLER* field is a *consideration* tolerance. If a keypoint is within *TOLER* of another keypoint, then those two keypoints are *candidates* to be merged. If, when "moving" the higher numbered keypoint, the distance exceeds the internal relative solid model tolerance, or the global solid model tolerance (*GTOLER*) if specified, the keypoints will *not* be merged. Lines, areas, and volumes are considered for merging in a similar manner.

The internal relative solid model tolerance should be overridden by the global solid model tolerance (*GTOLER*) only when absolutely necessary. *GTOLER* is an *absolute* tolerance; if specified, relative lengths of lines in the model will no longer be considered in the merge operation. If *GTOLER* is too large, you can "merge-out" portions of your model accidentally, effectively defeaturing the model. If *GTOLER* is used, it is recommended that **NUMMRG** be preceded by saving the database (since undesired merges of solid model entities could occur).

---

### Note

Use of the **NUMMRG** command does not cause changes to a model's geometry; only the topology is affected.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Circuit>Merge Nodes**

**Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts**

**Main Menu>Preprocessor>NumberingCtrls>Merge Items**

---



**NUMOFF**, *Label*, *VALUE*

Adds a number offset to defined items.

[PREP7: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Label***

Apply offset number to one of the following sets of items:

**NODE**

Nodes

**ELEM**

Elements

**KP**

Keypoints

**LINE**

Lines

**AREA**

Areas

**VOLU**

Volumes

**MAT**

Materials

**TYPE**

Element types

**REAL**

Real constants

**CP**

Coupled sets

**SECN**

Section numbers

**CE**

Constraint equations

**CSYS**

Coordinate systems

***VALUE***

Offset number value (cannot be negative).

**Notes**

Useful for offsetting current model data to prevent overlap if another model is read in. **CDWRITE** automatically writes the appropriate **NUMOFF** commands followed by the model data to `File.CDB`. Therefore, when the file is read, any model already existing in the database is offset before the model data on the file is read.

Offsetting material numbers with this command [**NUMOFF**,MAT] does *not* update the material number referenced by either of the following:

- A temperature-dependent convection or surface-to-surface radiation load [**SF**, **SFE**, **SFL**, **SFA**]
- Real constants for multi-material elements (such as **SOLID65**).

Therefore, a mismatch may exist between the material definitions and the material numbers referenced.

## Menu Paths

**Main Menu>Preprocessor>Numbering Ctrl>Add Num Offset**

**NUMSTR**, *Label*, *VALUE*

**Establishes starting numbers for automatically numbered items.**

[PREP7: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Label*

Apply starting number to one of the following sets of items:

#### **NODE**

Node numbers. *Value* defaults (and is continually reset) to 1 + maximum node number in model. Cannot be reset lower.

#### **ELEM**

Element numbers. *Value* defaults (and is continually reset) to 1 + maximum element number in model. Cannot be reset lower.

#### **KP**

Keypoint numbers. *Value* defaults to 1. Only undefined numbers are used. Existing keypoints are not overwritten.

#### **LINE**

Line numbers. *Value* defaults to 1. Only undefined numbers are used. Existing lines are not overwritten.

#### **AREA**

Area numbers. *Value* defaults to 1. Only undefined numbers are used. Existing areas are not overwritten.

#### **VOLU**

Volume numbers. *Value* defaults to 1. Only undefined numbers are used. Existing volumes are not overwritten.

#### **DEFA**

Default. Returns all starting numbers to their default values.

### **VALUE**

Starting number value.

## Notes

Establishes starting numbers for various items that may have numbers automatically assigned (such as element numbers with the **EGEN** command, and node and solid model entity numbers with the mesh [**AMESH**, **VMESH**, etc.] commands). Use **NUMSTR,STAT** to display settings. Use **NUMSTR,DEFA** to reset all specifications back to defaults. Defaults may be lowered by deleting and compressing items (i.e., **NDELE** and **NUMCMP,NODE** for nodes, etc.).

---

### Note

A mesh clear operation (**VCLEAR**, **ACLEAR**, **LCLEAR**, and **KCLEAR**) automatically sets starting node and element numbers to the highest unused numbers. If a specific starting node or element number is desired, issue **NUMSTR** after the clear operation.

## Menu Paths

**Main Menu>Preprocessor>Numbering Ctrl>Reset Start Num**  
**Main Menu>Preprocessor>Numbering Ctrl>Set Start Number**  
**Main Menu>Preprocessor>Numbering Ctrl>Start Num Status**

---

## NUMVAR, *NV*

Specifies the number of variables allowed in **POST26**.

[POST26: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NV*

Allow storage for *NV* variables. 200 maximum are allowed. Defaults to 10 (except for an explicit dynamics analysis, which defaults to 30). **TIME** (variable 1) should also be included in this number.

## Command Default

Allow storage for 10 variables (30 variables for an explicit dynamics analysis).

## Notes

Specifies the number of variables allowed for data read from the results file and for data resulting from an operation (if any). For efficiency, *NV* should not be larger than necessary. *NV* cannot be changed after data storage begins.

## Menu Paths

**Main Menu>TimeHist Postpro>Settings>File**

---

## NUSORT

Restores original order for nodal data.

[POST1: Results](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This command restores the nodal data to its original order (sorted in ascending node number sequence) after an **NSORT** command. Changing the selected nodal set [**NSEL**] also restores the original nodal order.

### Menu Paths

**Main Menu>General Postproc>List Results>Sorted Listing>Unsort Nodes**

**NWPAVE**, *N1, N2, N3, N4, N5, N6, N7, N8, N9*

Moves the working plane origin to the average location of nodes.

[DATABASE: Working Plane](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*N1, N2, N3, . . . , N9*

Nodes used in calculation of the average. At least one must be defined. If *N1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### Notes

Averaging is based on the active coordinate system.

This command is valid in any processor.

### Menu Paths

**Utility Menu>WorkPlane>Offset WP to>Nodes**

**NWPLAN**, *WN, NORIG, NXAX, NPLAN*

Defines the working plane using three nodes.

[DATABASE: Working Plane](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*WN*

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified. If fewer than three points are used, the viewing direction of window *WN* will be used instead to define the normal to the working plane.

**NORIG**

Node number defining the origin of the working plane coordinate system. If *NORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**NXAX**

Node number defining the x-axis orientation (defaults to the x-axis being parallel to the global X-axis; or if the normal to the working plane is parallel to the global X-axis, then defaults to being parallel to the global Y-axis).

**NPLAN**

Node number defining the working plane (the normal defaults to the present display view [**/VIEW**] of window *WN*).

## Notes

Defines a working plane to assist in picking operations using three nodes as an alternate to the **WPLANE** command. The three nodes also define the working plane coordinate system. A minimum of one node (at the working plane origin) is required. Immediate mode may also be active. See the **WPSTYL** command to set the style of the working plane display.

This command is valid in any processor.

## Menu Paths

**Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node  
Utility Menu>WorkPlane>Align WP with>Nodes**

**NWRITE**, *Fname*, *Ext*, --, *KAPPND*

**Writes nodes to a file.**

[PREP7: Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to *Jobname*.

***Ext***

Filename extension (8 character maximum).

The extension defaults to *NODE* if *Fname* is blank.

--

Unused field.

***KAPPND***

Append key:

**0**

Rewind file before the write operation.

1

Append data to the end of the existing file.

## Notes

Writes selected nodes [**NSEL**] to a file. The write operation is not necessary in a standard ANSYS run but is provided as a convenience to users wanting a coded node file. Data are written in a coded format. The format used is (I8, 6G20.13) to write out *NODE,X,Y,Z,THXY,THYZ,THZX*. If the last number is zero (i.e., *THZX* = 0), or the last set of numbers are zero, they are not written but are left blank. Therefore, you must use a formatted read to process this file. Coordinate values are in the global Cartesian system.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Nodes>Write Node File**

---

---

## O Commands

**OCDATA**, VAL1, VAL2, VAL3, . . . , VAL14

Defines an ocean environment using non-table data.

[PREP7: Ocean](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

VAL1, VAL2, VAL3, . . . , VAL14

Values describing the basic ocean environment or a wave condition.

### Notes

The **OCDATA** command specifies non-table data that defines the ocean environment, such as the ocean section number, the depth of the ocean to the mud line, the ratio of added mass over added mass for a circular cross section, or the wave type to apply. The terms VAL1, VAL2, etc. are specialized according to the input set required for the given ocean environment.

The program interprets the data input via the **OCDATA** command within the context of the most recently issued **OCTYPE** command.

Input values in the order indicated.

The following **OCDATA** usage notes are available according to ocean data type:

[Ocean Data Type: Basic \(OCTYPE,,BASIC\)](#)

[Ocean Data Type: Wave \(OCTYPE,,WAVE\)](#)

### **Ocean Data Type: Basic (OCTYPE,,BASIC)**

#### VAL1

*DEPTH* -- The depth of the ocean (that is, the distance from mean water level--fixed at Z = 0 on the XY plane--to the mud line. This value is *required*.

#### VAL2

*MATOC* -- The material number of the ocean. This value is required and is used to input the required density. It is also used to input the viscosity if the Reynolds number is used (**OCTABLE**).

#### VAL3

*KFLOOD* -- The inside-outside fluid-interaction key, as follows:

0 -- The pressures inside and outside of the pipe are independent. This is the default behavior.

1 -- The pressure inside of the pipe is set to equal the pressure outside of the pipe.

#### VAL4

*ci* -- The ratio of added mass of the external fluid over added mass for a circular cross section. This value defaults to 1.0 if *ci* = 0.0 or no value is specified. If this value *should* be 0.0, enter a negative number. The added mass represents the mass of the external fluid (ocean water) that moves with the

pipe when the pipe moves in the element y and/or z directions during a dynamic analysis. Using the default presumes that the added mass is equal to the mass of the external fluid displaced by the pipe.

**VAL5**

*Cb* -- The ratio of buoyancy force used over buoyancy force based on the outside diameter and water density. This value defaults to 1.0 if *Cb* = 0.0 or no value is specified. If this value *should* be 0.0, enter a negative number.

**Ocean Data Type: Wave (OCTYPE,,WAVE)****VAL1**

*KWAVE* -- The wave type to apply to the current profile:

- 0 or AIRY -- Small amplitude Airy wave type without modifications (default).
- 1 or WHEELER -- Small amplitude wave type with Wheeler empirical modification of depth decay function.
- 2 or STOKES -- Stokes fifth order wave type.
- 3 or STREAMFUNCTION -- Stream function wave type.
- 5 or IRREGULAR -- Irregular wave type.
- 6 or SHELLNEWWAVE -- Shell new wave type.
- 7 or CONSTRAINED -- Constrained new wave type.
- 101+ -- API for computing particle velocities and accelerations due to waves and current:

101 through 200 -- Data preprocessed (via *KWAVE* = 0 logic).

201+ -- Data not preprocessed.

For more information, see the description of the [userPartVelAcc subroutine](#) in the *Programmer's Manual*.

**VAL2**

*THETA* -- Angle of the wave direction  $\theta$  from the global Cartesian X axis toward the global Cartesian Y axis (in degrees).

**VAL3**

*WAVELOC* (valid when *KWAVE* = 0 through 3, and 101+) -- The wave location type:

- 0 -- Waves act on elements at their actual locations (default).
- 1 -- Elements are assumed to be at wave peak.
- 2 -- Upward vertical wave velocity acts on elements.
- 3 -- Downward vertical wave velocity acts on elements.
- 4 -- Elements are assumed to be at wave trough.

*SPECTRUM* (valid when *KWAVE* = 5 through 7) -- The wave spectrum type:

- 0 -- Pierson-Moskowitz (default).
- 1 -- JONSWAP.
- 2 -- User-defined spectrum.

**VAL4**

*KCRC* -- The wave-current interaction key.

Adjustments to the current profile are available via the *KCRC* constant of the water motion table (illustrated in [Figure 4 \(p. 1151\)](#)). Typically, these options are used only when the wave amplitude is large relative to the water depth, such that significant wave-current interaction exists.



- 0 -- Use the current profile (as input) for wave locations below the mean water level, and the top current profile value for wave locations above the mean water level (default).
- 1 -- Linearly stretch or compress the current profile from the mud line to the top of the wave.
- 2 -- Similar to  $KCRC = 1$ , but also adjusts the current profile horizontally such that total flow continuity is maintained with the input profile. All wave directions  $\theta(j)$  must be identical.
- The following option is valid only when  $KWAVE = 5$  through 7:*
- 3 -- Nonlinear stretch or compress the current profile, as recommended in *API RP 2A Codes of Practice for Designing and Constructing Fixed Offshore Platforms*.

**VAL5**

*KMF* -- The MacCamy-Fuchs adjustment key, typically used only for larger-diameter pipes in relatively shallow water:

- 0 -- Do not apply the adjustment (default).
- 1 -- Apply the adjustment (valid only when  $KWAVE = 0$  through 3).

**VAL6**

*PRKEY* -- The wavelength wave-printout key:

- 0 -- No extra printout (default).
- 1 -- Include the extra printout.
- 2 -- Print wave component details (valid only when  $KWAVE = 5$  through 7).

**The following input values are valid only when  $KWAVE = 5$  through 7:****VAL7**

*APC* -- Activate apparent period calculation when a wave is superimposed upon a current:

- 0 -- Not activated (default).
- 1 -- Activated.

**VAL8**

*DSA* -- Stretching depth factor:

Stretching is performed between a distance of  $DSA * H_s$  below the mean water level (MWL) and the water surface, where  $H_s$  is the significant wave height measured from the MWL. No stretching occurs outside this range, or if the wave surface is below the MWL. If  $DSA * H_s$  is negative, stretching is performed between that level above the MWL and the water surface. The default  $DSA$  value is 0.5.

**VAL9**

*DELTA* -- Delta stretching parameter ( $0.0 \leq DELTA \leq 1.0$ ):

A value of 0.0 corresponds to Wheeler stretching under wave crests, 1.0 corresponds to linear extrapolation of kinematics at mean water level to crest. (Default = 0.3.) If zero is required, specify a small positive number (0.01 or less) instead.

**VAL10**

Wave kinematics factor or wave spreading angle:

*KINE* ( $KWAVE = 5$  or 7) -- Wave kinematics factor ( $0.0 < KINE \leq 1.0$ ). The factor is used to account for wave spreading by modifying the horizontal wave velocities and accelerations. A value of 1.0 corresponds to uni-directional wave with no spreading. (Default = 1.0, no spreading.)

*SPANGLE* ( $KWAVE = 6$ ) -- Wave spreading angle in degrees ( $0.0 \leq SPANGLE \leq 40.0$ ). The angle is used to compute a wave spreading factor to modify the horizontal wave kinematics for nearly unidirectional seas.  $SPANGLE = 0.0$  corresponds to no spreading. (Default = 0.0, no spreading.)

**VAL11**

Random seed value for phase angle generation, or wave crest amplitude value:

*SEED* (*KWAVE* = 5) -- Initial seed for random phase angle generation. (Default = 1.)

*AMPMAX* (*KWAVE* = 6) -- Maximum wave crest amplitude (distance between the mean water level and maximum wave crest).

*AMPCONST* (*KWAVE* = 7) -- Constrained wave crest amplitude (distance between the mean water level and wave crest).

**The following input values are valid only when *KWAVE* = 6 or 7:**

**VAL12**

*TOFF* -- Time offset at which the maximum wave crest will occur. (Default = 0.0.)

**VAL13**

*ROFF* -- Position offset along the wave direction where the maximum wave crest will occur. (Default = 0.0.)

**VAL14**

*EVOLVING* (*KWAVE* = 6) -- Activate evolving wave:

0 -- Not activated (default).

1 -- Activated.

*SEED* (*KWAVE* = 7) -- Initial seed for random phase angle generation. (Default = 1.)

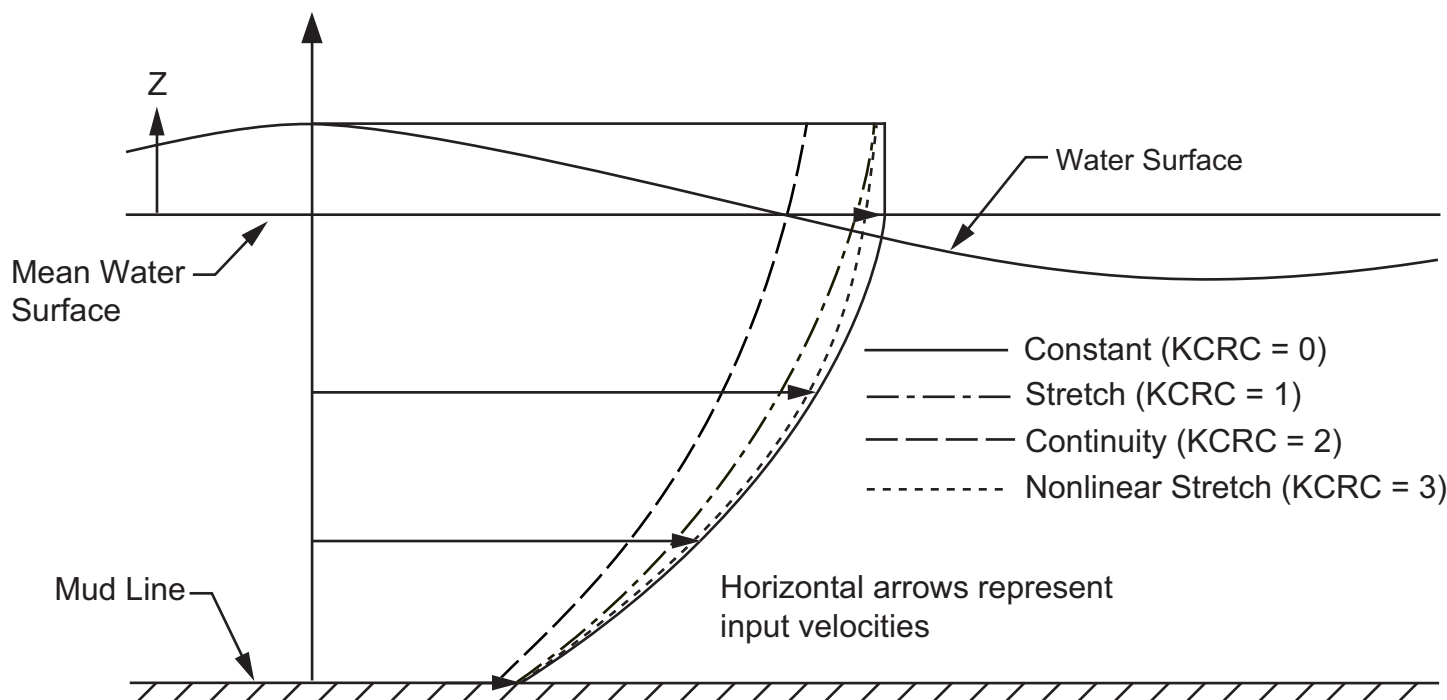
**Usage Hints for *KWAVE* = 5 Through 7**

Recommendations given in *API RP 2A Codes of Practice for Designing and Constructing Fixed Offshore Platforms* can be enforced via the following settings (valid only when *KWAVE* = 5 through 7):

- Set *KCRC* = 4 (*VAL4*) to apply nonlinear current stretching.
- Set *APC* = 1 (*VAL7*) to enable automatic computation of apparent wave period when a wave is superimposed upon a current.
- If *KWAVE* = 5 or 7, specify wave kinematics factor *KINE* (*VAL10*) to modify the horizontal velocities and accelerations due to wave spreading. The factor can be estimated from measurement or hindcast directional spectral wave data as  $\sqrt{(n+1)/(n+2)}$ , where *n* is the exponent in the cosine wave spreading function at the spectral peak frequency.
- If *KWAVE* = 6, specify wave spreading angle *SPANGLE* (*VAL10*) to modify the horizontal velocities and accelerations due to wave spreading. This parameter differs from the wave kinematics factor indicated in the API codes, but the principle is similar nevertheless.

For *VAL11* (when *KWAVE* = 6 or 7), if no maximum or constrained wave crest amplitude (*AMPCONST* or *AMPMAX*) is defined, the value is calculated based on the significant wave height obtained from the spectrum data (**OCTABLE**). For further information, see [Hydrodynamic Loads on Line Elements](#) in the [Theory Reference for the Mechanical APDL and Mechanical Applications](#).

For *VAL14*, the Shell new wave model (*KWAVE* = 6) is evolving (or *dispersive*) in that the crest height varies with time. By default, however, the wave is "frozen" so that it can be stepped through in a quasi-static manner.

**Figure 4 Velocity Profiles for Wave-Current Interaction**

## Menu Paths

Main Menu>Preprocessor>Sections>Pipe>Ocean>Add/Edit

**OCDELETE**, *OCFIRST*, *OCLAST*, *OCINC*, --, *LCHK*

Deletes ocean information from the ANSYS database.

PREP7:Ocean

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *OCFIRST*

First ocean ID to be deleted; defaults to first available ocean in the database.

### *OCLAST*

Last ocean ID to be deleted; defaults to last available ocean in the database.

### *OCINC*

Increment of the ocean ID; defaults to 1.

--

Unused field.

### *LCHK*

Specifies the level of element-associativity checking:

#### **NOCHECK**

No element-associativity check occurs. This option is the default.

**WARN**

When an ocean ID is associated with an element, ANSYS issues a message warning that the necessary entity has been deleted.

**CHECK**

The command terminates, and no ocean ID is deleted if it is associated with an element.

**Notes**

Deletes one or more specified ocean IDs and their associated data from the ANSYS database.

**Menu Paths**

**Main Menu>Preprocessor>Sections>Pipe>Ocean>Delete**

---

**OCLIST**, *SFIRST, SLAST, SINC, Details*

**Summarizes all currently defined ocean environments.**

[PREP7:Ocean](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**SFIRST**

First ocean environment to be summarized. Defaults to first available ocean ID in the database.

**SLAST**

Last ocean environment to be summarized. Defaults to last available ocean ID in the database.

**SINC**

Increment of the ocean ID. Defaults to 1.

**Details**

Determines the content of the summarized ocean information:

**BRIEF**

Lists only the ocean IDs requested. This option is the default.

**GROUP**

If a basic ocean environment calls for current and/or waves, this option lists them too.

**Notes**

The **OCLIST** command summarizes the ocean properties for all defined ocean environments in the current session of ANSYS.

**Menu Paths**

**This command cannot be accessed from a menu.**

---

**OCTABLE**, VAL1, VAL2, VAL3, VAL4, VAL5, VAL6

**Defines an ocean environment using table data.**

PREP7:Ocean

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

VAL1, VAL2, VAL3, . . . , VAL6

Values describing the basic ocean environment, a current condition, or a wave condition.

## Notes

The **OCTABLE** specifies table data that defines the ocean environment. The terms VAL1, VAL2, etc. are specialized according to the input set required for the given ocean environment.

The program interprets the data input via the **OCTABLE** command within the context of the most recently issued **OCTYPE** command.

There is no limit to the number of data input.

Input values in the order indicated.

The following **OCTABLE** usage notes are available according to ocean data type:

[Ocean Data Type: Basic \(OCTYPE,,BASIC\)](#)

[Ocean Data Type: Current \(OCTYPE,,CURR\)](#)

[Ocean Data Type: Wave \(OCTYPE,,WAVE\)](#)

### **Ocean Data Type: Basic (OCTYPE,,BASIC)**

Basic ocean data to provide in the value fields:

RE, CDy, CDz, CT, CMy, CMz

where

RE = Reynolds number for coefficients for this command. Input these values in *ascending* order from one command to the next.

CDy = Drag coefficient in the element y direction (normal).

CDz = Drag coefficient in the element z direction (normal). This value defaults to CDy.

CT = Drag coefficient in the element x direction (tangential).

CMy = Coefficient of inertia in the element y direction.

CMz = Coefficient of inertia in the element z direction.

### **Ocean Data Type: Current (OCTYPE,,CURR)**

Current data to provide in the value fields:

Z, W, Th, T

where

Z = Coordinate location of the drift current. Input these values in *descending* order from one command to the next. (The first Z value must be zero and the last one must be *-Depth*.) If the current is constant, only one **OCTABLE** command is necessary and this value is not required.

W = Velocity of the drift current at this location.

Th = Angle of the drift current from the global Cartesian X axis toward the global Cartesian Y axis (in degrees) at this location.

T = Temperature at this location.

## Ocean Data Type: Wave (OCTYPE,,WAVE)

When specifying an ocean wave type, issue the **OCTABLE** command to input either [wave location data](#) or [wave spectrum data](#).

### Wave Location Input Data

Wave location data to provide in the value fields (valid only when *KWAVE* = 0 through 3 on the **OCDATA** command):

*A, Per, Ps, Lw*

where

*A* = Peak-to-trough amplitude.

*Per* = Wave period.

*Ps* = Phase shift.

*Lw* = Wavelength. This value is optional.

#### Usage Notes

When using Stokes fifth order wave type (*KWAVE* = 2 on the **OCDATA** command), issue only one **OCTABLE** command. Wavelength (*Lw*) is not input.

When using stream function wave type (*KWAVE* = 3 on the **OCDATA** command),  $X(j) / (H * T * G)$  is defined in  $A(j)$ , where *j* is the number of the **OCTABLE** command. Also,  $DPT / LO = Per(2)$ ,  $L / LO = Per(3)$ ,  $H / DPT = Per(4)$ , and  $\psi / G * H * T = Per(5)$ . Phase shift is input in *Ps(1)*.

### Wave Spectrum Input Data

Wave spectrum data to provide in the value fields (valid only when *KWAVE* = 5 through 7 on the **OCDATA** command):

**SPECTRUM = 0** (Pierson-Moskowitz spectrum)

*HS, TP, NWC*

where

*HS* = Significant wave height of the spectrum.

*TP* = Peak period for the spectrum.

*NWC* = Number of wave components ( $1 \leq NWC \leq 1000$ ) to model the spectrum. (Default = 50.)

**SPECTRUM = 1** (JONSWAP spectrum)

*HS, TP, GAMMA, NWC*

where

*HS* = Significant wave height of the spectrum.

*TP* = Peak period for the spectrum.

*GAMMA* = Peak enhancement factor for the spectrum. (Default = 3.3.)

*NWC* = Number of wave components ( $1 \leq NWC \leq 1000$ ) to model the spectrum. (Default = 50.)

**SPECTRUM = 2** (User-defined spectrum)

*w, s, NWC*

*w* = Angular frequency (rad/s).

*s* = Spectral energy density (Length<sup>2</sup> / (rad/s))

*NWC* = Number of wave components ( $1 \leq NWC \leq 1000$ ) to model the spectrum. (Default = 50.)

### Hints for Wave Spectrum Input

When defining a Pierson-Moskowitz or JONSWAP spectrum ( $SPECTRUM = 0$  or  $1$ , respectively, on the **OCDATA** command), issue only one **OCTABLE** command.

When defining a Pierson-Moskowitz or JONSWAP spectrum for Shell new wave ( $KWAVE = 6$  on the **OCDATA** command),  $HS$  is calculated from the maximum wave crest amplitude ( $AMPMAX$  on the **OCDATA** command) if no value is specified. For further information, see [Hydrodynamic Loads on Line Elements](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

For a user-defined spectrum ( $SPECTRUM = 2$  on the **OCDATA** command), issue an **OCTABLE** command for *each* frequency data point defining the spectrum. Specify the frequency data in ascending order. The number of wave components ( $NWC$ ) is required on the *first* **OCTABLE** command only.

## Menu Paths

**Main Menu>Preprocessor>Sections>Pipe>Ocean>Add/Edit**

**OCTYPE**, *IDKEY*, *DataType*, *Name*, *OCIDC*, *OCIDW*

**Specifies the type of ocean environment data to follow.**

PREP7:Ocean

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

### **IDKEY**

The ocean environment, current, or wave ID number:

#### **OCID**

The ocean environment ID (matching the ocean ID assigned via the **SOCEAN** command). Specify this value when *DataType* = BASIC.

#### **OCIDC**

The ocean current ID. Specify this value when *DataType* = CURR.

#### **OCIDW**

The ocean wave ID. Specify this value when *DataType* = WAVE.

### **DataType**

The type of ocean data to be input following this command:

#### **BASIC**

The basic ocean environment, required for any ocean loading.

#### **CURR**

An optional drift current.

#### **WAVE**

An optional ocean wave state.

Specify basic, current, or wave input data via the **OCDATA** and **OCTABLE** commands. The example input fragment listed in the [Notes](#) section shows how to use the ocean environment data types.

### **Name**

An eight-character name for the ocean environment. An ocean name can consist of letters and numbers, but cannot contain punctuation, special characters, or spaces.

**OCIDC**

The optional ocean current ID. Specify this value only when *DataType* = BASIC.

**OCIDW**

The optional ocean wave ID. Specify this value only when *DataType* = BASIC.

**Notes**

The **OCTYPE** command specifies the type of ocean environment data to follow (basic, current, or wave) along with the appropriate ID number. Issue this command *before* inputting your ocean environment data (**OCDATA** and **OCTABLE**).

To associate your ocean environment with an element section, issue the **SOCEAN** command.

The ocean environment ID number *OCID* must be identical to the *OCID* value specified via the **SOCEAN** command.

Ocean loading applies only to pipe elements **PIPE288** and **PIPE289**.

The ocean ID number (*OCID*), ocean current ID number (*OCIDC*), or ocean wave ID number (*OCIDW*) must differ from each other and from any element section ID number.

An ocean current or wave ID number is accessible repeatedly. For example, it is not necessary to input an identical current table again just because the drag coefficients of the basic input table have changed.

The following example shows how you can use the basic (*DataType* = BASIC), current (*DataType* = CURR), and wave (*DataType* = WAVE) ocean data types within the context of a simple input file fragment:

```

Do=1.5           ! outside diameter
th=0.1          ! wall thickness
height=10       ! wave height
CS=2            ! surface current speed
Depth=100       ! water depth
matwat=2        ! material number id of the ocean
secpipe= 1      ! section number of the pipe
idwat=2         ! id number of the ocean
idcur=3         ! id number of the current loading
idwav=4         ! id number of the wave loading
!
sectype,secpipe,pipe,,pipetest
secdata,Do,th,16 ! 16 cells around circumference
socean,idwat     ! must be present for every section loaded by ocean
!
octype,idwat,basic,idcur,idwav
ocdata,Depth,matwat,0,-1,-1 ! suppress added mass and buoyancy
octable,,.5,.5,,2         ! CD = .5, CM = 2

octype,idcur,curr
octable, 0.0,CS           ! input free surface current speed
octable,-Depth,0.00      ! input ocean floor current speed of 0.0
!
octype,idwav,wave
ocdata,2                 ! request Stokes wave type
octable,height,8        ! wave period of 8 seconds

slist,all               ! lists pipe section AND
!                       mentions ocean loading (socean,idwat)
oclist,all              ! lists details of ocean loading

```



## Menu Paths

Main Menu>Preprocessor>Sections>Pipe>Ocean>Add/Edit

**OMEGA**, *OMEGX*, *OMEGY*, *OMEGZ*

Specifies the rotational velocity of the structure.

SOLUTION: Inertia

MP ME ST PR PRN DS DSS FL <> <> DY PP <> EME MFS

*OMEGX*, *OMEGY*, *OMEGZ*

Rotational velocity of the structure about the global Cartesian X, Y, and Z axes.

## Notes

This command specifies the rotational velocity of the structure about each of the global Cartesian axes (right-hand rule). Rotational velocities may be defined in these analysis types:

- Static (**ANTYPE**,STATIC)
- Harmonic (**ANTYPE**,HARMIC) -- Full or mode superposition
- Transient (**ANTYPE**,TRANS) -- Full or mode superposition
- Substructuring (**ANTYPE**,SUBSTR)
- Modal (**ANTYPE**,MODAL)

The **OMEGA** command supports tabular boundary conditions (*%TABNAME\_X%*, *%TABNAME\_Y%*, and *%TABNAME\_Z%*) for *OMEGA\_X*, *OMEGA\_Y*, and *OMEGA\_Z* input values (**\*DIM**) for full transient and harmonic analyses.

Rotational velocities are combined with the element mass matrices to form a body force load vector term. Units are radians/time. Related commands are **ACEL**, **CGLOC**, **CGOMGA**, **DCGOMG**, and **DOMEGA**.

If you have applied the Coriolis effect (**CORIOLIS**) using a [stationary reference frame](#), the **OMEGA** command takes the gyroscopic damping matrix into account for the elements listed in the "Stationary Reference Frame" heading in the notes section of the **CORIOLIS** command. The element axis must pass through the global Cartesian origin. ANSYS verifies that the rotation vector axis is parallel to the axis of the element; if not, the gyroscopic effect is not applied. After issuing the **OMEGA** command when the Coriolis or gyroscopic effect is present, a subsequently issued **CMOMEGA** command has no effect.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. ANSYS calculates a load vector and writes it to the *MODE* file, which you can apply via the **LVSCALE** command.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>Global  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Veloc>Global  
 Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>Global

**Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Veloc>Global**

---

**OPADD**, *NRES*, *NUM1*, *NUM2*, *C1*, *C2*

**Forms a set of optimization parameters by adding two sets.**

[OPTIMIZATION: Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NRES***

Number assigned to results set. If same as existing set, the existing values will be overwritten by these results (defaults to next available set number).

***NUM1***

Number of first design set to be used in operation.

***NUM2***

Number of second design set to be used in operation (may be blank).

***C1***

Scale factor applied to *NUM1* (defaults to 1.0).

***C2***

Scale factor applied to *NUM2* (defaults to 1.0).

## Notes

Forms a result set of parameters by adding two existing design sets according to the operation:

$$NRES = (C1 \times NUM1) + (C2 \times NUM2)$$

**OPADD** can also be used to scale results for a single set. If no set number is specified for the results set (*NRES*), it defaults to the next available set number. Because no compression of set numbers is done, the next available set number will be either 1 + the highest existing set number, or 1 + the highest set number used (if sets with higher numbers previously existed but have been removed).

## Menu Paths

**Main Menu>Design Opt>Design Sets>Combine**

---

**OPANL**, *Fname*, *Ext*, --

**Defines the analysis file to be used for optimization looping.**

[OPTIMIZATION: Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

**Ext**

Filename extension (8 character maximum).

--

Unused field.

## Command Default

If interactive, no default; if batch, use the batch input stream (File.BAT).

## Notes

The optimization looping file must be specified for an optimization which is performed interactively. The file must exist at the time **OPANL** is issued. In this file, where the **/PREP7** and **/OPT** commands occur, they must be the first nonblank characters on the line (i.e., do not use the \$ delimiter on any **/PREP7** or **/OPT** command lines).

## Menu Paths

Main Menu>Design Opt>Analysis File>Assign

## OPCLR

**Clears the optimization database.**

[OPTIMIZATION: Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Clears the optimization database. All settings are reset to their default values, and all design sets are deleted. This command is useful between multiple (independent) optimization analyses in the same batch run or interactive session.

## Menu Paths

Main Menu>Design Opt>Opt Database>Clear & Reset

## OPDATA, Fname, Ext, --

**Identifies the file where optimization data is to be saved.**

[OPTIMIZATION: Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

**Ext**

Filename extension (8 character maximum).

The extension defaults to OPT if *Fname* is blank.

--

Unused field.

## Command Default

Optimization data is saved on `File.OPT`.

## Notes

Identifies the file where optimization data is to be automatically saved during looping. Automatic saves occur at the end of each loop or iteration, and upon finishing out of the optimizer [**FINISH**]. (To save the data elsewhere in the optimizer, use **OPSAVE**.) Optimization data written to the file include DVs, SVs, saved design sets, analysis file name, etc. The data can be restored via the **OPRESU** command.

## Menu Paths

**Main Menu>Design Opt>Controls**

---

### **OPDEL, NSET1, NSET2**

**Deletes optimization design sets.**

[OPTIMIZATION: Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NSET1**

First design set in range to be deleted. Defaults to 0. If *NSET1* is blank and *NSET2* is  $\geq 1$ , all sets from the lowest existing set number to *NSET2* will be deleted. If *NSET1* = ALL, all design sets are removed.

**NSET2**

Last design set in range to be deleted. Defaults to *NSET1*.

## Notes

Deletes the design sets in a specified range. All sets occurring within that range (i.e.,  $NSET1 \leq N \leq NSET2$ , where N is an existing set number) are permanently removed from the optimization database. The original set numbers are retained for remaining design sets. Note that no compression of set numbers is done. Thus, if the highest set is deleted, the next available set number will be the previously existing highest set number + 1. (A maximum of 130 design sets can be stored in the optimization database.)

## Menu Paths

**Main Menu>Design Opt>Design Sets>Select/Delete**

---

**OPEQN**, *KFOBJ*, *KFSV*, *KWGHT*, *KOPPR*, *INOPT***Controls curve fitting for the subproblem approximation method.**

OPTIMIZATION: Specifications

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***KFOBJ***

Objective function curve fit:

- 0** Quadratic plus cross-term curve fit for objective function (default).
- 1** Linear curve fit.
- 2** Quadratic curve fit.
- 3** Quadratic plus cross-term fit (same as 0).

***KFSV***

State variables curve fit:

- 0** Quadratic curve fit for state variables (default).
- 1** Linear curve fit.
- 2** Quadratic curve fit (same as 0).
- 3** Quadratic plus cross-term curve fit.

***KWGHT***

Weighting factors:

- 0** Weights applied to design sets based on triple products of distance in design space, objective function values, and feasibility/infeasibility (*KWGHT* = 2, 3, and 4, below) (default).
- 1** All weights set to unity.
- 2** Weights based on distance in design space.
- 3** Weights based on objective function values.
- 4** Weights based on feasibility/infeasibility.

***KOPPR***

Optimization printout:

- 0** No printout of approximation data (default).

## OPERATE

---

**N**

Printout every  $n$ th optimization loop.

**INOPT**

Approximation reformulation:

**0**

Approximations are updated every loop (default).

**N**

Approximations are completely reformulated every  $n$ th loop.

## Notes

Controls the curve fitting for the optimization approximations used with the subproblem approximation method [**OPTYPE**,SUBP]. See [Function Approximations](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details.

## Menu Paths

**Main Menu>Design Opt>Method/Tool**

---

## OPERATE

Specifies "Operation data" as the subsequent status topic.

[POST26: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## OPEXE

Initiates optimization looping.

OPTIMIZATION: Run

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Initiates optimization looping using commands condensed from the analysis file [OPANL]. The type of optimization to be performed is specified by the OPTYPE command. Upon issuing this command, optimization looping begins and the design variables are updated as necessary based on the type of optimization chosen. Control remains with the ANSYS program until convergence or termination occurs, at which time the commands occurring after OPEXE are executed. This command is not allowed within a do-loop [\*DO].

## Menu Paths

Main Menu>Design Opt>Run

## OPFACT, Type

Defines the type of factorial evaluation to be performed.

OPTIMIZATION: Specifications

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Type

Type of factorial evaluation. Number of iterations is determined by the number of design variables ( $n$ ).

#### FULL

Perform a full factorial analysis (default);  $2^{**n}$  iterations are required.  $n$  must be  $\leq 7$ .

#### 1/2

Perform a 1/2 fractional factorial analysis.  $(1/2)^{*2^{**n}}$  iterations are required.  $n$  must be  $\leq 8$ .

#### 1/4

Perform a 1/4 fractional factorial analysis.  $(1/4)^{*2^{**n}}$  iterations are required.  $n$  must be  $\leq 9$ .

#### 1/8

Perform a 1/8 fractional factorial analysis.  $(1/8)^{*2^{**n}}$  iterations are required.  $n$  must be  $\leq 10$ .

#### 1/16

Perform a 1/16 fractional factorial analysis.  $(1/16)^{*2^{**n}}$  iterations are required.  $n$  must be  $\leq 10$ .

#### 1/32

Perform a 1/32 fractional factorial analysis.  $(1/32)^{*2^{**n}}$  iterations are required.  $n$  must be  $\leq 10$ .

#### 1/64

Perform a 1/64 fractional factorial analysis.  $(1/64)^{*2^{**n}}$  iterations are required.  $n$  must be  $\leq 10$ .

## Command Default

Perform a full factorial analysis.

## Notes

This command is valid for the factorial evaluation method of optimization [**OPTYPE**,FACT]. The factorial evaluation is defined as either full or fractional. The number of iterations performed is dependent on the number of design variables that have been defined ( $n$ ).

## Menu Paths

Main Menu>Design Opt>Method/Tool

---

### **OPFRST**, *NITR*, *SIZE*, *DELTA*

Defines specifications for the first order optimization method.

[OPTIMIZATION: Specifications](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ***NITR***

Maximum number of optimization iterations for the next execution [**OPEXE**]. Defaults to previously specified value, if any; otherwise defaults to 10.

#### ***SIZE***

Limit (in percent) that is applied to the size of each line search step. For example, setting *SIZE* = 10 will limit design variable changes to 10 percent of the maximum range of design space (defined by feasible design variable limits specified on **OPVAR** commands) at each iteration. Defaults to previously specified value, if any; otherwise, defaults to 100 (percent).

#### ***DELTA***

The forward difference (in percent) applied to the design variable range that is used to compute the gradient. For example, the shift in a DV is  $DELTA * (MAX - MIN) / 100$ , where MIN and MAX are specified on the **OPVAR** command. Defaults to previously specified value, if any; otherwise, defaults to 0.2 (percent).

## Command Default

*NITR* = 10, *SIZE* = 100, *DELTA* = 0.2.

## Notes

This command is valid for the first order method of optimization [**OPTYPE**,FIRST].

## Menu Paths

Main Menu>Design Opt>Method/Tool

---



**OPGRAD**, *Dset*, *DELTA*

**Specifies which design set will be used for gradient evaluation.**

OPTIMIZATION: Specifications

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Dset*****BEST**

Use the best design set as the evaluation point (default).

**LAST**

Use the last design set as the evaluation point.

***n***

Use design set number *n* as the evaluation point.

***DELTA***

The forward difference applied to the design variable range that is used to compute the gradient. For example, the shift in a design variable (DV) is defined as  $DELTA * (MAX - MIN) / 100$ , with MIN and MAX as specified on the **OPVAR** command. Defaults to 0.5.

**Command Default**

As described for each argument above.

**Notes**

This command is valid for the gradient evaluation method of optimization [**OPTYPE**,GRAD]. **OPGRAD** specifies which point (design set) in design space will be used to evaluate the gradient and what the forward difference will be.

**Menu Paths**

Main Menu>Design Opt>Method/Tool

**OPKEEP**, *Key*

**Specifies whether to save the best-set results and database file.**

OPTIMIZATION: Specifications

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Key***

Save key:

**OFF**

Do not save the results and database files for the best design set (default).

**ON**

Save results and database files (File.BRST and File.BDB) for the best design set during optimization looping.

## Command Default

Do not save the database and results for the best design set.

## Notes

Specifies whether or not the results file and database file corresponding to the best design set (based on all existing sets) should be saved during optimization. The saved files will be named `File.BRST` (or `File.BRTH` for thermal analysis, `File.BRMG` for magnetic analysis, `File.BRFL` for a FLOTRAN CFD analysis) and `File.BDB`. If `Key = ON`, both files will be saved at the end of each optimization loop which results in a new best design set. `File.RST` (results) and `File.DB` (database) will be saved as usual for the last design set, regardless of the `Key` setting. If multiple analyses are done within an optimization loop, then only the last analysis is considered for the `Key = ON` option. For example, if a thermal analysis is followed by a stress analysis, the files saved for the best set will be `File.BRST` and `File.BDB`, which will both correspond to the structural analysis.

## Menu Paths

Main Menu>Design Opt>Controls

**OPLFA**, *Name*, *Effect*, *EMIN*, *EMAX*

**Displays the results of a factorial evaluation.**

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Name

Parameter name. The parameter must have been previously defined as the objective function or a state variable [**OPVAR**].

### Effect

Display effects for parameter *Name*.

#### MAIN

Display main effects (default).

#### 2FAC

Display two-factor interactions.

#### 3FAC

Display three-factor interactions.

### EMIN

Minimum value of effect to be displayed. Default - computed minimum.

### EMAX

Maximum value of effect to be displayed. Default - computed maximum.

## Notes

Displays the results of a factorial analysis in bar chart form. Only the 10 largest interaction values will be displayed in the specified range (*EMIN* to *EMAX*). This command is only valid after a factorial evaluation [**OPTYPE**,**FACT**].

## Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Graph>Factorial

---

**OPLGR**, *Pname*, *Dvnam1*, *Dvnam2*, *Dvnam3*, *Dvnam4*, *Dvnam5*, *Dvnam6*

Graphs the results of a gradient evaluation.

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Pname*

Response parameter name. The parameter must have been previously defined as the objective function or a state variable [**OPVAR**].

*Dvnam1*, *Dvnam2*, *Dvnam3*, . . . , *Dvnam6*

Design variable names.

## Notes

This command is only valid after a gradient evaluation [**OPTYPE**,GRAD]. *Pname* is graphed with respect to a plus or minus 1% change in design variable (*Dvnam1*, ... *Dvnam6*).

## Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Graph>Gradient

---

**OPLIST**, *SET1*, *SET2*, *LKEY*

Displays the parameters for design sets.

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*SET1*, *SET2*

Display values of all analysis parameters from *SET1* (defaults to the highest existing set) to *SET2* (defaults to *SET1*). If *SET1* = ALL, display for all sets.

*LKEY*

Listing key:

**0**

List only scalar parameters related to optimization.

**1**

List all analysis scalar parameters.

## Notes

Displays the values of the parameters for specified design sets. Note that a **\*STATUS** command may also be issued to display various optimization data.

## Menu Paths

Main Menu>Design Opt>Design Sets>List

---

### OPLOOP, *Read, Dvar, Parms*

Specifies controls for optimization looping.

[OPTIMIZATION: Specifications](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Read*

Indicates where to begin reading the analysis file during optimization looping. Note that the **/PREP7** command (and the **/OPT** command) must occur as the first nonblank characters on a line in the analysis file (i.e., do not use the \$ delimiter). Defaults to previously specified label, if any; otherwise, defaults to TOP:

##### **TOP**

Read from the first line.

##### **PREP**

Read from the first occurrence of **/PREP7**.

#### *Dvar*

Indicates how to treat parameters which have been designated as design variables (DVs) during optimization looping. Defaults to previously specified label, if any; otherwise, defaults to IGNORE:

##### **IGNORE**

Do not process DV parameter assignments in the analysis file during looping.

##### **PROCESS**

Process DV parameter assignments in the analysis file during looping.

#### *Parms*

Indicates which types of parameters to save during optimization looping. Defaults to previously specified label, if any; otherwise, defaults to SCALAR:

##### **SCALAR**

Save scalar parameters only.

##### **ALL**

Save all parameters, both scalar and array.

## Command Default

*Read* = TOP, *Dvar* = IGNORE, and *Parms* = SCALAR.

## Menu Paths

Main Menu>Design Opt>Controls

---

**OPLSW**, *Pname*, *Dvnam1*, *Dvnam2*, *Dvnam3*, *Dvnam4*, *Dvnam5*, *Dvnam6*

**Graphs the results of a global sweep generation.**

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Pname***

Response parameter name. The parameter must have been previously defined as the objective function or a state variable (**OPVAR**).

***Dvnam1*, *Dvnam2*, *Dvnam3*, . . . , *Dvnam6***

Design variable names.

## Notes

This command is only valid after a global sweep generation [**OPTYPE**,SWEEP]. *Pname* is graphed with respect to the specified design variables (*Dvnam1*, ... *Dvnam6*). The design variables are normalized (0 to 1) along the X-axis.

## Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Graph>Sweeps

## OPMAKE

**Creates a design set using active scalar parameter values.**

OPTIMIZATION: Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

The next available design set number is assigned to the new design set. Because no compression of set numbers is done, the next available set number will be either 1 + the highest existing set number, or 1 + the highest set number used (if sets with higher numbers previously existed but have been removed). Multiple design sets can be created by repeated use of **PARRES** (to read parameters from a file) and **OPMAKE**.

## Menu Paths

Main Menu>Design Opt>Design Sets>Create

**OPNCONTROL**, *Lab*, *VALUE*, *NUMSTEP*

Sets decision parameter for automatically increasing the time step interval.

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**Lab****DOF**

DOF label used to base a decision for increasing the time step (substep) interval in a nonlinear or transient analysis. The only DOF label currently supported is TEMP.

**OPENUPFACTOR**

Factor for increasing the time step interval. Use *Lab* = OPENUPFACTOR (when **AUTOTS,ON** is invoked) and specify a *VALUE* greater than 1.0. Default for OPENUPFACTOR is 1.5. The upper limit for OPENUPFACTOR is 10.0. However, we recommend using a value not greater than 3.0.

**VALUE, NUMSTEP**

Two values used in algorithm for deciding if the time step interval can be increased. The time step interval is increased if the maximum absolute value of the incremental solution at the specified DOF label is less than *VALUE* for the number of contiguous time steps specified by *NUMSTEP*. The default for *VALUE* is 0.1 and the default for *NUMSTEP* is 3. *NUMSTEP* is used only when *Lab* = DOF.

**Notes**

This command is available only for nonlinear or full transient analysis, and only when **SOLCONTROL,ON**.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Open Control**

**Main Menu>Solution>Load Step Opts>Nonlinear>Open Control**

**OPPRNT**, *Key*

Activates detailed optimization summary printout.

[OPTIMIZATION: Specifications](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Key**

Summary key:

**OFF**

Do not print details (default).

**ON**

Print the detailed summary.

**FULL**

Same as *Key* = ON, but includes a full list of all design sets.

**Notes**

Activates detailed optimization summary printout to the primary output during optimization analysis.

## Menu Paths

Main Menu>Design Opt>Controls

---

### OPRAND, *NITR*, *NFEAS*

Defines the number of iterations for a random optimization.

[OPTIMIZATION: Specifications](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *NITR*

Maximum number of optimization iterations for the next execution [[OPEXE](#)]. Defaults to previously specified value, if any; otherwise, defaults to 1.

#### *NFEAS*

Desired total number of feasible design sets (including existing feasible sets). Once *NFEAS* feasible sets are achieved, random iterations are terminated. If 0 is input, *NITR* random iterations will be performed independent of the number of feasible designs that are obtained. Defaults to previously specified value, if any; otherwise, defaults to 0.

## Command Default

Perform 1 iteration.

## Notes

This command is valid for the random design generation method of optimization [[OPTYPE,RAND](#)].

## Menu Paths

Main Menu>Design Opt>Method/Tool

---

### OPRESU, *Fname*, *Ext*, --

Reads optimization data into the optimization database.

[OPTIMIZATION: Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

#### *Ext*

Filename extension (8 character maximum).

The extension defaults to OPT if *Fname* is blank.

--  
Unused field.

## Notes

Reads optimization data from the specified file into the optimization database. Optimization data must have been previously saved to this file [**OPSAVE**], or automatically written to this file in an earlier optimization analysis. All existing parameters will be deleted and replaced by those stored on the resumed file. (To save existing parameters, use **PARSAV** before **OPRESU**.)

## Menu Paths

Main Menu>Design Opt>Opt Database>Resume

---

### OPRFA, *Name*

Prints the results of a factorial evaluation.

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Name*

Parameter name. The parameter must have been previously defined as the objective function or a state variable [**OPVAR**]. If *Name* = ALL, print results for all optimization response variables (default).

## Notes

This command is only valid after a factorial evaluation [**OPTYPE**,FACT].

## Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Print

---

### OPRGR, *Name*

Prints the results of a gradient evaluation.

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Name*

Parameter name. The parameter must have been previously defined as the objective function or a state variable [**OPVAR**]. If *Name* = ALL, print results for all optimization response variables (default).

## Notes

This command is only valid after a gradient evaluation [**OPTYPE**,GRAD].



## Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Print

---

### OPRSW, *Name*

Prints the results of a global sweep generation.

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Name*

Parameter name. The parameter must have been previously defined as the objective function or a state variable (**OPVAR**). If *Name* = ALL, print results for all optimization response variables (default).

## Notes

This command is only valid after a global sweep generation [**OPTYPE**,SWEEP].

## Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Print

---

### OPSAVE, *Fname*, *Ext*, --

Writes all optimization data to a file.

OPTIMIZATION: Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

#### *Ext*

Filename extension (8 character maximum).

The extension defaults to OPT if *Fname* is blank.

--

Unused field.

## Notes

Writes all optimization data to File.OPT or a named file. Saved data includes optimization data only (DVs, SVs, saved design sets, analysis file name, etc.). The data can be restored [**OPRESU**] for an optimization restart.

## Menu Paths

Main Menu>Design Opt>Opt Database>Save

---

### OPSEL, *NSEL*

Selects design sets for subsequent optimization looping.

[OPTIMIZATION: Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *NSEL*

Number of best design sets to be selected. If *NSEL* = -1, select all feasible designs. If *NSEL* is positive and no objective function is defined, the following occurs: all infeasible design sets are removed; next the lowest number (oldest) feasible design sets are removed until *NSEL* sets are left (i.e., the latest feasible design sets are kept).

## Notes

A number of best design sets or all feasible design sets may be specified. ("Best" design sets are determined by the objective function value for each feasible design.) All design sets not selected are permanently removed from the optimization database. The original set numbers are retained for remaining design sets. Note that no compression of set numbers is done. Thus, if the highest set is removed, the next available set will still be the previously used highest set number + 1. (A maximum of 130 design sets can be stored in the optimization database.)

## Menu Paths

Main Menu>Design Opt>Design Sets>Select/Delete

---

### OPSUBP, *NITR*, *NINFS*

Defines number of iterations for subproblem approximation method.

[OPTIMIZATION: Specifications](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *NITR*

Maximum number of optimization iterations for the next execution [[OPEXE](#)]. Defaults to previously specified value, if any; otherwise, defaults to 30.

#### *NINFS*

Maximum number of consecutive infeasible design sets that are allowed. Once *NINFS* consecutive infeasible sets are encountered, iterations are terminated. Defaults to previously specified value, if any; otherwise, defaults to 7.

## Notes

This command is valid for the subproblem approximation method of optimization [[OPTYPE](#),SUBP].

## Menu Paths

Main Menu>Design Opt>Method/Tool

---

**OPSWEEP**, *Dset*, *NSPS*

Specifies the reference point and number of evaluation points for a sweep generation.

OPTIMIZATION: Specifications

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Dset*

**BEST**

Use the best design set as the reference point (default).

**LAST**

Use the last design set as the reference point.

*N*

Use design set number N as the reference point.

*NSPS*

Number of evaluation points used for each design variable sweep. Two to ten evaluation points are allowed. Defaults to 2 (i.e., the minimum and maximum design variable values).

## Command Default

As described by the arguments above.

## Notes

This command is valid for the sweep evaluation method of optimization [**OPTYPE**,SWEEP].

## Menu Paths

Main Menu>Design Opt>Method/Tool

---

## /OPT

Enters the design optimizer.

SESSION: Processor Entry

OPTIMIZATION: Specifications

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Enters the ANSYS optimizer for design optimization by iterative analyses. Upon issuing this command, if there are no existing design sets, all currently defined numerical scalar parameters are established as design set number 1.

This command is valid only at the Begin Level.

## Menu Paths

Main Menu>Design Opt

**OPTYPE**, *Mname*

**Specifies the optimization method to be used.**

OPTIMIZATION: Specifications

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS  
Product Restrictions

### *Mname*

Name of the optimization method to be used:

#### **SUBP**

Subproblem approximation method. Specifications for this method are defined on the **OPSUBP** command.

#### **FIRST**

First order optimization method. Specifications for this method are defined on the **OPFRST** command.

#### **RAND**

Perform random iterations. Specifications for this method are defined on the **OPRAND** command.

#### **RUN**

Perform a single optimization run with current parameter values for the defined design variables.

#### **FACT**

Perform factored design iterations. Specifications for this method are defined on the **OPFACT** command.

#### **GRAD**

Compute a gradient (design sensitivity) at a point in design space. Specifications for this method are defined on the **OPGRAD** command.

#### **SWEEP**

Perform global sweeps through global design space starting from a single design set. Specifications for this method are defined on the **OPSWEEP** command.

#### **USER**

User-supplied external optimization. See the *Advanced Analysis Techniques Guide* for user optimization information (USEROP routine). Parameters used for this method are defined on the **OPUSER** command.

## Command Default

Perform a single optimization run (*Mname* = RUN) with current parameter values for the defined design variables.

## Notes

Specifies the optimization method to be used upon execution [**OPEXE**].

## Product Restrictions

Command Option Mname	Available Products
SUBP	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS
FIRST	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS
OPRAND	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS
RUN	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS
FACT	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS
GRAD	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS
SWEEP	MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS
USER	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS

## Menu Paths

Main Menu>Design Opt>Method/Tool

**OPUSER**, *NITR*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*

Defines specifications for user-supplied external optimization.

[OPTIMIZATION: Specifications](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### *NITR*

Maximum number of optimization iterations for the next execution [**OPEXE**]. Defaults to previously specified value, if any; otherwise, defaults to 1.

### *VAL1, VAL2, VAL3, . . . , VAL8*

Numerical *values* (double precision) for user-supplied optimizer. Values may be the results of parameter evaluations. Values are written to `File.OPT` after the parameter set data. *VAL1--VAL8* default to previously specified values, if any; otherwise, default to 0.

## Notes

Defines the number of iterations and the parameter values for user-supplied external optimization [**OPTYPE,USER**]. Values are passed to the user routine (USEROP) through the optimization common block.

## Menu Paths

Main Menu>Design Opt>Method/Tool

**OPVAR**, *Name*, *Type*, *MIN*, *MAX*, *TOLER*

**Specifies the parameters to be treated as optimization variables.**

OPTIMIZATION: Specifications

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Name**

Parameter name (must be a scalar parameter).

**Type**

Optimization variable type:

**DV**

Design variable. *MAX* must also be specified.

**SV**

State variable, also referred to as a constrained variable. (Usually defined via a **\*GET** command.) *MIN*, or *MAX*, or both must be specified.

**OBJ**

Objective function (variable to be minimized). Only one objective function is allowed. *MIN* and *MAX* are not used.

**DEL**

Deletes this optimization variable (does not delete the parameter). This option only has meaning if the parameter *Name* was previously defined as an optimization variable (DV, SV, or OBJ). The parameter retains the value assigned during the last loop. *MIN* and *MAX* are not used.

**MIN**

Minimum value for this parameter. For *Type* = DV, *MIN* must be greater than 0.0 and defaults to  $0.001 * (MAX)$ . For *Type* = SV, there will be no lower limit if *MIN* is left blank, but if *MIN* = 0.0, the lower limit is zero. *MIN* is ignored for *Type* = OBJ. *MIN* is also ignored for *Type* = DEL (except for the special case when *Name* = (blank) and *Type* = DEL, as described next).

When the GUI is on, if a **Delete** operation (in a [Design Variables](#) or [State Variables](#) dialog box) writes this command to a log file (Jobname.LOG or Jobname.LGW), you will observe that *Name* is blank, *Type* = DEL, and *MIN* is an integer number. In this case, the GUI has assigned a value of *MIN* that corresponds to the location of a chosen variable name in the dialog box's list. It is *not* intended that you type in such a location value for *MIN* in an ANSYS session. However, a file that contains a GUI-generated **OPVAR** command of this form can be used for batch input or for use with the **/INPUT** command.

**MAX**

Maximum value for this parameter. Required for *Type* = DV and ignored for *Type* = OBJ or DEL. For *Type* = SV, there will be no upper limit if *MAX* is left blank, but if *MAX* = 0.0, the upper limit is zero.

**TOLER**

For *Type* = DV and OBJ, acceptable change between loops for convergence; defaults to  $0.01 * (\text{current value})$  for OBJ and  $0.01 * (MAX - MIN)$  for DV. For *Type* = SV, defines feasibility tolerance. For double-sided SV limits (*MAX* and *MIN* specified), the default is  $0.01 * (MAX - MIN)$ . For a single-sided SV limit (*MAX* or *MIN* specified), the default is  $0.01 * (\text{specified limit})$  or, if the absolute value of the limit is  $< 1$ , the default is  $0.01 * (\text{current SV value})$ . *TOLER* is ignored for *Type* = DEL.

**Command Default**

No optimization parameters defined.

## Notes

Specifies the parameters to be treated as optimization variables. Variable constraints (*MIN* and *MAX*) and tolerances (*TOLER*) are also specified on this command. If the specified parameter (*Name*) does not exist at the time **OPVAR** is issued, the **OPVAR** command defines that parameter and assigns a value of zero to it. Up to 60 DV and 100 SV parameters are allowed.

## Menu Paths

Main Menu>Design Opt>Design Variables  
 Main Menu>Design Opt>Objective  
 Main Menu>Design Opt>State Variables

---

## OUTOPT

Specifies "Output options" as the subsequent status topic.

SOLUTION: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (*Jobname.LOG*) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Solution>Output Options

---

## OUTPR, *Item*, *Freq*, *Cname*

Controls the solution printout.

SOLUTION: Misc Loads

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Item*

Item for print control:

#### **BASIC**

Basic quantities (nodal DOF solution, nodal reaction loads, and element solution) (default).

#### **NSOL**

Nodal DOF solution.

#### **RSOL**

Nodal reaction loads.

**ESOL**

Element solution.

**NLOAD**

Element nodal loads. When **nonlinear stabilization** is active, the stabilization force/moments are also printed.

**SFOR**

Stabilization force/moment at the applicable nodes (valid only when **nonlinear stabilization** is active).

**VENG**

Element energies. When **nonlinear stabilization** is active, the energy dissipation due to stabilization is also printed.

**V**

Nodal velocity (applicable to structural transient analysis only (**ANTYPE,TRANS**)).

**A**

Nodal acceleration (applicable to structural transient analysis only (**ANTYPE,TRANS**)).

**ALL**

All of the above solution items.

**Freq**

Print solution for this item every  $Freq^{\text{th}}$  (and the last) substep of each load step. If  $-n$ , print up to  $n$  equally spaced solutions (for automatic time stepping). If NONE, suppress all printout for this item for this load step. If ALL, print solution for this item for every substep. If LAST, print solution for this item only for the last substep of each load step. For a modal analysis, use NONE or ALL.

**Cname**

Name of the component, created with the **CM** command, defining the selected set of nodes or elements for which this specification is active. If blank, the set is all entities.

---

**Note**

The component named must be of the same type as the item, i.e. nodal or element. A component name is not allowed with the BASIC or ALL labels.

## Command Default

No printout.

## Notes

Controls the solution items to be printed, the frequency with which they are printed (in static, transient, or full harmonic analyses), and the set of nodes or elements to which this specification applies (in static, transient, or full harmonic analyses). An item is associated with either a node (**NSOL**, **RFORCE**, **V**, and **A** items) or an element (all of the remaining items). The specifications are processed in the order that they are input. Up to 50 specifications (**OUTPR** and **OUTRES** commands combined) may be defined. Use **OUTPR,STAT** to list the current specifications and use **OUTPR,ERASE** to erase all the current specifications.

As described above, **OUTPR** writes some or all items (depending on analysis type) for all elements. To restrict the solution printout, use **OUTPR** to selectively suppress ( $Freq = \text{NONE}$ ) the writing of solution data, or first suppress the writing of all solution data (**OUTPR,ALL,NONE**) and then selectively turn on the writing of solution data with subsequent **OUTPR** commands.



If the generalized plane strain feature is active and **OUTPR** is issued, the change of fiber length at the ending point during deformation and the rotation of the ending plane about X and Y during deformation will be printed if any displacement at the nodes is printed. The reaction forces at the ending point will be printed if any reaction force at the nodes is printed.

Nodal reaction loads ( $Item = RSOL$ ) are processed according to the specifications listed for the **PRRSOL** command.

Result printouts for interactive sessions are suppressed for models with more than 10 elements.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Solu Printout**  
**Main Menu>Solution>Load Step Opts>Output Ctrls>Solu Printout**

---

**/OUTPUT**, *Fname*, *Ext*, *--*, *Loc*

**Redirects text output to a file or to the screen.**

**SESSION: Run Controls**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

Filename and directory path (248 character maximum, including directory) to which text output will be redirected (defaults to `Jobname` if *Ext* is specified). For interactive runs, *Fname* = `TERM` (or blank) redirects output to the screen. For batch runs, *Fname* = blank (with all remaining command arguments blank) redirects output to the default system output file.

### *Ext*

Filename extension (8 character maximum).

*--*

Unused field.

### *Loc*

Location within a file to which output will be written:

#### **(blank)**

Output is written starting at the top of the file (default).

#### **APPEND**

Output is appended to the existing file.

## Command Default

Text output is written to the screen for interactive runs and to the system output file for batch runs (see the [Basic Analysis Guide](#)).

## Notes

Text output includes responses to every command and GUI function, notes, warnings, errors, and other informational messages. Upon execution of **/OUTPUT**, *Fname*, *Ext*, . . . , all subsequent text output is redirected to the file *Fname*.*Ext*. To redirect output back to the default location, issue **/OUTPUT** (no arguments).

---

### Note

When using the GUI, output from list operations [**NLIST**, **DLIST**, etc.] is always sent to a list window regardless of the **/OUTPUT** setting. The output can then be saved on a file or copied to the **/OUTPUT** location using the *File* menu in the list window.

This command is valid in any processor.

## Menu Paths

**Utility Menu>File>Switch Output to>File**  
**Utility Menu>File>Switch Output to>Output Window**

---

## OUTRES, *Item*, *Freq*, *Cname*

**Controls the solution data written to the database.**

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *Item*

Results item for database and file write control:

#### **ALL**

All solution items except SVAR and LOCI. This value is the default.

#### **CINT**

J-integral results.

#### **ERASE**

Resets **OUTRES** specifications to their default values.

#### **STAT**

Lists the current **OUTRES** specifications.

#### **BASIC**

Write only NSOL, RSOL, NLOAD, STRS, FGRAD, and FFLUX records to the results file and database.

#### **NSOL**

Nodal DOF solution.

#### **RSOL**

Nodal reaction loads.

#### **V**

Nodal velocity (applicable to structural full transient analysis only (**ANTYPE,TRANS**)).

#### **A**

Nodal acceleration (applicable to structural full transient analysis only (**ANTYPE,TRANS**)).

**ESOL**

Element solution (includes all items following):

**NLOAD**

Element nodal, input constraint, and force loads (also used with the **/POST1** commands **PRRFOR**, **NFORCE**, and **FSUM** to calculate reaction loads).

**STRS**

Element nodal stresses.

**EPEL**

Element elastic strains.

**EPTH**

Element thermal, initial, and swelling strains.

**EPPL**

Element plastic strains.

**EPCR**

Element creep strains.

**FGRAD**

Element nodal gradients.

**FFLUX**

Element nodal fluxes.

**LOCI**

Integration point locations.

**SVAR**

State variables (used only by USERMAT).

**MISC**

Element miscellaneous data (SMISC and NMISC items of the **ETABLE** command).

**Freq**

Specifies how often (that is, at which substeps) to write the specified solution results item. The following values are valid:

Value	Description
$n$	Writes the specified results item every $n$ th (and the last) substep of each load step.
$-n$	Writes up to $n$ equally spaced solutions (for automatic loading).
NONE	Suppresses writing of the specified results item for all substeps.
ALL	Writes the solution of the specified solution results item for every substep. This value is the default for a harmonic analysis ( <b>ANTYPE,HARMIC</b> ) and for any expansion pass ( <b>EXPASS,ON</b> ).
LAST	Writes the specified solution results item only for the last substep of each load step. This value is the default for a static ( <b>ANTYPE,STATIC</b> ) or transient ( <b>ANTYPE,TRANS</b> ) analysis.

Value	Description
%array%	<p>Where <i>array</i> is the name of an <math>n \times 1 \times 1</math> dimensional array parameter defining <math>n</math> key times, the data for the specified solution results item is written at those key times.</p> <p>Key times in the array parameter must appear in ascending order. Values must be greater than or equal to the beginning values of the load step, and less than or equal to the ending time values of the load step.</p> <p>For multiple-load-step problems, either change the parameter values to fall between the beginning and ending time values of the load step or erase the current settings and reissue the command with a new array parameter.</p> <p>For more information about defining array parameters, see the <b>*DIM</b> command documentation.</p>

**Cname**

The name of the component, created with the **CM** command, defining the selected set of elements or nodes for which this specification is active. If blank, the set is all entities. A component name is not allowed with the ALL, BASIC, or RSOL items.

**Command Default**

The **OUTRES** command writes the specified solution results item for every substep. The exceptions to the default behavior are as follows:

- For static (**ANTYPE**,STATIC) or transient (**ANTYPE**,TRANS) analyses, the default is to write the specified solution results item for the last substep of each load step.
- For a harmonic analysis (**ANTYPE**,HARMIC) and any expansion pass, the default is to write the specified solution results item for every substep.
- For reduced or mode superposition transients, the default is to write the reduced displacements file for every 4th substep (as well as the last substep); however, if gap conditions exist, the default is to write the specified solution results item of every substep.

**Notes**

The **OUTRES** command allows you to specify the following:

- The solution item (*Item*) to write to the database (and to the reduced displacement and results files)
- The frequency (*Freq*) at which the solution item is written (applicable to static, transient, or full harmonic analyses)
- The set of elements or nodes (*Cname*) to which your specification applies.

The command generates a specification for controlling data storage for each substep, activating storage of the specified results item for the specified substeps of the solution *and* suppressing storage of that item for *all other substeps*.

You can issue multiple **OUTRES** commands in an analysis. After the initial command creating the storage specification, subsequent **OUTRES** commands *modify* the specification set for each substep. The command processes your specifications at each substep in the order in which you input them. If you specify a given solution item twice, output is based upon the *last* specification. Therefore, issue multiple **OUTRES** commands carefully and in the proper sequence.

Besides **OUTRES**, another output-control command named **OUTPR** exists which controls solution *printout*. You can issue up to 50 output-control commands (either **OUTRES** or **OUTPR**, or some combination of both) in an analysis.

---

### Note

Issuing **OUTRES,ERASE** erases the existing output specifications and resets the counted number of **OUTRES** commands to zero. Issuing **OUTPR,ERASE** affects the **OUTPR** command in the same way.

A given **OUTRES** command has no effect on results items *not* specified. For example, an **OUTRES,ESOL,LAST** command does not affect NSOL data; that is, it neither activates nor suppresses NSOL data storage in any substep.

---

### Caution

In the results-item hierarchy, certain items are subsets of other items. For example, element solution (ESOL) data is a subset of all (ALL) solution data. Therefore, an **OUTRES,ALL** command can affect ESOL data. Likewise, an **OUTRES** command that controls ESOL data can affect a *portion* of ALL data. The [example OUTRES commands](#) illustrate the interrelationships between results items and the necessity for employing the **OUTRES** command thoughtfully.

To suppress ALL data at every substep, issue an **OUTRES,ALL,NONE** command. (An **OUTRES,ERASE** command *does not* suppress ALL data at every substep.)

The NSOL, RSOL, V, and A solution items are associated with nodes. All remaining solution items are associated with elements.

The boundary conditions (constraints and force loads) are written to the results file only if either nodal or reaction loads (NLOAD or RSOL items) are also written.

When specifying a *Freq* value, observe the following:

- For a modal analysis, the only valid values are NONE or ALL.
- If you issue multiple **OUTRES** commands during an analysis, you cannot specify a [key time array parameter](#) (*%array%*) in a given **OUTRES** command and then specify a different *Freq* option in a subsequent **OUTRES** command.

For a coupled-field analysis using the ANSYS Multi-field Solver, *Freq* cannot be NONE for *Item* = NSOL or ESOL because interface loads have to be retrieved from the database.

The **OUTRES** command is also valid in **/PREP7**.

## Example

When issuing an **OUTRES** command, think of a matrix in which you set switches on and off. When a switch is on, a results item is stored for the specified substep. When a switch is off, a results item is suppressed for a specified substep.

Assuming a static (**ANTYPE,STATIC**) analysis, this example shows how the matrix looks after issuing each **OUTRES** command in this six-substep solution.

```
NSUBST, 6
OUTRES, ERASE
OUTRES, NSOL, 2
OUTRES, ALL, 3
OUTRES, ESOL, 4
SOLVE
```

To simplify the example, only a subset of the available solution items appears in the matrix.

**OUTRES,ERASE** -- After issuing this command, the default output specifications are in effect, as shown:

Substep	Results Item Specification						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	off	off	off	off	off	off	off
3	off	off	off	off	off	off	off
4	off	off	off	off	off	off	off
5	off	off	off	off	off	off	off
6	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>

**OUTRES,NSOL,2** -- This command modifies the initial specifications so that NSOL is turned on for substeps 2, 4 and 6, and turned off for substeps 1, 3 and 5, as shown:

Substep	Results Item Specification						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	<b>ON</b>	off	off	off	off	off	off
3	off	off	off	off	off	off	off
4	<b>ON</b>	off	off	off	off	off	off
5	off	off	off	off	off	off	off
6	ON	ON	ON	ON	ON	ON	ON

**OUTRES,ALL,3** -- This command further modifies the specifications so that ALL is turned on for substeps 3 and 6, and turned off for substeps 1, 2, 4 and 5, as shown:

Substep	Results Item Specification						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
		NLOAD	STRS	FGRAD	EPEL	EPTH	
1	off	off	off	off	off	off	off
2	<b>off</b>	off	off	off	off	off	off
3	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>
4	<b>off</b>	off	off	off	off	off	off
5	off	off	off	off	off	off	off
6	ON	ON	ON	ON	ON	ON	ON

**OUTRES,ESOL,4** -- This command once again modifies the specifications so that ESOL is turned on for the fourth and last substeps, and turned off for substeps 1, 2, 3 and 5, as shown:

Substep	Results Item Specification						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
		NLOAD	STRS	FGRAD	EPEL	EPTH	
1	off	off	off	off	off	off	off
2	off	off	off	off	off	off	off
3	ON	ON	<b>off</b>	<b>off</b>	<b>off</b>	<b>off</b>	<b>off</b>
4	off	off	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>	<b>ON</b>
5	off	off	off	off	off	off	off
6	ON	ON	ON	ON	ON	ON	ON

## SOLVE

When obtaining the solution, results data are stored as follows:

Substep	Results Items Stored
1	<i>No data</i>
2	<i>No data</i>
3	NSOL and RSOL data
4	ESOL data
5	<i>No data</i>
6	ALL data

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic**

**Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrl>DB/Results File**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Basic**

**Main Menu>Solution>Load Step Opts>Output Ctrl>DB/Results File**

---



---

## P Commands

### **PADELE**, *DELOPT*

**Deletes a defined path.**

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *DELOPT*

Path delete option (one of the following):

#### **ALL**

Delete all defined paths.

#### **NAME**

Delete a specific path from the list of path definitions. (Substitute the actual path name for NAME.)

### **Command Default**

Deletes the currently active path.

### **Notes**

Paths are identified by individual path names. To review the current list of path names, issue the command [PATH,STATUS](#).

This command is valid in the general postprocessor.

### **Menu Paths**

**Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Path from array**

**Main Menu>General Postproc>Path Operations>Delete Path>All Paths**

**Main Menu>General Postproc>Path Operations>Delete Path>By Name**

**Main Menu>Preprocessor>Path Operations>Delete Path>All Paths**

**Main Menu>Preprocessor>Path Operations>Delete Path>By Name**

---

### **/PAGE**, *ILINE, ICHAR, BLINE, BCHAR*

**Defines the printout and screen page size.**

[POST1: Listing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *ILINE*

Number of lines (11 minimum) per "page" or screen. Defaults to 24. Applies to interactive non-GUI to the screen output only.

**ICCHAR**

Number of characters (41 to 132) per line before wraparound. Defaults to 80. Applies to interactive non-GUI to the screen output only.

**BLINE**

Number of lines (11 minimum) per page. Defaults to 56. Applies to batch mode [/BATCH], diverted [/OUTPUT], or interactive GUI [/MENU] output.

**BCHAR**

Number of characters (41 to 240 (system dependent)) per line before wraparound. Defaults to 132. Applies to batch mode [/BATCH], diverted [/OUTPUT], or interactive GUI [/MENU] output.

**Command Default**

As defined by the items above.

**Notes**

Defines the printout page size for batch runs and the screen page size for interactive runs. Applies to the POST1 **PRNSOL**, **PRESOL**, **PRETAB**, **PRRSOL**, and **PRPATH** commands. See the **/HEADER** command for additional controls (page ejects, headers, etc.) that affect the amount of printout. A blank (or out-of-range) value retains the previous setting. Issue **/PAGE,STAT** to display the current settings. Issue **/PAGE,DEFA** to reset the default specifications.

This command is valid in any processor.

**Menu Paths**

**This command cannot be accessed from a menu.**

---

**PAGET**, *PARRAY*, *POPT*

**Writes current path information into an array variable.**

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**PARRAY**

The name of the array parameter that the ANSYS program creates to store the path information. If the array parameter already exists, it will be replaced with the current path information.

**POPT**

Determines how data will be stored in the parameter specified with *PARRAY*:

**POINTS**

Store the path points, the nodes (if any), and coordinate system. (For information on defining paths and path points, see the descriptions of the **PATH** and **PPATH** commands.)

**TABLE**

Store the path data items. (See the **PDEF** command description for path data items.)

**LABEL**

Stores path data labels.

## Notes

Use the **PAGET** command together with the **PAPUT** command to store and retrieve path data in array variables for archiving purposes. When retrieving path information, restore the path points (POINTS option) first, then the path data (TABLE option), and then the path labels (LABEL option).

## Menu Paths

**Main Menu>General Postproc>Path Operations>Archive Path>Store>Path in array**

---

**PAPUT**, *PARRAY*, *POPT*

**Retrieves path information from an array variable.**

POST1: Path Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**PARRAY**

Name of the array variable containing the path information.

**POPT**

Specifies which path data to retrieve:

**POINTS**

Retrieve path point information (specified with the **PPATH** command and stored with the **PAGET,POINTS** command). The path data name will be assigned to the path points.

**TABLE**

Retrieve path data items (defined via the **PDEF** command and stored with the **PAGET,,TABLE** command).

**LABEL**

Retrieve path labels stored with the **PAGET,,LABEL** command.

## Notes

When retrieving path information, restore path points (POINTS option) first, then the path data (TABLE option), and then the path labels (LABEL option).

## Menu Paths

**Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Path from array**

---

**PARESU**, *Lab*, *Fname*, *Ext*, *--*

**Restores previously saved paths from a file.**

POST1: Path Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Lab**

Read operation:

**ALL**

Read all paths from the selected file (default).

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `PATH` if *Fname* is blank.

--

Unused field.

## Notes

This command removes all paths from virtual memory and then reads path data from a file written with the **PASAVE** command. All paths on the file will be restored. All paths currently in memory will be deleted.

## Menu Paths

**Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Paths from file**  
**Main Menu>Preprocessor>Path Operations>Retrieve>Paths from file**

---

## **PARTSEL**, *Type*, *PMIN*, *PMAX*, *PINC*

**Selects a subset of parts in an explicit dynamic analysis.**

[DATABASE: Selecting](#)

<> <> <> <> <> <> <> <> <> <> <> DY PP <> <> <>

***Type***

Label identifying type of select. Because **PARTSEL** is a command macro, the label must be enclosed in single quotes.

**'S'**

Select a new set (default).

**'R'**

Reselect a set from the current set.

**'A'**

Additionally select a set and extend the current set.

**'U'**

Unselect a set from the current set.

**'ALL'**

Select all parts.

**'NONE'**

Unselect all parts.

**'INVE'**

Invert the current selected set.

The following fields are used only with *Type* = 'S', 'R', 'A', or 'U':

***PMIN***

Minimum part number in the range to be selected.

***PMAX***

Maximum part number in the range to be selected (defaults to *PMIN*).

***PINC***

Part number increment in the range to be selected (default =1).

## Notes

**PARTSEL** invokes an ANSYS macro that selects parts in an explicit dynamic analysis. When **PARTSEL** is executed, an element component is automatically created for each existing part. For example, the elements that make up PART 1 are grouped into the element component \_PART1. Each time the **PARTSEL** command is executed, components for unselected parts will be unselected. To plot selected parts, choose **Utility Menu> Plot> Parts** in the GUI or issue the command **PARTSEL,'PLOT'**.

After selecting parts, if you change the selected set of nodes or elements and then plot parts, the nodes and elements associated with the previously selected parts (from the last **PARTSEL** command) will become the currently selected set.

---

### Note

A more efficient way to select and plot parts is to use the **ESEL** (with *ITEM* = PART) and **EPLLOT** commands. We recommend using **ESEL** instead of **PARTSEL** since **PARTSEL** will be phased out in a future release. Note that the menu path mentioned above for plotting parts does not work with the **ESEL** command; use **Utility Menu> Plot> Elements** instead.

In an explicit dynamic small restart analysis (**EDSTART,2**), **PARTSEL** can be used to unselect a part during the solution even if it is referenced in some way (such as in a contact definition). (Note that **ESEL** cannot be used for this purpose.) However, in a new analysis or a full restart analysis (**EDSTART,3**), all parts that are used in some type of definition must be selected at the time of solution.

This command is valid in any processor.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Utility Menu>Select>Select Parts**

---

**PARRES**, *Lab*, *Fname*, *Ext*, --**Reads parameters from a file.**[APDL: Parameters](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

## Argument Descriptions

### *Lab*

Read operation:

**NEW --**

Replace current parameter set with these parameters (default).

**CHANGE --**

Extend current parameter set with these parameters, replacing any that already exist.

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to PARM if *Fname* is blank.

--

Unused field.

## Notes

Reads parameters from a coded file. The parameter file may have been written with the **PARSAV** command. The parameters read may replace or change the current parameter set.

This command is valid in any processor.

## Menu Paths

**Utility Menu>Parameters>Restore Parameters**

---

**PARSAV**, *Lab*, *Fname*, *Ext*, --**Writes parameters to a file.**[APDL: Parameters](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

## Argument Descriptions

### *Lab*

Write operation:

**SCALAR --**

Write only scalar parameters (default).

**ALL --**

Write scalar and array parameters. Parameters may be numeric or alphanumeric.

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `PARM` if *Fname* is blank.

--

Unused field.

## Notes

Writes the current parameters to a coded file. Previous parameters on this file, if any, will be overwritten. The parameter file may be read with the **PARRES** command.

**PARSAV/PARRES** operations truncate some long decimal strings, and can cause differing values in your solution data when other operations are performed. A good practice is to limit the number of decimal places you will use before and after these operations.

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Save Parameters

---

**PASAVE**, *Lab*, *Fname*, *Ext*, --

**Saves selected paths to an external file.**

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Lab***

Write operation:

**S**

Saves only selected paths.

**ALL**

Saves all paths (default).

**Pname**

Saves the named path (from the **PSEL** command).

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `PATH` if *Fname* is blank.

--

Unused field.

## Notes

Saves the paths selected with the **PSEL** command to an external file (`Jobname.path` by default). Previous paths on this file, if any, will be overwritten. The path file may be read with the **PARESU** command.

This command is valid in `POST1`.

## Menu Paths

**Main Menu>General Postproc>Path Operations>Archive Path>Store>Paths in file**  
**Main Menu>Preprocessor>Path Operations>Store>Paths in file**

---

## **PATH**, *NAME*, *nPts*, *nSets*, *nDiv*

**Defines a path name and establishes parameters for the path.**

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NAME***

Name for this path (eight characters maximum. If *nPts* is blank, set the current path to the path with this name. If *nPts* is greater than zero, create a path of this name. If a path with this name already exists, replace it with a new path. If the *NAME* value is `STATUS`, display the status for path settings.

***nPts***

The number of points used to define this path. The minimum number is two, and the maximum is 1000. Default is 2.

***nSets***

The number of sets of data which you can map to this path. You must specify at least four: X, Y, Z, and S. Default is 30.

***nDiv***

The number of divisions between adjacent points. Default is 20. There is no maximum number of divisions.

## Notes

The **PATH** command is used to define parameters for establishing a path. The path geometry is created by the **PPATH** command. Multiple paths may be defined and named; however, only one path may be active



for data interpolation [**PDEF**] and data operations [**PCALC**, etc.]. Path geometry points and data are stored in memory while in POST1. If you leave POST1, the path information is erased. Path geometry and data may be saved in a file by archiving the data using the **PASAVE** command. Path information may be restored by retrieving the data using the **PARESU** command.

For overlapping nodes, the lowest numbered node is assigned to the path.

The number of divisions defined using *nDiv* does NOT affect the number of divisions used by **PLSECT** and **PRSECT**.

For information on displaying paths you have defined, see the *Basic Analysis Guide*.

## Menu Paths

**Main Menu>General Postproc>List Results>Path Items**  
**Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Path from array**  
**Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Paths from file**  
**Main Menu>General Postproc>Path Operations>Define Path>By Location**  
**Main Menu>General Postproc>Path Operations>Define Path>By Nodes**  
**Main Menu>General Postproc>Path Operations>Define Path>On Working Plane**  
**Main Menu>General Postproc>Path Operations>Define Path>Path Status>Defined Paths**  
**Main Menu>General Postproc>Path Operations>Delete Path>All Paths**  
**Main Menu>General Postproc>Path Operations>Delete Path>By Name**  
**Main Menu>General Postproc>Path Operations>Recall Path**  
**Main Menu>Preprocessor>Path Operations>Define Path>By Location**  
**Main Menu>Preprocessor>Path Operations>Define Path>By Nodes**  
**Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane**  
**Main Menu>Preprocessor>Path Operations>Define Path>Path Status>Defined Paths**  
**Main Menu>Preprocessor>Path Operations>Delete Path>All Paths**  
**Main Menu>Preprocessor>Path Operations>Delete Path>By Name**  
**Main Menu>Preprocessor>Path Operations>Recall Path**  
**Main Menu>Preprocessor>Path Operations>Retrieve>Paths from file**  
**Utility Menu>List>Status>General Postproc>Path Operations**

---

## PAUSE

Temporarily releases the current product license.

SESSION:Run Controls

MP ME ST PR PRN DS DSS FL EM EH DY PP VT EME MFS

## Notes

The **PAUSE** command temporarily releases (or pauses) the current product license so that another application can use it.

This application consumes a license as soon as you launch it, and retains that license until it is finished. If you launch the product interactively, the license is retained until you either close the application or issue a **PAUSE** command via the command line.

No other operation (other than **SAVE** or **/EXIT**) is possible in the current application while use of the product license is paused.

When the second application has finished and releases the license, issue an **UNPAUSE** command via the command line to restore use of the license to the current application.

For more information, see the *ANSYS, Inc. Licensing Guide*.

## Menu Paths

**This command cannot be accessed from a menu.**

**/PBC**, *Item*, *--*, *KEY*, *MIN*, *MAX*, *ABS*

**Shows boundary condition (BC) symbols and values on displays.**

GRAPHICS: Labeling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Item*

Label identifying the item:

#### **U**

Applied translational constraints (UX, UY, UZ).

#### **ROT**

Applied rotational constraints (ROTX, ROTY, ROTZ).

#### **TEMP**

Applied temperatures (TEMP, TBOT, TE2, TE3, . . . , TTOP).

#### **PRES**

Applied fluid pressures.

#### **V**

Applied flow velocities (VX, VY, VZ).

#### **SP0<sub>*n*</sub>**

Mass fraction of species *n*, where *n* = 1 to 6 (FLOTRAN). If a species is given a user-defined name [**MSSPEC**], use that name instead of SP0<sub>*n*</sub>.

#### **ENKE**

Turbulent kinetic energy (FLOTRAN).

#### **ENDS**

Turbulent energy dissipation (FLOTRAN).

#### **VOLT**

Applied voltages.

#### **MAG**

Applied scalar magnetic potentials.

#### **A**

Applied vector magnetic potentials.

#### **CHRG**

Applied electric charge.

**F or FORC**

Applied structural forces (FX, FY, FZ).

**M or MOME**

Applied structural moments (MX, MY, MZ).

**HEAT**

Applied heat flows (HEAT, HBOT, HE2, HE3, . . . , HTOP).

**FLOW**

Applied fluid flow.

**AMPS**

Applied current flow.

**FLUX**

Applied magnetic flux.

**CSG**

Applied magnetic current segments.

**MAST**

Master degrees of freedom.

**CP**

Coupled nodes.

**CE**

Nodes in constraint equations.

**NFOR**

POST1 nodal forces.

**NMOM**

POST1 nodal moments

**RFOR**

POST1 reaction forces.

**RMOM**

POST1 reaction moments (MX, MY, MZ).

**PATH**

Path geometry (undistorted) associated with the **PATH** command after a **PDEF** or **PVECT** command has been issued.

**ACEL**

Global acceleration (ACELX, ACELY, ACELZ vector).

**OMEG**

Global angular velocity (OMEGX, OMEGY, OMEGZ vector) and acceleration (DOME GX, DOME GY, DOME GZ vector).

**WELD**

Applied spotwelds (ANSYS LS-DYNA).

**ALL**

Represents all appropriate labels.

--

Unused field.

**KEY**

Symbol key:

**0**

Do not show symbol.

**1**

Show symbol.

**2**

Plot value next to symbol.

**MIN**

Minimum value in a range of values plotted on screen.

**MAX**

Maximum value in a range of values plotted on screen.

**ABS**

Absolute number. If *KEY* = 2 and *ABS* = 0, a number falling between the *MIN* and *MAX* is displayed. If *ABS* is not specified, it defaults to 0. If *KEY* = 2 and *ABS* = 1, an absolute value falling between the *MIN* and *MAX* is displayed. *ABS* = 1 lets you eliminate the display of numbers whose absolute values are less than a desired tolerance. For example, if *ABS* = 1, *MIN* = 10 and *MAX* = 1e8, values such as .83646 and -5.59737 are not displayed.

## Command Default

No symbols displayed.

## Notes

The **/PBC** command adds degree of freedom constraint, force load, and other symbols to displays.

Symbols are applied to the selected nodes only. All arrow and arrowhead symbols are oriented in the nodal coordinate system and lie in two perpendicular planes. Force arrows are scaled proportional to their magnitude. (If *KEY* = 1, use **/VSCALE** to change arrow length.) For scalar quantities, the specific component direction (i.e., x, y, or z) of the symbol has no meaning, but the positive or negative sense (e.g., positive or negative x) represents a positive or negative scalar value, respectively.

The effects of the **/PBC** command are not cumulative (that is, the command does not modify an existing setting from a previously issued **/PBC** command). If you issue multiple **/PBC** commands during an analysis, only the setting specified by the most recent **/PBC** command applies.

Use **/PSTATUS** or **/PBC,STAT** to display settings. Use **/PBC,DEFA** to reset all specifications back to default. See the **/PSF** and **/PBF** commands for other display symbols.

In a *cyclic symmetry analysis*, the **/PBC** command is deactivated when *cyclic expansion* is active (**/CYCEXPAND,,ON**). To view boundary conditions on the *basic sector*, deactivate cyclic expansion (**/CYCEXPAND,,OFF**) and issue this command: **/PBC,ALL,,1**

Issuing the command **/PBC,PATH,,1** displays all defined paths.

The **/PBC** command is valid in any processor.

## Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane

Main Menu>General Postproc>Path Operations>Plot Paths

Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

Main Menu>Preprocessor>Path Operations>Plot Paths

Utility Menu>PlotCtrls>Symbols

**/PBF**, *Item*, --, *KEY*

Shows magnitude of body force loads on displays.

GRAPHICS:Labeling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Item*

Label identifying the item:

#### **TEMP**

Applied temperatures.

#### **FLUE**

Applied fluences.

#### **HGEN**

Applied heat generation rates.

#### **JS**

Applied current density magnitude.

#### **JSX**

X-component of current density.

#### **JSY**

Y-component of current density.

#### **JSZ**

Z-component of current density.

#### **PHASE**

Phase angle of applied load.

#### **MVDI**

Applied magnetic virtual displacements flag.

#### **CHRGD**

Applied electric charge density.

#### **VLTG**

Applied voltage drop.

#### **FORC**

Applied force density (FLOTRAN only).

--

Unused field.

### **KEY**

Symbol key:

- 0**  
Do not show body force load contours.
- 1**  
Show body force load contours.
- 2**  
Show current density as a vector (not a contour).

### **Command Default**

No body force load contours displayed.

### **Notes**

Shows body force loads as contours on displays for the selected elements.

The effects of the **/PBF** command are not cumulative (that is, the command does not modify an existing setting from a previously issued **/PBF** command). If you issue multiple **/PBF** commands during an analysis, only the setting specified by the most recent **/PBF** command applies.

Use **/PSTATUS** or **/PBF,STAT** to display settings. Use **/PBF,DEFA** to reset all specifications back to default. See also the **/PSF** and **/PBC** command for other display contours.

Portions of this command are not supported by PowerGraphics [**/GRAPHICS,POWER**].

This command is valid in any processor.

### **Menu Paths**

**Utility Menu>PlotCtrls>Symbols**

---

**PCALC**, *Oper*, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*, *CONST*

**Forms additional labeled path items by operating on existing path items.**

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ***Oper***

Type of operation to be performed. See "[Notes](#)" (p. 1203) below for specific descriptions of each operation:

#### **ADD**

Adds two existing path items.

#### **MULT**

Multiplies two existing path items.

#### **DIV**

Divides two existing path items (a divide by zero results in a value of zero).

#### **EXP**

Exponentiates and adds existing path items.

#### **DERI**

Finds a derivative.

**INTG**

Finds an integral.

**SIN**

Sine.

**COS**

Cosine.

**ASIN**

Arcsine.

**ACOS**

Arccosine.

**LOG**

Natural log.

**LabR**

Label assigned to the resulting path item.

**Lab1**

First labeled path item in operation.

**Lab2**

Second labeled path item in operation. *Lab2* must not be blank for the MULT, DIV, DERI, and INTG operations.

**FACT1**

Factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

**FACT2**

Factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

**CONST**

Constant value (defaults to 0.0).

**Notes**

If *Oper* = ADD, the command format is:

**PCALC,ADD,LabR,Lab1,Lab2,FACT1,FACT2,CONST**

This operation adds two existing path items according to the operation:

$$LabR = (FACT1 \times Lab1) + (FACT2 \times Lab2) + CONST$$

It may be used to scale the results for a single path item.

If *Oper* = MULT, the command format is:

**PCALC,MULT,LabR,Lab1,Lab2,FACT1**

*Lab2* must not be blank. This operation multiplies two existing path items according to the operation:

$$LabR = Lab1 \times Lab2 \times FACT1$$

If *Oper* = DIV, the command format is:

**PCALC**,DIV,*LabR*,*Lab1*,*Lab2*,*FACT1*

*Lab2* must not be blank. This operation divides two existing path items according to the operation:

$$LabR = (Lab1/Lab2) \times FACT1$$

If *Oper* = EXP, the command format is:

**PCALC**,EXP,*LabR*,*Lab1*,*Lab2*,*FACT1*,*FACT2*

This operation exponentiates and adds existing path items according to the operation:

$$LabR = (|Lab1|^{FACT1}) + (|Lab2|^{FACT2})$$

If *Oper* = DERI, the command format is:

**PCALC**,DERI,*LabR*,*Lab1*,*Lab2*,*FACT1*

*Lab2* must not be blank. This operation finds a derivative according to the operation:

$$LabR = FACT1 \times d(Lab1)/d(Lab2)$$

If *Oper* = INTG, the command format is:

**PCALC**,INTG,*LabR*,*Lab1*,*Lab2*,*FACT1*

*Lab2* must not be blank. This operation finds an integral according to the operation:

$$LabR = FACT1 \times \int_s Lab1 d(Lab2)$$

Use *S* for *Lab2* to integrate *Lab1* with respect to the path length. *S*, the distance along the path, is automatically calculated by the program when a path item is created with the **PDEF** command.

If *Oper* = SIN, COS, ASIN, ACOS, or LOG, the command format is:

**PCALC**,*Oper*,*LabR*,*Lab1*,,*FACT1*,*CONST*

where the function (SIN, COS, ASIN, ACOS or LOG) is substituted for *Oper* and *Lab2* is blank.

The operation finds the resulting path item according to one of the following formulas:

$$LabR = FACT2 \times \sin(FACT1 \times Lab1) + CONST$$

$$LabR = FACT2 \times \cos(FACT1 \times Lab1) + CONST$$

$$LabR = FACT2 \times \sin^{-1}(FACT1 \times Lab1) + CONST$$

$$LabR = FACT2 \times \cos^{-1}(FACT1 \times Lab1) + CONST$$

$$LabR = FACT2 \times \log(FACT1 \times Lab1) + CONST$$



## Menu Paths

**Main Menu>General Postproc>Path Operations>Add**  
**Main Menu>General Postproc>Path Operations>ArcCosine**  
**Main Menu>General Postproc>Path Operations>ArcSine**  
**Main Menu>General Postproc>Path Operations>Cosine**  
**Main Menu>General Postproc>Path Operations>Differentiate**  
**Main Menu>General Postproc>Path Operations>Divide**  
**Main Menu>General Postproc>Path Operations>Exponentiate**  
**Main Menu>General Postproc>Path Operations>Integrate**  
**Main Menu>General Postproc>Path Operations>Multiply**  
**Main Menu>General Postproc>Path Operations>Natural Log**  
**Main Menu>General Postproc>Path Operations>Sine**

**PCGOPT**, *Lev\_Diff*, *--*, *ReduceIO*, *StrmCk*, *Wrtfull*, *Memory*

**Controls PCG solver options.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN DS DSS <> EM <> <> PP <> EME MFS

### *Lev\_Diff*

Indicates the level of difficulty of the analysis. Valid settings are AUTO or 0 (default), 1, 2, 3, 4, or 5. This option applies to both the PCG solver when used in static and full transient analyses and to the PCG Lanczos method in modal analyses. Use AUTO to let ANSYS automatically choose the proper level of difficulty for the model. Lower values (1 or 2) generally provide the best performance for well-conditioned problems. Values of 3 or 4 generally provide the best performance for ill-conditioned problems; however, higher values may increase the solution time for well-conditioned problems. Higher level-of-difficulty values typically require more memory. Using the highest value of 5 essentially performs a factorization of the global matrix (similar to the sparse solver) and may require a very large amount of memory. If necessary, use *Memory* to reduce the memory usage when using *Lev\_Diff* = 5. *Lev\_Diff* = 5 is generally recommended for small- to medium-sized problems when using the PCG Lanczos mode extraction method.

For example, models containing elongated elements (i.e., elements with high aspect ratios) and models containing contact elements can lead to ill-conditioned problems. To determine if your problem is ill-conditioned, view the *Jobname.PCS* file to see the number of PCG iterations needed to reach a converged solution. Generally, static or full transient solutions that require more than 1500 iterations are considered to be ill-conditioned for the PCG solver.

--

Unused field.

### *ReduceIO*

Controls whether the PCG solver will attempt to reduce I/O performed during equation solution:

#### **AUTO**

Automatically chooses whether to reduce I/O or not (default).

#### **YES**

Reduces I/O performed during equation solution in order to reduce total solver time.

#### **NO**

Does NOT reduce I/O performed during equation solution.

This option applies to both the PCG solver when used in static and full transient analyses and to the PCG Lanczos method in modal analyses.

**StrmCk**

Controls whether or not a Sturm sequence check is performed:

**OFF**

Does NOT perform Sturm sequence check (default).

**ON**

Performs Sturm sequence check

This option applies only when using the PCG Lanczos method in modal analyses. When using this option, a factorization must be performed and will require a very large amount of memory for extra computations. This option is generally recommended for small- to medium-sized problems. If the Sturm sequence check takes a large amount of computing time, use the `Jobname.ABT` file to abort the Sturm check, or press the **STOP** button if in interactive mode.

**Wrtfull**

Controls whether or not the `.FULL` file is written.

**ON**

Write `.FULL` file (default)

**OFF**

Do not write `.FULL` file.

This option applies only when using the PCG Lanczos method in modal analyses because the `.FULL` file is never written when using the PCG solver in static or full transient analyses. If using **MSAVE,ON** and conditions for the **MSAVE** command are met, a complete `.FULL` file is never written, regardless of this option.

This option is useful in Distributed ANSYS because assembling the global stiffness and mass matrices on the master machine before writing the `.FULL` file can take a considerable amount of memory. By setting `Wrtfull = OFF`, this assembly process is skipped on the master machine, decreasing the amount of memory required to compute the modes and mode shapes. `Wrtfull = OFF` does not affect the results for the modes and mode shapes. However, without a `.FULL` file, the participation factor table computations do not occur.

To generate the `.FULL` file, such as for a harmonic, transient mode superposition, or spectrum analysis, rerun the modal analysis with `Wrtfull = ON`, or use the **WRFULL** command.

**Memory**

Controls whether to run using in-core or out-of-core mode when using `Lev_Diff = 5`.

**AUTO**

Automatically chooses which mode to use. (default).

**INCORE**

Run using in-core mode.

**OOC**

Run using out-of-core mode.

This option applies to both the PCG solver when used in static and full transient analyses and to the PCG Lanczos method in modal analyses. However, this option only applies when using `Lev_Diff = 5` and does not apply to Distributed ANSYS. Forcing the **INCORE** option will require a large amount of memory and is only recommended for small-to-medium sized problems. Using the **OOC** option will make

the PCG solver use the least amount of memory (relative only to  $Lev\_Diff = 5$ ) at the cost of additional I/O, which will likely decrease performance as compared to an in-core run.

## Notes

*ReduceIO* works independently of the **MSAVE** command in the PCG solver. Setting *ReduceIO* to YES can significantly increase the memory usage in the PCG solver.

To minimize the memory used by the PCG solver with respect to the  $Lev\_Diff$  option only, set  $Lev\_Diff = 1$  if you do not have sufficient memory to run the PCG solver with  $Lev\_Diff = \text{AUTO}$ .

The **MSAVE,ON** command is not valid when using  $Lev\_Diff = 5$ . In this case, the  $Lev\_Diff$  value will automatically be reset to 2. The **MSAVE,ON** command is also not valid with the *StrmCk* option. In this case, *StrmCk* will be set to OFF.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>Analysis Options**

---

## PCIRC, RAD1, RAD2, THETA1, THETA2

**Creates a circular area centered about the working plane origin.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **RAD1, RAD2**

Inner and outer radii (either order) of the circle. A value of either zero or blank for either *RAD1* or *RAD2*, or the same value for both *RAD1* and *RAD2*, defines a solid circle.

### **THETA1, THETA2**

Starting and ending angles (either order) of the circular area. Used for creating a circular sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0.0° and the ending angle defaults to 360.0°. See the [Modeling and Meshing Guide](#) for an illustration.

## Notes

Defines a solid circular area or circular sector centered about the working plane origin. For a solid circle of 360°, the area will be defined with four keypoints and four lines. See the **CYL4** and **CYL5** commands for alternate ways to create circles.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Circle>By Dimensions**

---

**/PCIRCLE**, *XCENTR*, *YCENTR*, *XLRAD***Creates an annotation circle (GUI).**[GRAPHICS: Annotation](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***XCENTR***

Circle X center location (-1.0 &lt; X &lt; 2.0).

***YCENTR***

Circle Y center location (-1.0 &lt; Y &lt; 1.0).

***XLRAD***

Circle radius length.

**Notes**

Creates an annotation circle to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All circles are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** and the **/PSPEC** command to set the attributes of the circle.

This command is valid in any processor.

**Menu Paths****Utility Menu>PlotCtrls>Annotation>Create Annotation**

---

**/PCOPY**, *KEY***Automatically generates hard copies for HP UNIX work stations.**[GRAPHICS: Set Up](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***KEY***

Copy key:

**0**

No specification setting for automatic hard copy of display.

**1**

Set specification for automatic hard copy after each display.

**NOW**(Action) Produce hard copy of current display (*KEY* is not reset to 1).**Command Default**

No automatic hard copy of display.

## Notes

Sets automatic hard copy specification. This command is available only on HP work stations, and only during interactive runs with the **/SHOW** specification active (for terminals with hard copy capability).

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**PCROSS**, *LabXR*, *LabYR*, *LabZR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

**Calculates the cross product of two path vectors along the current path.**

POST1: Path Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***LabXR***

Label assigned to X-component of resultant vector.

***LabYR***

Label assigned to Y-component of resultant vector.

***LabZR***

Label assigned to Z-component of resultant vector.

***LabX1***

X-component of first vector label (labeled path item).

***LabY1***

Y-component of first vector label.

***LabZ1***

Z-component of first vector label.

***LabX2***

X-component of second vector label (labeled path item).

***LabY2***

Y-component of second vector label.

***LabZ2***

Z-component of second vector label.

## Menu Paths

**Main Menu>General Postproc>Path Operations>Cross Product**

---

**PDANL**, *Fname*, *Ext*, --

**Defines the analysis file to be used for probabilistic looping.**

PROBABILISTIC: Deterministic Model

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

#### ***Ext***

Filename extension (8 character maximum).

--

Unused field.

### **Command Default**

If interactive, no default; if batch, use the batch input stream (`File.BAT`).

### **Notes**

The analysis file containing the deterministic, parameterized model must be specified if the probabilistic analysis is performed interactively. The file must exist at the time the **PDANL** command is issued. In this file, where **/PREP7** and **/PDS** occur, they must be the first nonblank characters on the line (do not use the \$ delimiter on any **/PREP7** and **/PDS** command lines).

By default the analysis files specified with **PDANL** are executed from the top. All definitions of random input variables (for example, APDL parameters defined as random input variables with the **PDVAR** command, using **\*SET** or *Pname* = ...) are ignored in the analysis file. The PDS takes control of setting the values of random input variable values for each loop.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### **Menu Paths**

**Main Menu>Prob Design>Analysis File>Assign**

**PDCDF**, *Rlab*, *Name*, *Type*, *CONF*, *NMAX*

**Plots the cumulative distribution function.**

PROBABILISTIC: Postprocessing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ***Rlab***

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses).

**Name**

Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command.

**Type**

Type of cumulative distribution curve to be displayed.

**EMP**

Show an empirical cumulative distribution curve.

**GAUS**

Show a cumulative distribution curve in a normal distribution plot. A random variable based on a normal or Gaussian distribution appears as a straight line in this type of plot.

**LOGN**

Show a cumulative distribution curve in a log-normal plot. A random variable based on a log-normal distribution appears as a straight line in this type of plot.

**WEIB**

Show a cumulative distribution curve in a Weibull distribution plot. A random variable based on a Weibull distribution appears as a straight line in this type of plot.

**CONF**

Confidence level. The confidence level is used to plot confidence bounds around the cumulative distribution function. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%). Plotting of the confidence bound is suppressed for  $CONF \leq 0.5$ . This parameter is ignored for the postprocessing of response surface methods results.

**NMAX**

Maximum number of points to be plotted for the distribution curve. This must be a positive number and it defaults to 100. If the sample size is less than *NMAX*, all sample data is represented in the plot. If the sample size is larger than *NMAX*, the probabilistic design system classifies the sample into *NMAX* classes of appropriate size.

**Command Default**

*Rlab* as described above, *Type* = EMP, *CONF* = 0.95, *NMAX* = 100.

**Notes**

Plots the cumulative distribution function.

The **PDCDF** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

**Menu Paths**

**Main Menu>Prob Design>Prob Results>Statistics>CumulativeDF**

**PDCFLD**, *ParR*, *Entity*, *Ctype*, *CLENGTH*

**Calculates a correlation field and stores it into an ANSYS array.**

PROBABILISTIC: Preprocessing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***ParR***

Parameter name. *ParR* is a one-dimensional array with the dimension  $N * (N - 1)/2$ , where  $N$  is either the number of the selected nodes or the number of the selected elements (depending on the *Entity* field). The **PDCFLD** command automatically sets *ParR* as a one-dimensional array, (so you do not have to use the **\*DIM** command). If you use the **PDCFLD** command twice with the ANSYS parameter *ParR*, then the values stored in the array are automatically overwritten. If the number of selected FE entities is different from the previous **PDCFLD** command, then the array *ParR* is re-dimensioned automatically.

***Entity***

Specifies which FE entity the calculation of the correlation field is based on. This field must not be blank.

**NODE**

Calculate the correlation coefficients based on the distance between the selected nodes.

**ELEM**

Calculate the correlation coefficients based on the distance between the centroids of the selected elements.

***Ctype***

Specifies the equation used to calculate the correlation coefficients as a function of the nodal or element centroid distances. This field must not be blank.

**NONE**

The random field is not correlated. This means the correlation coefficients are determined according to

$$\rho_{ij} = 1 \text{ for } i = j$$

$$\rho_{ij} = 0 \text{ for } i \neq j$$

Here,  $\rho_{ij}$  is the correlation coefficient between the  $i$ -th and  $j$ -th selected FE entity (node or element centroid).

**LEXP**

Calculate the correlation coefficient according to a linear-exponential decay function.

$$\rho_{ij} = \exp\left(-\frac{D(\{x_i\}, \{x_j\})}{C_L}\right)$$

Here,  $D(\{x_i\}, \{x_j\})$  is the "domain distance" between  $\{x_i\}$ ,  $\{x_j\}$ , and  $\{x_i\}$  and  $\{x_j\}$  are the coordinate vectors of the  $i$ -th and  $j$ -th selected FE entity (node or element centroid), and  $C_L$  is the correlation length of the random field as specified in the *CLENGTH* field.

**QEXP**

The correlation coefficient is calculated according to a quadratic- exponential decay function.



$$\rho_{ij} = \exp \left( - \left( \frac{D(\{x_i\}, \{x_j\})}{C_L} \right)^2 \right)$$

Here,  $D(\{x_i\}, \{x_j\})$  is the “domain distance” between  $\{x_i\}$ ,  $\{x_j\}$ , and  $\{x_i\}$  and  $\{x_j\}$  are the coordinate vectors of the  $i$ -th and  $j$ -th selected FE entity (node or element centroid), and  $C_L$  is the correlation length of the random field as specified in the *CLENGTH* field.

### DIST

Calculate only  $D(\{x_i\}, \{x_j\})$ .  $D(\{x_i\}, \{x_j\})$  is the “domain distance” between  $\{x_i\}$ ,  $\{x_j\}$ , and  $\{x_i\}$  and  $\{x_j\}$  are the coordinate vectors of the  $i$ -th and  $j$ -th selected FE entity (node or element centroid). The *CLENGTH* argument is ignored for this option.

### CLENGTH

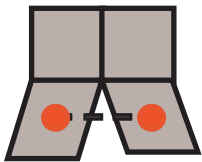
Correlation length of the correlation field. The correlation length is a characteristic length that influences how strongly two elements of a random field are correlated with each other. The larger the value of *CLENGTH*, the stronger the correlation between the random field elements. *CLENGTH* is required for *Ctype* = LEXP and *Ctype* = QEXP; it must be a nonzero, positive number.

## Notes

Calculates a correlation field for a probabilistic analysis involving a random field. Random fields are random effects with a spatial distribution; the value of a random field not only varies from simulation to simulation at any given location, but also from location to location. The correlation field describes the correlation coefficient between two different spatial locations. Random fields can be either based on element properties (typically material) or nodal properties (typically surface shape defined by nodal coordinates). Hence, random fields are either associated with the selected nodes or the selected elements. If a random field is associated with elements, then the correlation coefficients of the random field are calculated based on the distance of the element centroids.

For more information, see [Probabilistic Design](#) in the *Advanced Analysis Techniques Guide*.

Note that for correlation fields, the “domain distance”  $D(\{x_i\}, \{x_j\})$  is not the spatial distance  $|\{x_i\} - \{x_j\}|$ , but the length of a path between  $\{x_i\}$  and  $\{x_j\}$  that always remains inside the finite element domain. However, exceptions are possible in extreme meshing cases. For elements that share at least one node, the **PDCFLD** evaluates the distance by directly connecting the element centroids with a straight line. If these neighboring elements form a sharp inward corner then it is possible that the “domain distance” path lies partly outside the finite element domain, as illustrated below.



After the correlation coefficients have been calculated and stored in the ANSYS parameter *ParR*, then use the **PDCORR** command to define the correlations between the elements of the random field.

## Menu Paths

**Main Menu>Prob Design>Prob Definitns>Correl Field**

**PDCLR**, *Type*

**Clears the probabilistic design database.**

[PROBABILISTIC: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Specifies the part of the probabilistic database to be cleared.

**ALL**

Clear the entire probabilistic database. Both the preprocessing and postprocessing parts are cleared.

**POST**

Clear only the postprocessing part of the probabilistic database. This is necessary if you want to modify the deterministic model (the analysis file) or probabilistic model (random variables, correlations between random variables, or the random output parameter) after a probabilistic analysis has already been performed.

**Notes**

Clears the probabilistic design system (PDS) database. The settings are reset to their default values and the memory is cleared. Remember that the result files containing the results of the probabilistic loops are never deleted unless you intentionally delete them. We recommend that you use this command before switching to a new probabilistic analysis using a different probabilistic model (a different analysis loop file or deterministic model), or changing random input variables, their correlations, or random output parameters. Clearing the probabilistic database is not necessary if the probabilistic model remains the same and will be analyzed with a different probabilistic method or different method options. See the [PDEXE](#) command for restrictions. Before issuing the **PDCLR** command, you should save the probabilistic database using the [PDSAVE](#) command.

**Menu Paths**

**Main Menu>Prob Design>Prob Database>Clear & Reset**

**PDCMAT**, *Rlab, Matrix, Name1, Name2, Corr, SLEVEL, Popt*

**Prints the correlation coefficient matrix.**

[PROBABILISTIC: Postprocessing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Rlab**

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a [PDEXE](#) command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an [RSFIT](#) command (for Response Surface Analyses).

**Matrix**

Keyword for the type of correlation coefficient matrix.

**IO**

Matrix of correlation coefficients between random input variables and output parameters.

**II**

Matrix of correlation coefficients between random input variables and other random input variables

**OO**

Matrix of correlation coefficients between random output parameters and other random output parameters.

**S**

Correlation coefficient between a single random parameter (input or output) and another random parameter (input or output). The probabilistic design parameters must be specified in *Name1* and *Name2* for this option.

**Name1, Name2**

Parameter names. The parameters must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command. These parameters are used for *Matrix* = S only and are ignored for the other *Matrix* keywords.

**Corr**

Keyword for the type of correlation coefficients to be used for the output.

**RANK**

Spearman rank-order correlation coefficient (default).

**LIN**

Pearson linear correlation coefficient.

**SLEVEL**

Significance level. The value for the significance level must be between 0.0 and 1.0. The default value is 0.025 (2.5%).

**Popt**

Specifies whether the probabilities should be printed with the correlation coefficients.

**0**

Print only the correlation coefficients.

**1**

Print both the correlation coefficients and the probabilities (default).

**Command Default**

*Rlab* as described above, *Matrix* = IO, *Corr* = RANK, *SLEVEL* = 0.025, *Popt* = 1

**Notes**

Prints the correlation coefficient matrix.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

For all correlation coefficients the probabilistic design tool evaluates the probability that the correlation coefficient can be neglected. The evaluation of this probability is based on statistical test theory. The larger this probability is the likelier it is that the correlation coefficient does not really reflect an observable statistical interdependence between the parameters involved. If this probability exceeds the significance level as specified by the *SLEVEL* parameter, the correlation coefficient should be regarded as negligible or insignificant. The higher the significance level *SLEVEL*, the more correlation coefficients are considered significant.

Using the *Popt* parameter you can also get a list of the probabilities and review them as to how far they exceed the significance level or how far they stay below it.

The **PDCMAT** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

## Menu Paths

Main Menu>Prob Design>Prob Results>Trends>Correl Matrix

---

**PDCORR**, *Name1*, *Name2*, *CORR*

Specifies the correlation between two random input variables.

[PROBABILISTIC: Preprocessing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Name1*

Parameter name. The parameter must have been previously defined as a random input variable with the **PDVAR** command.

### *Name2*

Parameter name. The parameter must have been previously defined as a random input variable with the **PDVAR** command. Must be different from *Name1*.

### *CORR*

Specification of the correlation:

#### Value

Sets the correlation coefficient between *Name1* and *Name2* to the specified value. If this correlation coefficient was already defined it will be changed to this new value. The correlation coefficient must be between -1.0 and +1.0.

#### DEL

Delete the previously defined correlation between *Name1* and *Name2*.

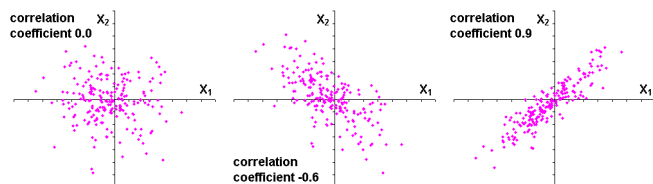
## Command Default

No correlations defined between random input variables.

## Notes

Specifies the correlations between two random input variables. The PDS tool generates correlated samples if correlations exist. This applies to both probabilistic methods (Monte Carlo Simulation and Response Surface Methods).

If there are correlated random input variables, the sampled locations of the random input variables reflect the correlation as illustrated below for the case of two standard normal distributed variables  $X_1$  and  $X_2$ . The illustrations show no correlation (correlation coefficient 0.0), a relatively moderate negative correlation (correlation coefficient -0.6), and a relative strong positive correlation (correlation coefficient 0.9).



## Menu Paths

Main Menu>Prob Design>Prob Definitns>Correlation

**PDDMCS**, *NSIM*, --, *Astop*, *ACCMEAN*, *ACCSTDEV*, *CHECK*, *Seed*

Specifies options for Monte Carlo Simulations using direct sampling.

PROBABILISTIC:Methods

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **NSIM**

Number of simulation loops of the analysis.

--

Unused field.

### **Astop**

Autostop option label.

#### **AUTO**

Enable Autostop. When Autostop is used, the PDS feature continues the simulation loops until the convergence criteria for the mean value and the standard deviation have been met or until the number of simulations *NSIM* are complete, whichever comes first. The convergence criteria (mean value and standard deviations of all random output parameters) are specified by the *ACCMEAN* and *ACCSTDEV* parameters. The criteria are met if the mean value *and* the standard deviations converge within the accuracy specified in the *ACCMEAN* and *ACCSTDEV* options. The convergence check is done every *i*-th loop, where *i* is specified in the *CHECK* parameter.

#### **ALL**

Disable Autostop option. All Monte Carlo Simulations as specified by *NSIM* are performed (default).

### **ACCMEAN**

Accuracy of the mean values of all random output parameters that must be met for the Autostop option. Default is 0.01 (1%). *ACCMEAN* is ignored for *Astop* = ALL. The convergence of the mean values is met if for all random output parameters *y* the equation holds:

$$\frac{|\bar{y}(i) - \bar{y}(i - \text{CHECK})|}{\bar{y}(i)} \leq \text{ACCMEAN} \quad \text{with } i = 2 \cdot \text{CHECK}, 3 \cdot \text{CHECK}, \dots$$

where the value of *CHECK* is specified in the *CHECK* option.

### **ACCSTDEV**

Accuracy of the standard deviations of all random output parameters that must be met for the Autostop option. The default is 0.02 (2%). *ACCSTDEV* is ignored for *Astop* = ALL. The convergence for the standard deviations is met if for all random output parameters *y* the equation holds:

$$\frac{|\sigma_y(i) - \sigma_y(i - \text{CHECK})|}{\sigma_y(i)} \leq \text{ACCSTDEV} \quad \text{with } i = 2 \cdot \text{CHECK}, 3 \cdot \text{CHECK}, \dots$$

where the value of *CHECK* is specified in the *CHECK* option.

### **CHECK**

Sets how often conditions for convergence are checked for Autostop. The PDS checks if the convergence criteria are met every *i*-th loop, where *i* is given by the *CHECK* parameter. The default value is 10. It is not recommended to use *CHECK* = 1, because it could cause Autostop to terminate the simulations prematurely. The mean values and standard deviation might not show large differences between consecutive simulation loops but might still have a visible "global" trend if viewed over several simulations. This behavior indicates that convergence has not really been achieved. If you set *CHECK* = 1 (or another small number), then Autostop is not able to detect such a global trend. *CHECK* is ignored for *Astop* = ALL.

### **Seed**

Seed value label. Random number generators require a seed value that is used to calculate the next random number. After each random number generation finishes, the seed value is updated and is used again to calculate the next random number. ANSYS initializes the seed value with the system time when the ANSYS session started.

### **CONT**

Continues updating using the derived seed value (default).

### **TIME**

Initializes the seed value with the system time. You can use this if you want the seed value set to a specific value for one analysis and then you want to continue with a "random" seed in the next analysis. It is not recommended to "randomize" the seed value with the *Seed* = TIME option for multiple analyses. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

### **INIT**

Initializes the seed value using 123457 (a typical recommendation). This option leads to identical random numbers for all random input variables when the exact analysis will be repeated, making it useful for benchmarking and validation purposes (where identical random numbers are desired). If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

### **Value**

Uses the specified (positive) value for the initialization of the seed value. This option has the same effect as *Seed* = INIT, except you can choose an arbitrary (positive) number for the initialization. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

## **Command Default**

*NSIM* = 30, *Astop* = AUTO, *ACCMEAN* = 0.01, *ACCSTDEV* = 0.02, *CHECK* = 10, *Seed* = CONT.

## **Notes**

Defines the number of simulations, the specification of the Autostop option, how frequently the Autostop option is checked and the seed value for random number generation. If the Autostop option is enabled, then the number of simulations given for *NSIM* can be regarded as the maximum number of simulations

to be performed. The Autostop option may terminate the simulations before *NSIM* simulations are done if the convergence criteria are met.

## Menu Paths

Main Menu>Prob Design>Prob Method>Monte Carlo Sims

**PDDOEL**, *Name*, *Method*, *Vtype*, *Lopt*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*

**Defines design of experiment levels for an individual random input variable.**

PROBABILISTIC: Preprocessing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Name*

Parameter name. The parameter name must have been previously defined as a random input variable using the **PDVAR** command.

### *Method*

Specifies the response surface method for which the levels of the design of experiment are to be defined. This field must not be left blank.

#### **CCD**

Use the Central Composite Design method. The design experiment levels of a central composite design are defined in the fields *VAL1* to *VAL5*.

#### **BBM**

Use the Box-Behnken Matrix method. The design experiment levels of a Box-Behnken Matrix design are defined in the fields *VAL1* to *VAL3*. The fields *VAL4* and *VAL5* are ignored

### *Vtype*

Specifies the type of the values of the design of experiment levels.

#### **PROB**

The design of experiment levels are specified in terms of probabilities. This is the default.

#### **PHYS**

The design of experiment levels are specified in terms of physical values.

### *Lopt*

Specifies the type of the design of experiment levels, indicating if they are defined by lower and upper bound only (default) or all specified by the user.

#### **BND**

You specify the lower and upper bounds for the design of experiment levels. The values for intermediate levels are calculated automatically at run time (default). The lower and upper levels of the design of experiment itself can be specified either in terms of probabilities or in terms of physical values, depending on the *Vtype* field.

For *Lopt* = BND and *Method* = CCD only the entries *VAL1* and *VAL5* are processed and they represent the lower and upper bound values of a central composite design. The intermediate levels *VAL2* to *VAL4* are evaluated automatically. For *Lopt* = BND and *Method* = BBM only the entries *VAL1* and *VAL3* are processed and they represent the lower and upper bound values of a Box-Behnken Matrix design respectively. The intermediate level *VAL2* is evaluated automatically.

NOTE: The intermediate levels between the lower and upper bounds are calculated so they are at equal intervals along the curve (interpolated linearly in the physical space whether the curve is symmetrical or not) regardless of whether the lower and upper bounds are specified as probabilities (*vtype* = PROB) or as physical values (*vtype* = PHYS).

### ALL

You explicitly specify all necessary design of experiment levels. The design of experiment levels can be in terms of probabilities or in terms of physical values, depending on the *vtype* field.

### VAL1, VAL2, VAL3, . . . , VAL5

Values for the levels of the design of experiment for the random input variable *Name*. Must be specified in ascending order. All probabilities must be between 0.0 and 1.0.

For response surface methods based on a Central Composite Design (*Method* = CCD) *VAL1* through *VAL5* correspond to the five design of experiment levels of this method.

For response surface methods based on a Box-Behnken Matrix design (*Method* = BBM) *VAL1* through *VAL3* correspond to the three design of experiment levels of this method. The entries for *VAL4* and *VAL5* are ignored for *Method* = BBM.

## Command Default

For *Meth* = CCD, *VAL1* = 0.005, *VAL2* = 0.05, *VAL3* = 0.5, *VAL4* = 0.95, *VAL5* = 0.995.

For *Meth* = BBM, *VAL1* = 0.01, *VAL2* = 0.5, *VAL3* = 0.99.

## Notes

If *vtype* = PHYS, you must enter values for *VAL1* through *VAL3* or *VAL5* (depending on the *Method* and *Lopt* option you choose). If *vtype* = PROB and you do not enter values, they default to the values shown below.

For *Method* = CCD:

	<i>Lopt</i> = BND	<i>Lopt</i> = ALL
<i>vtype</i> = PHYS	<i>VAL1</i> and <i>VAL5</i> must not be blank.	<i>VAL1</i> through <i>VAL5</i> must not be blank.
<i>vtype</i> = PROB	<i>VAL1</i> = 0.005, <i>VAL5</i> = 0.995	<i>VAL1</i> = 0.005, <i>VAL2</i> = 0.05, <i>VAL3</i> = 0.50, <i>VAL4</i> = 0.95, <i>VAL5</i> = 0.995

For *Method* = BBM:

	<i>Lopt</i> = BND	<i>Lopt</i> = ALL
<i>vtype</i> = Phys	<i>VAL1</i> and <i>VAL3</i> must not be blank.	<i>VAL1</i> through <i>VAL3</i> must not be blank.
<i>vtype</i> = Prob	<i>VAL1</i> = 0.01, <i>VAL3</i> = 0.99	<i>VAL1</i> = 0.01, <i>VAL2</i> = 0.50, <i>VAL3</i> = 0.99

See [Probabilistic Design](#) in the *Advanced Analysis Techniques Guide* for more information on the PDS methods.

## Menu Paths

**Main Menu>Prob Design>Prob Method>Response Surface**



**PDEF**, *Lab, Item, Comp, Avglab***Interpolates an item onto a path.**

POST1: Path Operations

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**Lab**

Label assigned to the resulting path item (8 characters maximum). This item may be used as input for other path operations.

**Item**

Label identifying the item for interpolation. Valid item labels are shown in [Table 239: PDEF - Valid Item and Component Labels](#) (p. 1222) below. Some items also require a component label.

**Comp**

Component of the item (if required). Valid component labels are shown in [Table 239: PDEF - Valid Item and Component Labels](#) (p. 1222) below.

**Avglab**

Option to average across element boundaries:

**AVG**

Average element results across elements (default).

**NOAV**

Do not average element results across elements. If the parameter *DISCON* = MAT on the **PMAP** command, this option is automatically invoked.

**Notes**

Defines and interpolates a labeled path item along a predefined path (**PATH**). Path item results are in the global Cartesian coordinate directions unless transformed (**RSYS**). A path item must be defined before it can be used with other path operations. Additional path items may be defined from the **PVECT**, **PCALC**, **PDOT**, and **PCROSS** commands. Path items may be listed (**PRPATH**) or displayed (**PLPATH**, **PLPAGM**). A maximum number of path items permitted is established by the *nSets* argument specified with the **PATH** command.

When you create the first path item (**PDEF** or **PVECT**), the program automatically interpolates four path items which are used to describe the geometry of the path. These predefined items are the position of the interpolated path points (labels XG, YG, and ZG) in global Cartesian coordinates, and the path length (label S). For alternate methods of mapping the path geometry (to include, for example, material discontinuity) see the **PMAP** command. These items may also be listed or displayed with the **PRPATH**, **PLPATH**, and **PLPAGM** commands.

If specifying that load case operations act on principal/equivalent stresses (**SUMTYPE**, PRIN), derived quantities (principal and equivalent stresses/strains) will be zero for path plots. A typical use for such a case involves mode combinations in a response spectrum analysis.

The number of interpolation points on the path is defined by the *nDiv* argument on the **PATH** command. See [Mapping Nodal and Element Data onto the Path](#) in the *Theory Reference for the Mechanical APDL and*

*Mechanical Applications* for details. Use **PDEF,STAT** to list the path item labels. Use **PDEF,CLEAR** to erase all labeled path items, except the path geometry items (XG, YG, ZG, S).

**Table 239 PDEF - Valid Item and Component Labels**

Item	Comp	Description
Valid item and component labels for nodal degree of freedom results are:		
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy (FLOTRAN).
ENDS		Turbulent energy dissipation (FLOTRAN).
Valid item and component labels for element results are:		
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or Equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPth	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.

Item	Comp	Description
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
For contact results PowerGraphics is applicable for 3-D models only.		
CONT	STAT2	Contact status.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
"	GAP	Contact gap distance.
"	FLUX	Total heat flux at contact surface.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
DC	X, Y, Z, SUM	Component conduction current density or vector sum (for elements that support conduction current calculation)
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component electromagnetic force or vector sum.
ETAB	Lab	Any user-defined element table label (see <b>ETABLE</b> command).
BFE	TEMP	Applied and calculated temperatures along a defined path.

**Valid item labels for FLOTRAN nodal results are:**

TTOT	Total temperature.
HFLU	Heat flux.
HFLM	Heat transfer (film) coefficient.
COND	Fluid laminar conductivity.
PCOE	Pressure coefficient.
PTOT	Total (stagnation) pressure.
MACH	Mach number.
STRM	Stream function. (2-D applications only.)
DENS	Fluid density.
VISC	Fluid laminar viscosity.
EVIS	Fluid effective viscosity.

Item	Comp	Description
CMUV		Turbulent viscosity coefficient.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.
SPHT		Specific heat.
RDFL		Radiation heat flux.

1. For [SHELL131](#) and [SHELL132](#) elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, ..., TTOP instead of TEMP.
2. For more information on the meaning of contact status and its possible values, see [Reviewing Results in POST1](#) in the [Contact Technology Guide](#).

## Menu Paths

**Main Menu>General Postproc>Path Operations>Clear Path Items**

**Main Menu>General Postproc>Path Operations>Define Path>Path Status>Current Path**

**Main Menu>General Postproc>Path Operations>Map onto Path**

**Main Menu>General Postproc>Path Operations>Map onto Path>FE Results**

**PDEXE**, *Slab*, *MRUN*, *NFAIL*, *FOPT*, *Fname*

**Executes the probabilistic analysis.**

[PROBABILISTIC: Run Probabilistic Analysis](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Slab*

Solution label. This is the name given to the files containing the results of different probabilistic analyses performed with a particular probabilistic model. Different solutions using different probabilistic methods or method options are stored in separate solution sets identified with this solution label. If the solution label is blank, ANSYS automatically assigns *Slab* = "SOLU $nn$ ", where  $nn$  is the current two-digit number of the probabilistic analysis (PDS allows a maximum of 10 solution sets identified by 10 solution labels); for example, the first analysis is SOLU01, the second is SOLU02, and so on up to SOLU10). This field cannot contain blanks. Maximum length of this field is 32 characters; if the field contains more than 32 characters, it will be truncated to 32 characters.

### *MRUN*

Keyword for the processing of the simulation loops

#### **SER**

Executes the simulation loops of the probabilistic analysis using a single machine; serial processing (default).

#### **PAR**

Executes the simulation loops of the probabilistic analysis in parallel using the ANSYS parallel-processing tool. See [Probabilistic Design](#) in the [Advanced Analysis Techniques Guide](#).

### *NFAIL*

This parameter sets the number of loops that are allowed to fail before the execution of the probabilistic analysis is terminated. For example, if geometric parameters are used as random variables then for some

simulations (loops) the loop will fail because meshing was not successful. This might happen infrequently which might be acceptable. However, if the probabilistic problem is incorrectly defined then loops could fail frequently or even in every loop. You can define an upper limit for the number of failed loops that will be tolerated before the probabilistic analysis will be terminated. If you use  $NFAIL = 0$  (default) then failed loops are not tolerated and the probabilistic analysis will terminate the first time a loop fails.

$NFAIL$  is used only if  $MRUN = PAR$ .

#### **FOPT**

Keyword for the file option. This determines if a copy of the sample file should be saved or not.

#### **DEL**

The sample file will be deleted (default).

#### **COPY**

A copy of the sample file will be saved as the file name specified with the  $Fname$  option.

#### **Fname**

Name of the copy of the sample file. The directory for storing the sample file is always the current working directory. The file extension is always `.csmpl`. This option is ignored for  $FOPT = DEL$ .

### **Command Default**

$Slab = SOLU_{nn}$  (see above) ,  $MRUN = SER$ ,  $NFAIL = 0$ ,  $FOPT = DEL$ .

### **Notes**

Runs the probabilistic analysis. The results of the probabilistic analysis are identified with the solution label specified here. This solution label is used for the postprocessing of the results. The maximum number of probabilistic analyses that can be executed, stored, and post-processed based on the same deterministic and probabilistic model in one session with the PDS is 10. However, it is not counted as a separate probabilistic analysis if the same solution label is used multiple times in order to append to or overwrite previous results.

The probabilistic results are stored in result files in ASCII format.

If you specify a solution label that has already been used, the results will be appended at the end of the results file.

After the **PDEXE** command has been issued, you can no longer change the probabilistic model using probabilistic preprocessing commands; doing so would invalidate the probabilistic analysis results and potentially corrupt the probabilistic database. Also, you are not allowed to change the probabilistic analysis file using a **PDANL** command after the first **PDEXE** has been executed.

If you need to change the probabilistic model (analysis file, random input variables, etc.) after the **PDEXE** command has been issued, you should use the **PDCLR**, **POST** command to clear the probabilistic results. We recommend that you first save the results using the **PDSAVE** command.

By default, ANSYS creates a sample file (called `Jobname.samp`) containing all values of all random input parameters; unless you specify otherwise, this file is deleted after the analysis is finished. With the  $FOPT$  and  $Fname$  options you can determine if and where a copy of that file should be stored for later use in another analysis.

## Menu Paths

Main Menu>Prob Design>Run>Exec Serial >Run Serial

**PDHIST**, *Rlab*, *Name*, *NCL*, *Type*

Plots the frequency histogram.

PROBABILISTIC: Postprocessing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Rlab*

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses).

### *Name*

Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command.

### *NCL*

Number of classes for the histogram plot. This is the number of bars shown in the histogram. *NCL* must be a positive number. If this field is left blank, ANSYS calculates an appropriate number of classes based on the sample size. ANSYS divides the range between the smallest and largest sample value into *NCL* classes of equal width and determines the histogram frequencies by counting the number of hits that fall in the classes.

### *Type*

Type of histogram.

#### **ABS**

Absolute frequency histogram. This is the actual number of hits in each class.

#### **REL**

Relative frequency histogram (default). This is the number of hits in the individual classes divided by the total number of samples.

#### **NORM**

Normalized frequency histogram. This is the number of hits in the individual classes divided by the total number of samples and divided by the width of the class. This normalization makes the histogram comparable to the probability density function.

## Command Default

*Rlab* as described above, *NCL* as described above, *Type* = REL.

## Notes

Plots the frequency histogram.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

The **PDHIST** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

## Menu Paths

Main Menu>Prob Design>Prob Results>Statistics>Histogram

**PDINQR**, *Rpar*, *Name*, *Type*, *VAL*

Evaluates statistical characteristics of a random input variable.

PROBABILISTIC: Preprocessing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Rpar*

Name of scalar ANSYS parameter into which the characteristic value is stored.

### *Name*

Parameter name. The parameter must have been previously defined as a random input variable using the **PDVAR** command.

### *Type*

Keyword for the type of information you need to retrieve.

### **MEAN**

Mean value of the random input variable specified in *Name*.

### **STDV**

Standard deviation of the random input variable

### **PDF**

Probability density function of the random input variable specified in *Name* at a given value specified by *VAL*.

### **CDF**

Cumulative distribution function of the random input variable specified in *Name* at a given value specified by *VAL*.

### **INV**

Inverse cumulative distribution function of the random input variable specified in *Name* at a given probability specified by *VAL*.

### *VAL*

Value needed for *Type* = PDF, CDF, or INV. For *Type* = PDF and *Type* = CDF, this is the value of the random input variable at which the probabilistic density or cumulative distribution function should be evaluated. For *Type* = INV, *VAL* indicates the probability at which you want the inverse cumulative distribution function evaluated.

## Notes

Evaluates statistical characteristics of a random input variable. The result is stored in the ANSYS parameter *Rpar*.

## Menu Paths

Main Menu>Prob Design>Prob Definitns>Inquire

---

**PDLHS**, *NSIM*, *NREP*, *ISopt*, *--*, *Astop*, *ACCMEAN*, *ACCSTDV*, *CHECK*, *Seed*

**Specifies options for Monte Carlo Simulations using Latin-Hypercube sampling.**

[PROBABILISTIC:Methods](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NSIM***

Number of simulation loops per repetition cycle.

### ***NREP***

Number of repetition cycles of the analysis.

### ***ISopt***

Latin-Hypercube sampling divides the domain of each random input variable into intervals of equal probability. The interval sampling option *ISopt* determines where the samples are located within each interval.

### **RAND**

Picks a random location within the interval (default).

### **MEAN**

Picks the mean value location within the interval.

### **MEDI**

Picks the median value location within the interval.

--

Unused field.

### ***Astop***

Autostop option label.

### **AUTO**

Enable Autostop. When Autostop is used, the PDS feature continues the simulation loops until the convergence criteria for the mean value and the standard deviation have been met or until the number of simulations *NSIM* are complete, whichever comes first. The convergence criteria (mean value and standard deviations of all random output parameters) are specified by the *ACCMEAN* and *ACCSTDEV* parameters. The criteria are met if the mean value *and* the standard deviations converge within the accuracy specified in the *ACCMEAN* and *ACCSTDEV* options. The convergence check is done every *i*-th loop, where *i* is specified in the *CHECK* parameter.

### **ALL**

Disable Autostop option. All Monte Carlo Simulations as specified by *NSIM* and *NREP* are performed (default).

### ***ACCMEAN***

Accuracy of the mean values of all random output parameters that must be met to activate Autostop. Default is 0.01 (1%). *ACCMEAN* is ignored for *Astop* = ALL. The convergence for the mean values is met if for all random output parameters *y* the following equation is true:



$$\frac{|\bar{y}(i) - \bar{y}(i - \text{CHECK})|}{\bar{y}(i)} \leq \text{ACCMEAN} \quad \text{with } i = 2 \cdot \text{CHECK}, 3 \cdot \text{CHECK}, \dots$$

where the value of *CHECK* is given by the *CHECK* option.

### **ACCSTDEV**

Accuracy of the standard deviations of all random output parameters that must be met to activate Autostop. The default is 0.02 (2%). *ACCSTDEV* is ignored for *Astop* = ALL. The convergence for the standard deviations is met if for all random output parameters *y* the following equation is true:

$$\frac{|\sigma_y(i) - \sigma_y(i - \text{CHECK})|}{\sigma_y(i)} \leq \text{ACCSTDEV} \quad \text{with } i = 2 \cdot \text{CHECK}, 3 \cdot \text{CHECK}, \dots$$

where the value of *CHECK* is given by the *CHECK* option.

### **CHECK**

Sets how often conditions for convergence are checked for Autostop. The PDS feature checks if the convergence criteria are met every *i*-th loop, where *i* is given by the *CHECK* parameter. The default value is 10. It is not recommended to use *CHECK* = 1, because it could cause Autostop to terminate the simulations prematurely. The mean values and standard deviation might not show large differences between all simulation loops but might still have a visible "global" trend if viewed over several simulations. This behavior indicates that convergence has not really been achieved. If you set *CHECK* = 1, then Autostop is not able to detect such a global trend. *CHECK* is ignored for *Astop* = ALL.

### **Seed**

Seed value label. Random number generators require a seed value that is used to calculate the next random number. After each random number generation finishes, the seed value is updated and is used again to calculate the next random number. ANSYS initializes the seed value with the system time when the ANSYS session started.

### **CONT**

Continues updating using the derived seed value (default).

### **TIME**

Initializes the seed value with the system time. You can use this if you want the seed value set to a specific value for one analysis and then you want to continue with a "random" seed in the next analysis. It is not recommended to "randomize" the seed value with the *Seed* = TIME option for multiple analyses. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

### **INIT**

Initializes the seed value using 123457 (a typical recommendation). This option leads to identical random numbers for all random input variables when the exact analysis will be repeated, making it useful for benchmarking and validation purposes (where identical random numbers are desired). If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

### **Value**

Uses the specified (positive) value for the initialization of the seed value. This option has the same effect as *Seed* = INIT, except you can choose an arbitrary (positive) number for the initialization. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

## Command Default

$NSIM = 30, NREP = 1, ISopt = RAND, Astop = ALL, ACCMEAN = 0.01, ACCSTDEV = 0.02, CHECK = 10, Seed = CONT.$

## Notes

Defines the number of simulations per repetition cycle, number of repetition cycles, specification of the Autostop option, checking frequency for the Autostop option, and the seed value for random number generation.

For Latin-Hypercube sampling, it is advantageous to divide the total number of requested simulations into a few repetitions. This adds more randomness to the sampling process. If  $NTOT$  is the total number of simulations, then as a rough rule of thumb  $NTOT$  should be  $NREP = \sqrt[4]{NTOT}$  repetitions. The number obtained with this rule of thumb must be adjusted such that  $NTOT = NREP * NSIM$ . For example if  $NTOT = 1000$  then  $NREP = \sqrt[4]{1000} = 10$ , so the 1000 simulations can be done in 100 simulations with 10 repetitions. If for example  $NTOT = 100$  then  $NREP = \sqrt[4]{100} = 3.16$ , which means that the 100 simulations could be broken up into either  $2 * 50$  or  $4 * 25$  simulations.

If Autostop is enabled then the maximum number of simulations to be performed is given by  $NSIM * NREP$ . The Autostop option will terminate the simulations before the  $NSIM * NREP$  simulations are done if the convergence criteria are met.

## Menu Paths

Main Menu>Prob Design>Prob Method>Monte Carlo Sims

## PDMETH, Method, Samp

Specifies the probabilistic analysis method.

[PROBABILISTIC:Methods](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Method

Label for the probabilistic analysis method.

#### MCS

Monte Carlo Simulation

#### RSM

Response Surface Method

### Samp

Label for the sampling techniques. The sampling technique determines the values of the random input variables during the simulation loops.

#### DIR

Direct or Crude Monte Carlo Sampling. This technique randomly samples the random input variables according to their distribution functions without "memory" of previous simulations.

The parameters for a Monte Carlo Simulation using direct sampling are specified with the **PDDMCS** command.

### LHS

Latin Hypercube Sampling (default). Valid only for *Method* = MCS. For this sampling technique the random input variables are sampled randomly according to their distribution functions, efficiently stratifying the samples into layers and avoiding the re-use of those layers. The sampling process has a "memory" of previous simulations, which prevents accumulation of clusters of samples. In addition, this sampling strategy forces the extreme ends of a distribution function to participate in the sampling. This generally leads to smoother distribution functions of the sampled set.

The parameters for a Monte Carlo Simulation using Latin-Hypercube sampling are specified with the **PDLHS** command.

### USER

User specified sampling. Valid only for *Method* = MCS. In this case you provide a file containing the sampling "points" (values) of all random input variables for all simulation loops. These samples are simply executed and it is your responsibility to specify the samples correctly. The probabilistic design system can perform only limited checks on the samples you provide. ANSYS allows Monte Carlo specific postprocessing operations on the results generated with user-specified samples. The parameters for the user-supplied sampling technique are specified with the **PDUSER** command.

### CCD

Central Composite Design. Valid only for *Method* = RSM. A central composite design is composed of a center point, axis points, and corner points, called factorial points. Using large numbers of random input variables produces prohibitively large numbers of factorial points; therefore, ANSYS automatically reduces the number of factorial points by switching to a fractional plan for the factorial part of the design. See the **PDDOEL** command for more information.

---

#### Note

This option is only valid for 2 to 20 random input variables. You will receive an error if you have specified fewer than 2 or more than 20 random input variables.

### BBM

Box-Behnken Matrix Design. Valid only for *Method* = RSM. A Box-Behnken Matrix design is composed of a center point plus the points at the middle of the edges of the hypercube in the space of random input variables. A Box-Behnken design might be advantageous if the corner points of the hypercube represent very extreme conditions that are undesirable and therefore should not be used for the sampling. See the **PDDOEL** command for more information.

---

#### Note

This option is only valid for 3 to 12 random input variables. You will receive an error if you have specified fewer than 3 or more than 12 random input variables.

### USER

User specified sampling. In this case you provide a file containing the sampling "points" (values) of all random input variables for all simulation loops. These samples are simply executed and it is your responsibility to specify the samples correctly. The PDS can perform only limited checks on the samples you provide, if user-supplied sampling technique are specified with the **PDUSER** command.

## Command Default

*Method* = MCS, *Samp* = LHS.

## Notes

Specifies the probabilistic analysis method and the sampling technique used for the individual probabilistic analysis method.

## Menu Paths

**Main Menu>Prob Design>Prob Method>Monte Carlo Sims**  
**Main Menu>Prob Design>Prob Method>Response Surface**

---

**PDOT**, *LabR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

**Calculates the dot product of two path vectors along the current path.**

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***LabR***

Label assigned to dot product result.

### ***LabX1***

X-component of first vector label (labeled path item).

### ***LabY1***

Y-component of first vector label (labeled path item).

### ***LabZ1***

Z-component of first vector label (labeled path item).

### ***LabX2***

X-component of second vector label (labeled path item).

### ***LabY2***

Y-component of second vector label (labeled path item).

### ***LabZ2***

Z-component of second vector label (labeled path item).

## Menu Paths

**Main Menu>General Postproc>Path Operations>Dot Product**

---

**PDPINV**, *Rlab*, *Name*, *PROB*, --, *CONF*

**Prints the result of the inversion of a probability.**

PROBABILISTIC: Postprocessing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***Rlab***

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses).

### ***Name***

Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command.

### ***PROB***

Target probability for which the random parameter value should be determined.

--

Unused field.

### ***CONF***

Confidence level. The confidence level is used to print the confidence bounds on the random parameter value. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%).

Printing of confidence bound is suppressed for  $CONF \leq 0.5$ . This parameter is ignored for response surface methods results postprocessing.

## **Command Default**

*Rlab* as described above,  $CONF = 0.95$ .

## **Notes**

Prints the value for the random parameter *Name* at which the probability that there are simulation values lower than that value is equal to *PROB*. This corresponds to an inversion of the cumulative distribution function (see **PDCDF** command) at a given probability. In this sense the **PDPINV** is doing the opposite of the **PDPROB** command. The **PDPROB** command evaluates a probability for a given random parameter value and the **PDPINV** command evaluates the random parameter value that corresponds to a given probability.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

The confidence level is a probability expressing the confidence that the value for the requested result is in fact between the confidence bounds. The larger the confidence level, the wider the confidence bounds. Printing the confidence bounds only makes sense for postprocessing Monte Carlo simulation results, where the confidence bounds represent the accuracy of the results. With increasing sample sizes, the width of the confidence bounds gets smaller for the same confidence level. For response surface analysis methods, the number of simulations done on the response surface is usually very large; therefore, the accuracy of the results is determined by the response surface fit and not by the confidence level.

The **PDPINV** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

## Menu Paths

Main Menu>Prob Design>Prob Results>Statistics>Inverse Prob

**PDPLOT**, *Name*, *PLOW*, *PUP*

Plots the distribution curves of a defined random input variable.

[PROBABILISTIC: Preprocessing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Name

Parameter name. The parameter name must have been previously defined as a random input variable using the **PDVAR** command.

### PLOW

Lower probability level used to determine the lower boundary of the curve. This probability must be between 0.0 and 1.0 and it defaults to 0.0025. This parameter is used to determine the lower plotting range (boundary) in case the random input variable does not have a minimum value (such as Gauss).

### PUP

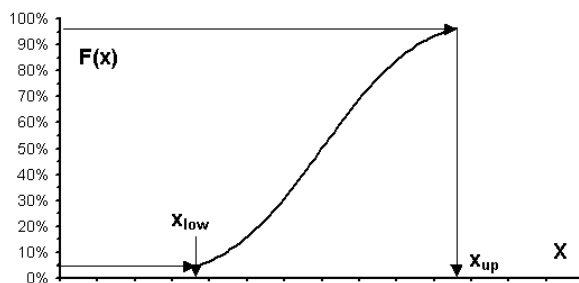
Upper probability level used to determine the upper boundary of the curve. This probability must be between 0.0 and 1.0 and it defaults to 0.9975.

## Command Default

*PLOW* = 0.0025 (applied if distribution type has no minimum), *PUP* = 0.9975 (applied if distribution type has no maximum).

## Notes

Plots the distribution of a defined random input variable. The **PDPLOT** command generates a probability density function plot as well as a cumulative distribution function plot of the random variable. The probabilities *PLOW* and *PUP* are used to determine the plot range of the random input variable values. To do this, the probabilities are converted into random input variable values using the inverse cumulative distribution function of the random input variable as shown in the following illustration.



Using the probabilities ensures that the boundaries are always feasible and meaningful for the random input variable regardless of its distribution type.

If *PLOW* is left blank, then a minimum value of the distribution is used for plotting, provided it exists (for example, uniform distribution). If the distribution type has no minimum value (for example, a Gaussian dis-

tribution) then the default value is used to determine the lower plotting range value. The same applies for the maximum value, if *PUP* is left blank.

## Menu Paths

Main Menu>Prob Design>Prob Definitns>Plot

**PDPROB**, *Rlab*, *Name*, *Relation*, *LIMIT*, --, *CONF*

Prints a probability result.

[PROBABILISTIC: Postprocessing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Rlab*

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses).

### *Name*

Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter using the **PDVAR** command.

### *Relation*

Relation label for the relation between probabilistic design parameter *Name* and the limit value *LIMIT*:

#### **LT**

Less than (default).

#### **GT**

Greater than.

### *LIMIT*

Limit value.

### --

Unused field.

### *CONF*

Confidence level. The confidence level is used to print the confidence bounds on the probability. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%). Printing of confidence bound is suppressed for  $CONF \leq 0.5$ . This parameter is ignored for response surface methods results postprocessing.

## Command Default

*Rlab* as described above, *Relation* = LT, *CONF* = 0.95.

## Notes

Prints the probability that the probabilistic design input or output variable denoted with *Name* is smaller or larger than a certain limit value.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

Use the relation label *Relation* to specify if you want to print a traditional probability value (LT) or the exceedence probability (GT). The *LIMIT* directly specifies at which value of *Name* (the design parameter) the probability should be evaluated. If *LIMIT* is between two sample values of *Name* the resulting probability is linearly interpolated between the sample values. If *LIMIT* is smaller than all sample values of *Name* the probability is 0.0 for *Relation* = LT and 1.0 for *Relation* = GT. If *LIMIT* is greater than all sample values for *Name* the probability is 1.0 for *Relation* = LT and 0.0 for *Relation* = GT.

The confidence level is a probability expressing the confidence that the value for the requested probability is in fact between the confidence bounds. The larger the confidence level, the wider the confidence bounds. Printing the confidence bounds only makes sense for postprocessing Monte Carlo simulation results, where the confidence bounds represent the accuracy of the results. With increasing sample sizes, the width of the confidence bounds gets smaller for the same confidence level. For response surface analysis methods, the number of simulations done on the response surface is usually very large; therefore, the accuracy of the results is determined by the response surface fit and not by the confidence level.

The **PDPROB** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

## Menu Paths

**Main Menu>Prob Design>Prob Results>Statistics>Probabilities**

**PDRESU**, *Fname*, *Ext*, --

**Reads the probabilistic model data and loads it into the database.**

[PROBABILISTIC: Database](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to `pds` if *Fname* is blank; otherwise, no default.

--

Unused field.

## Command Default

*Fname* = `jobname`, *Ext* = `pds`, the default directory is the current working directory.



## Notes

Reads the probabilistic model data from the specified file and loads it into the database. Probabilistic analyses results are not stored in the database with the **PDRESU** command, rather they reside in separate results files. Analyses results are loaded automatically (one-by-one and on demand) when a probabilistic postprocessing command is issued.

## Menu Paths

**Main Menu>Prob Design>Prob Database>Resume**

**PDROPT**, *RVAR*, *CORR*, *STAT*, *SHIS*, *HIST*, *CDF*, *SENS*, *CMAT*, *CONF*

**Specifies the options for an HTML report.**

[PROBABILISTIC:Postprocessing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***RVAR***

Specifies in which form to show the definitions of random variables in the report.

**0**

Using tables (including name and distribution parameter) and figures (including a probability density function plot and a cumulative distribution plot) (default).

**1**

Using tables only.

**2**

Using figures only.

**3**

None.

### ***CORR***

Specifies if a table describing the correlation between random variables should be included in the report.

**0**

Yes, include this table (default).

**1**

No, do not include this table.

### ***STAT***

Specifies which statistics to include in the report. In general, statistics are provided in a tabular form.

**0**

Statistics of the random output parameters only (default).

**1**

Statistics of the random input variables only.

**2**

Statistics of both the random input variables and the random output parameters.

**3**

None.

**SHIS**

Specifies which sample history plots to include in the report. This option applies to the random output parameters only.

**0**

None (default).

**1**

Mean value and standard deviation as a sample plot.

**2**

Mean value, standard deviation and sample values as a sample plot.

**3**

All types of sample plots - mean value, standard deviation, minimum value, maximum values, and the sample values.

**HIST**

Specifies which histogram plots to include in the report.

**0**

Histogram of the random output parameters only (default).

**1**

Histogram of the random input variables only.

**2**

Histogram of both the random input variables and the random output parameters.

**3**

None.

**CDF**

Specifies which cumulative distribution function (CDF) plots to include in the report.

**0**

CDF of the random output parameters only (default).

**1**

CDF of the random input variables only.

**2**

CDF of both the random input variables and the random output parameters.

**3**

None.

**SENS**

Specifies which sensitivity plots to include in the report.

**0**

Plots the sensitivities for all random output parameters based on Spearman-rank-order correlation coefficient (default).

**1**

Plots the sensitivities for all random output parameters based on linear (Pearson) correlation coefficient.

**2**

Plots the sensitivities according to option *SENS=1* and *SENS=2*.

**3**

None.

**CMAT**

Specifies which correlation matrices to include in the report.

- 0** Correlation matrix between random output parameters and random output parameters only (default).
- 1** Correlation matrix between random input variables and random output parameters only.
- 2** Correlation matrix between random input variables and random input variables only.
- 3** Correlation matrices according to option *CMAT*=0 and *CMAT*=1.
- 4** Correlation matrices according to option *CMAT*=0 and *CMAT*=2.
- 5** Correlation matrices according to option *CMAT*=1 and *CMAT*=2.
- 6** Correlation matrices according to option *CMAT*=0, *CMAT*=1, and *CMAT*=2.
- 7** None.

**CONF**

Confidence level. The confidence level is used to plot confidence bounds for the history value. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%). Confidence bound(s) plotting is suppressed for  $CONF \leq 0.5$ . This option is ignored if the report does not include plots for which confidence bounds are applicable.

**Command Default**

*RVAR* = 0, *CORR* = 0, *STAT* = 0, *SHIS* = 0, *HIST* = 0, *CDF* = 0, *SENS* = 0, *CMAT* = 0, *CONF* = 0.95.

**Notes**

Specifies the options for an HTML report. An HTML report includes a description of the deterministic model, the probabilistic model, the probabilistic methods used for the analyses, and the results obtained from the analyses. The deterministic model is documented in the report by including a link to the analysis file (see **PDANL** command). In addition, an element plot of the component is shown, if available, based on the current view settings. The command **ALLSEL** is issued automatically prior to the respective plot command.

**Menu Paths**

**Main Menu>Prob Design>Prob Results>Report>Report Options**

## /PDS

Enters the probabilistic design system.

PROBABILISTIC: Auxiliary

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Enters the Probabilistic Design System (PDS). This command is valid only at the Begin Level.

## Menu Paths

Main Menu>Prob Design

---

**PDSAVE**, *Fname*, *Ext*, --

Writes the probabilistic model data to a file.

PROBABILISTIC: Database

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to `pds` if *Fname* is blank; otherwise, no default.

--

Unused field.

## Command Default

*Fname* = `Jobname`, *Ext* = `pds`, the default directory is the current working directory.

## Notes

Writes the probabilistic model data to a file. Saved data include probabilistic data only; the results of the probabilistic analyses are not stored in the file (rather, these are stored in separate result files).

## Menu Paths

Main Menu>Prob Design>Prob Database>Save

---

**PDSCAT**, *Rlab*, *Name1*, *Name2*, *Type*, *ORDER*, *NMAX*

### Plots a scatter graph.

PROBABILISTIC: Postprocessing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Rlab*

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command, or the response surface set label defined in an **RSFIT** command.

#### *Name1*, *Name2*

Parameter names. The parameters must have been previously defined as a random input variable or a random output parameter using the **PDVAR** command. The parameter data for *Name1* is shown on the X-axis and the parameter data for *Name2* is shown on the Y-axis in the plot.

#### *Type*

Keyword for the type of trendline curve.

##### **POLY**

Polynomial trendline (default).

##### **NONE**

A trendline is not plotted.

#### *ORDER*

Order of the polynomial trendline. This parameter is used only for *Type* = POLY. *ORDER* must be a positive number. There is no maximum for *ORDER* provided there are enough data points to evaluate a polynomial of the requested order. Default is 1.

#### *NMAX*

Maximum number of points plotted in the scatter plot. If there are more sample data, then only the first *NMAX* points are plotted. The default value is 10,000.

### Command Default

*Rlab* as described above, *Type* = POLY, *ORDER* = 1, *NMAX* = 10,000

### Notes

Plots a scatter graph with or without a trendline. The scatter plot shows the simulated points for two random parameters. Random input variables and random output parameters are valid for both X- and Y-axis. The mean value of both parameters are marked with separate green lines. The point where the green lines cross marks the statistical center of gravity of the cloud of all simulated data points of the two parameters.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

If the parameter data for *Name1* includes negative values, fitting a logarithmic trendline is not possible and the logarithmic trendline plot is suppressed if requested. The same applies for an exponential trendline if the data for the *Name2* includes negative values.

Because of the amount of data involved, the scatter plot is limited to *NMAX* points. If shown, the trendline is evaluated only on the *NMAX* points shown. However, the statistical information shown in the plot, such

as the mean value lines for both parameters and the correlation coefficients listed in the legend are based on the full set of samples. If required, you can increase *NMAX* to plot more points, but this will affect the time needed to process the **PDSCAT** command. If *NMAX* is less than the total amount of simulated points, which is typically possible for Response Surface Methods, you will see an appropriate warning in the plot legend.

## Menu Paths

**Main Menu>Prob Design>Prob Results>Trends>Scatter Plot**

---

**PDSSENS**, *Rlab*, *Name*, *Chart*, *Type*, *SLEVEL*

**Plots the probabilistic sensitivities.**

[PROBABILISTIC: Postprocessing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Rlab*

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses). The **PDSSENS** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

### *Name*

Parameter name. The parameter must have been previously defined as a random output parameter using the **PDVAR** command.

### *Chart*

Keyword for the type of chart to be plotted.

#### **BAR**

Bar chart of the absolute sensitivities.

#### **PIE**

Pie chart of relative and normalized sensitivities.

#### **BOTH**

Both pie and bar charts plotted side by side (default).

### *Type*

Keyword for the type of correlation coefficients used to evaluate the sensitivities.

#### **RANK**

Spearman rank-order correlation coefficient (default).

#### **LIN**

Pearson linear correlation coefficient.

### *SLEVEL*

Significance level. The value for the significance level must be between 0.0 and 1.0 and it defaults to 0.025 (2.5%).

## Command Default

*Rlab* as described above, *Chart* = BOTH, *Corr* = RANK, *SLEVEL* = 0.025

## Notes

Plots the probabilistic sensitivities.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

Evaluation of the probabilistic sensitivities is based on the correlation coefficients between all random input variables and the random output parameter specified by *Name*. You can choose which correlation coefficient should be used for that evaluation using the *Corr* option. For all sensitivity values, the probabilistic design tool evaluates the probability that the sensitivity can be neglected, based on statistical test theory. If this probability exceeds the significance level as specified by the *SLEVEL* parameter, the sensitivity value should be regarded as negligible or insignificant. The higher the significance level (*SLEVEL*) the more sensitivities are considered as significant. The sensitivity plot includes the significant sensitivities only and lists the insignificant ones separately.

## Menu Paths

**Main Menu>Prob Design>Prob Results>Trends>Sensitivities**

**PDSHIS**, *Rlab*, *Name*, *Type*, *CONF*

**Plots the sample history values.**

[PROBABILISTIC:Postprocessing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Rlab*

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command, or the response surface set label defined in an **RSFIT** command.

### *Name*

Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter using the **PDVAR** command.

### *Type*

Keyword to identify which type of data is to be plotted:

#### **SAMP**

Sampled values (default).

#### **MEAN**

Mean values for results based on Monte Carlo simulation methods only.

#### **STDV**

Standard deviations for results based on Monte Carlo simulation methods only.

#### **MIN**

Minimum values for results based on Monte Carlo simulation methods only.

#### **MAX**

Maximum values for results based on Monte Carlo simulation methods only.

MEAN, STDV, MIN, and MAX are only valid for Monte Carlo methods.

**CONF**

Confidence level. The confidence level is used to plot confidence bounds for the history value. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%). Confidence bound(s) plotting is suppressed for  $CONF \leq 0.5$ . This option is ignored for  $Type = SAMP$  (no confidence bounds are plotted).

**Command Default**

*Rlab* as described above,  $Type = SAMP$ ,  $CONF = 0.95$

**Notes**

Plots the sample history values as a function of the number of simulation loops.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

The confidence level is a probability expressing the confidence that the value for the requested probability is in fact between the confidence bounds. The larger the confidence level, the wider the confidence bounds. For  $Type = MEAN$  and  $Type = STDV$ , lower and upper confidence curves are plotted. For  $Type = MEAN$ , the mean value curve starts at the first simulation and the confidence bounds start with simulation number 2. For  $Type = MIN$  only the upper confidence bound is shown (the interpretation is that by a certain probability the true minimum value is below this curve). This probability (or confidence) is set using *CONF*. Likewise, for  $Type = MAX$ , only the lower confidence bound is shown. For all *Type* options, confidence curves are plotted starting with the simulation at which enough data is available to calculate the bounds. However, for scaling reasons, no confidence bounds are plotted for simulation numbers 1 to 5 even if they might be available.

**Menu Paths**

**Main Menu>Prob Design>Prob Results>Statistics>Sampl History**

**PDUSER**, *Fname*, *Ext*, --

**Specifies options for user-specified sampling methods.**

[PROBABILISTIC: Methods](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

**Ext**

Filename extension (8 character maximum).

--

Unused field.



## Notes

If user-specified sampling methods are requested with the **PDMETH**, MCS, USER command or the **PDMETH**, RSM, USER command, then you need to specify which file contains the sample data. For more information on the format of this file, see [Probabilistic Design](#) in the *Advanced Analysis Techniques Guide*.

## Menu Paths

**Main Menu>Prob Design>Prob Method>Monte Carlo Sims**

**Main Menu>Prob Design>Prob Method>Response Surface**

**PDVAR**, *Name*, *Type*, *PAR1*, *PAR2*, *PAR3*, *PAR4*

**Specifies the parameters to be treated as probabilistic design variables.**

PROBABILISTIC: Preprocessing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Name

Parameter name (must be a scalar ANSYS parameter). The parameter must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command. See the **\*SET** command for restrictions about ANSYS parameters.

### Type

Probabilistic design variable type. This is the statistical distribution type. For more information on each of these types, see [Probabilistic Design](#) in the *Advanced Analysis Techniques Guide*.

#### BETA

Beta distributed random variable.

*PAR1* = Shape parameter. Defaults to 2.0.

*PAR2* = Shape parameter. Defaults to 2.0.

*PAR3* = Lower minimum value. Defaults to 0.0.

*PAR4* = Upper maximum value. Defaults to 1.0.

#### EXPO

Exponential distributed random variable.

*PAR1* = Decay parameter  $\lambda$ . Must be larger than 0.0 and defaults to 1.0.

*PAR2* = Shift or minimum value. Defaults to 0.0.

*PAR3*, *PAR4* are ignored.

#### GAMA

Gamma distributed random variable.

*PAR1* = Decay parameter  $\lambda$ . Must be larger than 0.0 and defaults to 1.0.

*PAR2* = Exponential parameter k. Must be larger than 0.0 and defaults to 1.0.

*PAR3, PAR4* are ignored. Exponential distributed random variable.

**GAUS**

Gaussian (Normal) distributed random variable.

*PAR1* = Mean value. Defaults to 0.0.

*PAR2* = Standard deviation. Must be larger than 0.0 and defaults to 1.0.

*PAR3, PAR4* are ignored.

**LOG1**

Lognormal distributed random variable specified directly with the statistical parameters mean value and standard deviation.

*PAR1* = Mean value. Must be larger than 0.0 and defaults to 1.0.

*PAR2* = Standard deviation. Must be larger than 0.0 and defaults to 1.0.

*PAR3, PAR4* are ignored.

**LOG2**

Lognormal distributed random variable specified with the statistical parameters mean value and standard deviation of the logarithm of the random values.

*PAR1* and *PAR2* must also be defined. *PAR1* = Mean value of the logarithm of the data. Defaults to 0.0.

*PAR2* = Standard deviation of the logarithm of the data. Must be larger than 0.0 and defaults to 1.0.

*PAR3, PAR4* are ignored.

**UNIF**

Uniform distributed random variable. Note that *PAR1* must be less than *PAR2*.

*PAR1* = Minimum value. Defaults to 0.0.

*PAR2* = Maximum value. Defaults to 1.0.

*PAR3, PAR4* are ignored.

**TGAU**

Truncated Gaussian distributed random variable. Note that *PAR3* must be less than *PAR4*.

*PAR1* = Mean value of the untruncated Gaussian distribution. Defaults to 0.0.

*PAR2* = Standard deviation of the untruncated Gaussian distribution. Must be larger than 0.0 and defaults to 1.0.

*PAR3* = Minimum value and lower truncation boundary. Defaults to -3.0.

*PAR4* = Maximum value and upper truncation boundary. Defaults to +3.0.

**TRIA**

Triangular distributed random variable. Note that *PAR1* must be less than *PAR2* which must be less than *PAR3*.

*PAR1* = Minimum value. Defaults to 0.0.

*PAR2* = Most Likely Value (MLV). Defaults to 0.5.

*PAR3* = Maximum value. Defaults to 1.0.

*PAR4* is ignored.

### **WEIB**

Weibull (Type III smallest) distributed random variable. Note that *PAR2* must be greater than *PAR3*. If *PAR3* = 0.0, the random distribution is equivalent to a two-parameter Weibull distribution.

*PAR1* = Weibull exponent. Must be larger than 0.0 and defaults to 1.0.

*PAR2* = Characteristic value. Must be larger than 0.0 and defaults to 1.0.

*PAR3* = Shift or minimum value. Defaults to 0.0.

*PAR4* is ignored.

### **RESP**

Random output or response parameter. *PAR1* to *PAR4* are not used.

### **DEL**

Deletes this probabilistic design variable (does not delete the ANSYS parameter). This option is only valid if the parameter *Name* was previously defined as a probabilistic design variable (using *Type* = BETA, ..., WEIB or *Type* = RESP). The parameter retains the value assigned during the last probabilistic design loop. *PAR1* to *PAR4* are not used.

### ***PAR1, PAR2, PAR3, PAR4***

Parameters of the distribution function. The parameters must be specified according to the requirements of the individual distribution types described above.

## **Notes**

Specifies the parameters to be treated as probabilistic design variables. A random input variable is specified by the name of the ANSYS parameter, the type of the distribution function (*Type*) and its distribution parameters (*PAR1, ..., PAR4*). A random output parameter is specified by the name of the ANSYS parameter and the type identifying it as a random output parameter (*Type* = RESP).

## **Menu Paths**

**Main Menu>Prob Design>Random Input**

**PDWRITE**, *File, Fnam, Lnam*

**Generates an HTML report for the probabilistic analyses.**

[PROBABILISTIC:Postprocessing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**File**

File name and directory path (248 characters maximum, including directory) from which to read the report. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

**Fnam**

First name of the author of the report (32 characters maximum). This first name must not include blanks.

**Lnam**

Last name of the author of the report (32 characters maximum). This last name must not include blanks.

**Notes**

Generates an HTML report for the probabilistic analysis. An HTML report includes a description of the deterministic model, the probabilistic model, the probabilistic methods used for the analyses and the results obtained from the analyses.

**Menu Paths**

**Main Menu>Prob Design>Prob Results>Report>Generate Report**

**PERBC2D**, *LOC1, LOC2, LOCTOL, R1, R2, TOLR, OPT, PLNOPT*

**Generates periodic constraints for 2-D planar magnetic field analyses.**

[PREP7:Special Purpose](#)

MP <> <> <> <> <> <> <> EM EH <> PP <> EME <>

**LOC1**

Constant coordinate location of the first plane of nodes. For *PLNOPT* = 1 or 2, the constant coordinate location is the global Cartesian coordinate system [**CSYS**,0] location in the X or Y direction respectively. For *PLNOPT* = 0, the location is the angle in the global cylindrical coordinate system [**CSYS**,1].

**LOC2**

Constant coordinate location of the second plane of nodes. For *PLNOPT* = 1 or 2, the constant coordinate location is the global Cartesian coordinate system [**CSYS**,0] location in the X or Y direction respectively. For *PLNOPT* = 0, the location is the angle (in degrees) in the global cylindrical coordinate system [**CSYS**,1].

**LOCTOL**

Tolerance on the constant coordinate location for node selection. Defaults to .00001 for *PLNOPT* = 1 or 2 and .001 degrees for *PLNOPT* = 0.

**R1**

Minimum coordinate location along the second plane of nodes. For *PLNOPT* = 1 or 2, the coordinate location is the global Cartesian coordinate system location in the Y or X direction respectively. For *PLNOPT* = 0, the coordinate location is the radial coordinate value in the global cylindrical coordinate system. Periodic conditions are not applied to nodes at this location.

**R2**

Maximum coordinate location along the second plane of nodes. For  $PLNOPT = 1$  or  $2$ , the coordinate location is the global Cartesian coordinate system location in the Y or X direction respectively. For  $PLNOPT = 0$ , the coordinate location is the radial coordinate value in the global cylindrical coordinate system. Periodic conditions are not applied to nodes at this location.

**TOLR**

Tolerance dimension on node selection along the plane of nodes. Defaults to .00001.

**OPT**

Periodic option:

**0**

Odd symmetry (default). Apply constraint equations such that  $AZ(i) = -AZ(j)$ .

**1**

Even symmetry. Apply node coupling such that  $AZ(i) = AZ(j)$ .

**PLNOPT**

Symmetry plane option:

**0**

Planes of constant angle in the global cylindrical coordinate system [**CSYS**,1].

**1**

Planes parallel to the global Cartesian X axis [**CSYS**,0].

**2**

Planes parallel to the global Cartesian Y axis [**CSYS**,0].

**Notes**

**PERBC2D** invokes an ANSYS macro which generates periodic boundary condition constraints for 2-D planar magnetic field analysis. The macro is restricted to node pairs sharing common coordinate values along symmetry planes separated by a constant coordinate value. Planes (or lines) must lie at either constant angles ( $PLNOPT = 0$ ), constant X values ( $PLNOPT = 1$ ), or constant Y values ( $PLNOPT = 2$ ). **PERBC2D** applies constraint equations ( $OPT = 0$ , odd symmetry) or node coupling ( $OPT = 1$ , even symmetry) to each node pair sharing a common coordinate value along the symmetry planes. By default, periodic conditions are not applied at the first and last node pairs on the symmetry planes unless the input location values, R1 and R2, are adjusted to be less than or greater than the actual node coordinate values. Nodes are selected for application of the constraints using the **NSEL** command with tolerances on the constant coordinate location (*LOCTOL*) and the coordinate location along the plane (RTOL).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Periodic BCs**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Periodic BCs**

**PERI**, *DX, DY, DZ*

**Specifies periodic boundary conditions in an incompressible flow analysis.**

[PREP7:FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

***DX***

Offset in the X direction of the second boundary from the first.

***DY***

Offset in the Y direction of the second boundary from the first.

***DZ***

Offset in the Z direction of the second boundary from the first.

**Notes**

Specified offsets must be consistent with the flow field coordinate system, as dictated by the [FLUID141](#) and [FLUID142](#) KEYOPT(3) setting.

The meshes at the two boundaries must be identical.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**This command cannot be accessed from a menu.**

**PERTURB**, *Type, MatKey, ContKey, LoadControl*

**Sets linear perturbation analysis options.**

[SOLUTION:Analysis Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***Type***

Type of linear perturbation analysis to be performed:

**MODAL**

Perform a linear perturbation modal analysis.

**BUCKLING**

Perform a linear perturbation eigenvalue buckling analysis. This option is only valid in the ANSYS Workbench product.

**Note**

For a linear perturbation buckling analysis, the base analysis must be a linear static analysis. Bonded contact elements can be used in the base analysis.

**OFF**

Do not perform a linear perturbation analysis (default).

**MatKey**

Key that controls how the linear perturbation analysis will use material properties. This key controls all the structural elements, except for contact elements.

**AUTO**

The program automatically decides the material properties for linear perturbation (default). The materials are handled in the following way:

- For pure linear elastic materials used in the base analysis, the same properties are used in the linear perturbation.
- For hyperelastic materials used in the base analysis, the material properties are assumed to be linear elastic in the linear perturbation analysis. The material property data (or material Jacobian) is obtained based on the tangent of the hyperelastic material's constitutive law at the point where restart is made.
- For any nonlinear materials other than hyperelastic materials in the base analysis, the material properties are assumed to be linear elastic in the linear perturbation analysis. The material data is the same as the linear portion of the nonlinear materials (that is, the parts defined by **MP** commands).

See [Linear Perturbation Analysis](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information.

**ContKey**

Key that controls contact status for the linear perturbation analysis. This key controls all contact elements ([TARGE169](#), [TARGE170](#), and [CONTA171](#) through [CONTA178](#)) globally for all contact pairs. Alternatively, contact status can be controlled locally per contact pair by using the **CNKMOD** command. Note that local contact controls specified by **CNKMOD** override global settings specified by *ContKey* = CURRENT. For *ContKey* = STICKING or BONDED, the contact status always follows local controls defined by the **CNKMOD** command first, and is then adjusted by the global sticking or bonded control.

**CURRENT**

Use the current contact status from the restart snapshot (default). If the previous run is nonlinear, then the nonlinear contact status at the point of restart is frozen and used throughout the linear perturbation analysis.

**STICKING**

For frictional contact pairs, use sticking contact stiffness and status even when the status is sliding (i.e., no sliding status is allowed). This option only applies to contact pairs whose frictional coefficient is greater than zero.

**BONDED**

Any contact pairs that are in the closed (sticking/sliding) state will be moved to bonded. Contact pairs that have a status of far/near field will remain open.

**LoadControl**

Key that controls how the load vector of  $\{F_{\text{perturbed}}\}$  is calculated. This control is provided for convenience of load generation for linear perturbation analysis. In general, a new set of loads is required for a linear perturbation analysis. This key controls all mechanical loads except for thermal loads; thermal loads are always kept (i.e., not deleted).

**ALLKEEP**

Keep all the boundary conditions (loads and constraints) from the end of the load step of the current restart point. This option is convenient for further load application and is useful for a linear perturbation analysis restarted from a previous linear analysis. For this option,  $\{F_{\text{end}}\}$  is the total load vector at the *end* of the load step at the restart point.

**INERKEEP**

Delete all loads and constraints from the restart step, except for displacement constraints and inertia loads (default). All displacement constraints and inertia loads are kept for convenience when performing the linear perturbation analysis. Note that nonzero and tabular displacement constraints are actually external loads.

**PARKEEP**

Delete all loads and constraints from the restart step, except for displacement constraints. All displacement constraints are kept for convenience when performing the linear perturbation analysis. Note that nonzero and tabular displacement constraints are actually external loads.

**NOKEEP**

Delete all the loads and constraints, including all displacement constraints. For this option,  $\{F_{end}\}$  is zero unless thermal loads are present.

**Command Default**

Linear perturbation analysis is disabled ( $Type = OFF$ ) by default. When the linear perturbation analysis is enabled, linear material property behavior is assumed for stress calculations; contact status for all contact pairs from the point of restart will be used by default; and all loads and constraints from the restart step are deleted, except for displacement constraints and inertia loads, by default.

**Notes**

This command controls options relating to [linear perturbation analyses](#). PERTURB must be issued in the first phase of a linear perturbation analysis.

A linear perturbation analysis consists of two phases (two **SOLVE** commands). The first phase starts with the **ANTYPE,,RESTART,,,PERTURB** command and ends with the **SOLVE,ELFORM** command. The purpose of the first phase is to re-establish a snapshot of the stiffness matrices at the specified restart point. The second phase, ending with the second **SOLVE** command, is for the actual linear perturbation analysis.

The total perturbed loads are calculated as follows:

$$\{F_{perturbed}\} = \{F_{end}\} + \{F_{add}\}$$

where:

$\{F_{end}\}$  = total loads at the *end* of the load step of the current restart point (load applications are read from the `.LDHI` file). By default, all of the loads in  $\{F_{end}\}$  are deleted except for displacement boundary conditions and inertia loads (see the description of `LoadControl` above).

$\{F_{add}\}$  = Additional (new) loads prescribed by the user in the second phase of the linear perturbation analysis (after the first **SOLVE** command is invoked).

In the first phase of a linear perturbation analysis, the **ANTYPE,,RESTART** command will resume the `JOB-name.RDB` database and read in the `.LDHI` file to establish the  $\{F_{end}\}$  load. New load application (adding to  $\{F_{add}\}$ ) or load removal (changing  $\{F_{end}\}$ ) can be done only in the second phase of the linear perturbation analysis (after the first **SOLVE** command). This allows flexibility in controlling the final  $\{F_{perturbed}\}$  that will be used.

For  $Type = MODAL$ ,  $\{F_{perturbed}\}$  will be calculated and stored in the `.FULL` and `.MODE` files for a subsequent mode-superposition, PSD, or other type of modal-based linear dynamic analysis. Linear dynamic options



such as multiple load generations (**MODCONT**,ON), enforced motion (**MODCONT**,,ON), and residual vector methods (**RESVEC**,ON) can be used in a linear perturbation analysis. For these methods, the **MODCONT** or **RESVEC** command should be invoked in the second phase (after the first **SOLVE**) of the linear perturbation procedure. For the enforced motion method, the base identification number should be specified (**D** command) in the second phase of the linear perturbation analysis. This base identification number will be used later in the downstream mode superposition or other mode superposition based analysis.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Restart**

**Main Menu>Solution>Analysis Type>Restart**

---

**PFACT**, *TBLNO*, *Excit*, *Parcor*

**Calculates participation factors for the PSD or multi-point response spectrum table.**

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> <> EME MFS

### *TBLNO*

Input PSD (Power Spectral Density) table number for which participation factors are to be calculated.

### *Excit*

Label defining the location of excitation:

#### **BASE**

Base excitation (default).

#### **NODE**

Nodal excitation.

### *Parcor*

Label defining excitation type (applies only to **SPOPT**,PSD analysis). Used only when partially correlated excitation is due to wave propagation or spatial correlation. Defaults to partially correlated excitation as defined by **COVAL** and **QDVAL** commands.

#### **WAVE**

Excitation defined by **PSDWAV** command.

#### **SPAT**

Excitation defined by **PSDSPL** command.

## Notes

Calculates the participation factors for a particular PSD or multi-point response spectrum table defined with the **PSDVAL** or **SPVAL** command. The `Jobname.DB` file must contain modal solution data in order for this command to calculate the participation factor. There must be a **PFACT** command for each excitation spectrum. You are limited to 100 excitations.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Calculate PF

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Calculate PF

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Calculate PF

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Calculate PF

**PGRAPH**, *Option*, *Fname*, *Fext*, --

Specifies the location from which graphics data will be retrieved for viewing.

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Option*

Switch that controls the PGR file write operations :

#### **Off**

Use the data currently in the data base for display (default).

#### **On**

Use the PowerGraphics data in the PGR file for display.

### *Fname*

File name and directory path (248 characters maximum, including directory) from which to read the PGR file. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Jobname*, or *File* if no jobname is specified.

### *Fext*

Filename extension (8 character maximum).

--

Unused field.

## Menu Paths

Main Menu>Solution>Output Controls>PGR File

Main Menu>General Postproc>Write PGR File

**PGRSET**, *Lstep*, *SBSTEP*, --, *KIMG*, *TIME*, --, *NSET*

Defines the data set to be read from the PGR file.

[POST1: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lstep*

Load step number of the data set to be read (defaults to 1):

#### *N*

Read load step *N*.

**FIRST**

Store the first data set on the PGR file (*SBSTEP* and *TIME* are ignored).

**LAST**

Store the last data set on the PGR file. (*SBSTEP* and *TIME* are ignored).

**NEXT**

Store the next data set on the PGR file (*SBSTEP* and *TIME* are ignored). If at the last data set, the first data set will be read as the next.

**LIST**

Scan the PGR file and list a summary of each load step. (*FACT*, *KIMG*, *TIME* and *ANGLE* are ignored.)

***SBSTEP***

Substep number (within *Lstep*). If *Lstep* = LIST, *SBSTEP* = 0 or 1 lists the basic step information, whereas *SBSTEP* = 2 also lists the load step title, and labels imaginary data sets if they exist. Default maximum is 1000. When the number of substeps exceeds this limit, you need to issue **SET**, *Lstep*, LAST to bring in the 1000th load step. Use **/CONFIG** to increase the limit.

--

Unused field.

***KIMG***

Used only with results from complex analyses.

**0**

Store real part of complex solution.

**1**

Store imaginary part.

***TIME***

Time-point identifying the data set to be read. For the harmonic response analyses, time corresponds to the frequency. If both *Lstep* and *SBSTEP* are zero (or blank), read data set at time = *TIME*. If *TIME* is beyond the last time point on the file, the last time point will be used. If *TIME* does not match a time value on the PGR file, the nearest time point in the data set will be used.

--

Unused field.

***NSET***

Data set number of the data set to be read. If a positive value for *NSET* is entered, *Lstep*, *SBSTEP*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **PGRSET**, LIST.

**Notes**

Defines the data set to be read from the PGR file into the database. Various operations may also be performed during the read operation.

**Menu Paths**

**Main Menu>Solution>OutputCtrls>PGR File**

**Main Menu>General Postproc>Write PGR File**

**PGSAVE**, *Fname*, *Fext*, *--*, *DataType*, *InteriorKey*, *Append*

**Creates a PowerGraphics (PGR) file from results data.**

SOLUTION: [Misc Loads](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to *Jobname*, or *File* if no jobname is specified.

***Fext***

Filename extension (8 character maximum).

--

Unused field.

***DataType***

Data type to create. This specification applies to discontinuous results data such as stress, strain, and field data. .

**0**

Save as nodal-averaged data (default). Used by the **PLNSOL** command.

**1**

Reserved.

**2**

Save both nodal averaged and unaveraged data. Used by the **PLESOL** and **PLNSOL** commands.

***InteriorKey***

Key that controls whether or not internal model data is saved to the PGR file. Internal data is required in order to use graphical slicing, capping, vector displays, or Isosurface displays for data display. Interior data is also required for the **AVRES**, FULL option, where surface results take into consideration the interior element contributions.

**0**

Save exterior surface data only. (default)

**1**

Save exterior and interior data.

***Append***

Append or overwrite data to the specified.

**0**

Append the data to the existing file (default). Note that the geometry must not have been changed between the write operations.

**1**

Overwrite the data file.

## Notes

The user may elect to write results to the PGR file in a specified order. The specifications (including **/ESHAPE** parameters) must be defined before saving the file. The GUI will allow the user to write the current set, or all sets from the results file.

For contact analysis, only flex-flex analysis is supported.

## Menu Paths

**Main Menu>Solution>OutputCtrls>PGR File**  
**Main Menu>General Postproc>Write PGR File**

---

**PGSELE**, *Type, Item, --, VMIN, VMAX, VINC*

Select a subset of elements for display with the PGR viewer.

[GRAPHICS: Labeling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Type*

Label identifying type of select

#### **S**

Select a new set (default)

#### **R**

Reselect a set from the current set

#### **A**

Additionally select a set, extending the current set.

#### **U**

Unselect a set from the current set.

#### **ALL**

Restore the full set. Subsequent command arguments are ignored.

#### **INVE**

Invert the current set (change selected to unselected; change unselected to selected). Subsequent command arguments are ignored.

#### **STAT**

Display the current select status. Subsequent command arguments are ignored.

### *Item*

Label identifying the data. Valid labels are:

If *Type* = STAT and *Item* = (blank) or ELEM, the element selection information is provided.

Type - Element type number

Mat - Material number

Real - Real constant number

Sec - Section identification number

Comp or CM - Component name(s). (see notes, below)

Pick - Pick entities via a picking window. (see notes, below)

(Defaults to Type).

--

Unused field.

**VMIN**

Minimum value of *Item* range. Ranges are attribute numbers.

**VMAX**

Maximum value for *Item* range. VMAX defaults to VMIN for input values.

**VINC**

Value increment within *Item* range. Used only with integer ranges (such as for attribute numbers). Defaults to 1. VINC cannot be negative.

## Command Default

All elements are selected.

## Notes

When *Item* = Pick, interactive picking is enabled. The displayed facets of the elements you pick will be flagged according to how they are resolved in PowerGraphics. Exterior elements will be flagged on each facet surface, while elements with no external (surface) component will be flagged at the centroid of that element.

When *Item* = Comp, the subsequent values for *VMIN*, *VMAX*, and *VINC*, are replaced by valid component names (32 character max). You can name up to 16 components in this fashion. All named components must contain elements.

This command is valid in POST1

## Menu Paths

**Main Menu>General Postproc>Results Viewer**

---

**PGWRITE**, *Label*, *Fname*, *Fext*, --, *DataType*, *InteriorKey*, *Append*

**Writes selected solution data to the PGR file for faster post processing access.**

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Label**

Switch that controls the PGR file write operations :

**Off**

Do not write the PGR file during the solution.

**On**

Write the PGR file during solution (default).

**STAT**

Display the current status of the PGR file settings.

**Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to *Jobname*, or *File* if no jobname is specified.

**Fext**

Filename extension (8 character maximum).

--

Unused field.

**DataType**

Data type to create. This specification applies to discontinuous results data such as stress, strain, and field data. .

**0**

Save as nodal-averaged data (default). Used by the **PLNSOL** command.

**1**

Reserved.

**2**

Save both nodal averaged and unaveraged data. Used by the **PLESOL** and **PLNSOL** commands.

**InteriorKey**

Key that controls whether or not internal model data is saved to the PGR file. Internal data is required in order to use graphical slicing, capping, vector displays, or Isosurface displays for data display. Interior data is also required for the **AVRES,,FULL** option, where surface results take into consideration the interior element contributions.

**0**

Save exterior surface data only. (default)

**1**

Save exterior and interior data.

**Append**

Append or overwrite data to the specified file. .

**0**

Append the data to the existing file (default). Note that the geometry must not have been changed between the write operations.

**1**

Overwrite the data file.

**Notes**

In interactive mode, you cannot read the PGR file while **PGWRITE** is active. If you solve your model and want to plot the model while still in **SOLUTION**, you must first close the PGR file by issuing **PGWRITE,Off**. This occurs automatically when you exit **SOLUTION** and enter **POST1**.

When you wish to view your stress results in another coordinate system, you must generate your PGR file from the results file, in **POST1**, in that coordinate system.

## Menu Paths

Main Menu>Solution>Output Ctrls>PGR File  
Main Menu>General Postproc>Write PGR File

**PHYSICS**, *Option*, *Title*, *Fname*, *Ext*, --

**Writes, reads, or lists all element information**

[PREP7: Special Purpose](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Option*

Specifies what to do with element information:

#### **WRITE**

Write all appropriate element types, key options, real constants, material properties, solution analysis options, load step options, constraint equations, coupled nodes, defined components, and GUI preference settings to the file specified with the *Fname* and *Ext* arguments.

#### **READ**

Deletes all solution information (material properties, solution options, load step options, constraint equations, coupled nodes, results, and GUI preference settings) then reads all the information listed above into the ANSYS database from the location specified by the *Fname* and *Ext* arguments.

#### **LIST**

Lists currently defined physics files and their titles.

#### **DELETE**

Deletes a specified physics file and its title from the database.

#### **CLEAR**

Deletes all material properties, solution options, load step options, constraint equations, coupled nodes, results, and GUI preference settings from the database. Does NOT clear the active physics file title from the database.

#### **STATUS**

Displays information about all active elements and settings.

### *Title*

A user-defined title that quickly identifies a set of physics settings. For example, you might use "Fluid," "Structural," or "Magnetic" as titles. A title can contain up to 64 characters. It can be entered in lower or upper case. Lower case is internally converted to upper case within the program.

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`. Previous data on this file, if any, are overwritten.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to `PH $n$`  if *Fname* is blank, where *n* is a number between one and nine, depending on how many times you have issued the **PHYSICS** command. (You can have up to nine separate physics



files.) If you issue the command more than nine times, the ANSYS program will require you to delete an existing file.

--

Unused field.

## Notes

Use the **PHYSICS** command when you are performing a multiphysics analysis that involves two different disciplines (for example, CFD and structural analysis) and you cannot solve both analyses simultaneously. Once you have set up physics environments for both analyses, you can use the **PHYSICS,READ** command to change between the defined physics environments. For more information about doing multiphysics analyses, see [Sequential Coupled-Field Analysis](#) in the *Coupled-Field Analysis Guide*.

The **PHYSICS** command outputs all solution information, including analysis options, to the `Jobname.PHn` file described above. Although it also outputs components, the ANSYS program does not list entities (nodes, elements, lines, etc.).

**PHYSICS,WRITE** will overwrite existing physics files with the same title (even if the name is different). In other words, if the directory has a physics file with the same title as the active physics file title, but a different name, the **PHYSICS,WRITE** command will overwrite the existing physics file and use the existing filename, not the filename specified on the **PHYSICS,WRITE** command.

## Menu Paths

**Main Menu>Preprocessor>Physics>Environment>Clear**  
**Main Menu>Preprocessor>Physics>Environment>Delete**  
**Main Menu>Preprocessor>Physics>Environment>List**  
**Main Menu>Preprocessor>Physics>Environment>Read**  
**Main Menu>Preprocessor>Physics>Environment>Status**  
**Main Menu>Preprocessor>Physics>Environment>Write**  
**Main Menu>Solution>Physics>Environment>Clear**  
**Main Menu>Solution>Physics>Environment>Delete**  
**Main Menu>Solution>Physics>Environment>List**  
**Main Menu>Solution>Physics>Environment>Read**  
**Main Menu>Solution>Physics>Environment>Status**  
**Main Menu>Solution>Physics>Environment>Write**

---

**/PICE**, *Item*, --, *KEY*

Shows initial conditions on elements as contours on displays.

[GRAPHICS: Labeling](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

*Item*

Label identifying the item:

**VFRC**

Volume fraction.

--

Unused field.

**KEY**

Symbol key:

- 0** Do not show initial condition contours.
- 1** Show initial condition contours.

**Command Default**

No initial condition contours displayed.

**Notes**

Shows initial conditions as contours on displays for the selected elements. Use **/PSTATUS** or **/PICE,STAT** to display settings. Use **/PICE,DEFA** to reset all specifications back to default.

**/PICE** is overridden if **/PBF**, **/PSF**, and **/PBC** are all on.

This command is valid in any processor.

**Menu Paths**

Utility Menu>PlotCtrls>Symbols

---

**PIVCHECK**, *KEY*, *PRNTCNTRL*

**Prevents a batch mode, linear static analysis from stopping when a negative or zero equation solver pivot value is encountered.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

**KEY**

Determines whether to stop or continue an analysis when a negative or zero equation solver pivot value is encountered:

**ON**

Default value. ANSYS checks for negative or zero pivot values on linear static analyses performed with the sparse and PCG solvers. When one is encountered, an error is issued, stopping the job. A negative pivot value may be valid for some multiphysics analyses (e.g. Electromagnetic or Thermal); this key has no effect in these cases.

**OFF**

Pivots will not be checked. This key continues the analysis in spite of a zero or negative pivot value. The program will proceed until other error checking routines are encountered.

**PRNTCNTRL**

Provides print options. Print output with these options will be sent to the default output file, not to the files created by the nonlinear diagnostic tools (**NLDIAG**).

**ONCE or blank**

Default value. Print maximum and minimum pivot information once per loadstep if the SPARSE solver (the default solver) is used.

**EVERY**

Print maximum and minimum pivot information every Newton-Raphson iteration or at every iteration within a substep of every loadstep. This option is provided for nonlinear diagnostics.

**Command Default**

ON, stop the analysis and report an error status to the user.

**Notes**

This command is valid only for linear static analyses. In a nonlinear analysis, a negative pivot may be valid. Normally, rigid body motions in a nonlinear analysis will be trapped by error routines checking infinitely large displacements (DOF limit exceeded) or nonconvergence status. An under-constrained model may avoid the pivot check, but fail with a DOF limit exceeded error.

This command is applicable only to batch mode (not in the GUI). While in an ANSYS interactive session, it is generally desirable to remain in the session, even though a pivot error has occurred. Warning messages will be displayed on the screen, allowing the user to take corrective action.

Finally, machine precision will determine if a warning or an error is issued for the small pivot case. In all cases there will be some notification that the model needs to be checked closely for an accurate solution.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>Analysis Options**

**PLCAMP**, *Option, SLOPE, UNIT, FREQB, Cname, STABVAL*

**Plots Campbell diagram data for applications involving rotating structure dynamics.**

POST1: Special Purpose

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**Option**

Flag to activate or deactivate sorting of forward or backward whirl frequencies:

**0 (OFF or NO)**

No sorting.

**1 (ON or YES)**

Sort. This value is the default.

**SLOPE**

The slope of the line to be plotted. This value must be greater than 0. The line represents the number of excitations per revolution of the rotor. For example, *SLOPE* = 1 represents one excitation per revolution, usually resulting from unbalance. The default is no line.

**UNIT**

Specifies the unit of measurement for rotational angular velocities:

**RDS**

Rotational angular velocities in radians per second (rad/s). This value is the default.

**RPM**

Rotational angular velocities in revolutions per minute (RPMs).

**FREQB**

The beginning, or lower end, of the frequency range of interest. The default is zero.

**Cname**

The rotating component name.

**STABVAL**

Flag to plot the stability values:

**0 (OFF or NO)**

Plot the frequencies (the imaginary parts of the eigenvalues in Hz). This value is the default.

**1 (ON or YES)**

Plot the stability values (the real parts of the eigenvalues in Hz).

**2**

Plot the logarithmic decrements.

For more information about complex eigenvalues and corresponding logarithmic decrements, see [Complex Eigensolutions](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

## Notes

To take the gyroscopic effect into account when plotting a Campbell diagram, first issue the **CORIOLIS** command in the SOLUTION module.

The **PLCAMP** command is valid only for modal analyses (QR damped [**MODOPT**,**QRDAMP**] or damped [**MODOPT**,**DAMP**] methods only). The command works over *two or more* load step results generated with an ascending order of rotational velocity (**OMEGA** or **CMOMEGA**).

Complex eigenvalues are necessary (**MODOPT**,**QRDAMP**,, ,*Cpxmod* = ON), and you must specify the number of modes to expand (**MXPAND**) at each load step.

In some cases where modes are not in the same order from one load step to the other, sorting (*Option* = 1) the frequencies can help to obtain a correct plot. Sorting is based on the comparison between complex mode shapes calculated at two successive load steps.

At each load step, ANSYS compares the mode shape to the loads at other loads steps to determine whirl direction at the load step. If applicable, a label appears (in the plot legend) representing each whirl mode (BW for backward whirl and FW for forward whirl).

ANSYS considers the *SLOPE* value only if the frequencies are plotted (*STABVAL* = OFF).

At each load step, ANSYS checks for instability (based on the sign of the real part of the eigenvalue). The labels "stable" or "unstable" appear in the plot legend for each frequency curve.

The rotational velocities of a named component (*Cname*) are displayed on the X-axis.

To plot the Campbell diagram for a prestressed structure, first issue a **CAMPBELL,ON** command in the static portion of the analysis.

In general, ANSYS, recommends plotting a Campbell diagram only when your analysis is performed in a *stationary* reference frame (**CORIOLIS**,,,,*RefFrame* = ON).

For a usage example of the **PLCAMP** command, see [Sample Campbell Diagram Analysis](#).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Rotor Dynamics>Plot Campbell

**PLCINT**, *ACTION*, *ID*, *Node*, *Cont*, *Dtype*

Plots the fracture parameter (**CINT**) result data.

[POST1:Results](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

*Action*

### **PATH**

Plots **CINT** quantities according to path number (default).

### **FRONT**

Plots **CINT** quantities distribution along the crack front.

*ID*

Crack ID number.

*Node*

Crack tip node number (default = ALL).

Use only for *ACTION* = PATH. Plots **CINT** contour for an individual crack tip node.

*Cont*

Contour number (Default = ALL).

Use only for *ACTION* = FRONT. Plots **CINT** distribution along the crack for a given path.

*Dtype*

Data type to output:

### **JINT**

J-integral (default)

### **IIN1**

Interaction integral 1

### **IIN2**

Interaction integral 2

### **IIN3**

Interaction integral 3

### **K1**

Mode 1 stress-intensity factor

**K2**

Mode 2 stress-intensity factor

**K3**

Mode 3 stress-intensity factor

**G1**

Mode 1 energy release rate

**G2**

Mode 2 energy release rate

**G3**

Mode 3 energy release rate

**GT**

Total energy release rate

## Menu Paths

**This command cannot be accessed from a menu.**

---

## PLCPLX, KEY

**Specifies the part of a complex variable to display.**

[POST26:Display](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### **KEY**

Complex variable part:

**0**

Amplitude.

**1**

Phase angle.

**2**

Real part.

**3**

Imaginary part.

## Notes

Used only with harmonic analyses ([ANTYPE,HARMIC](#)).

All results data are stored in the form of real and imaginary components and converted to amplitude and/or phase angle as specified via the **PLCPLX** command. The conversion is not valid for derived results (such as principal stress/strain, equivalent stress/strain and USUM).

## Menu Paths

**Main Menu>TimeHist Postpro>Settings>Graph**

---

**PLCRACK**, *LOC*, *NUM*

Displays cracking and crushing locations in **SOLID65** elements.

POST1: Special Purpose

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**LOC**

Location at which symbols are to be displayed:

- 0** Plot symbols at integration points (default).
- 1** Plot symbol at element centroids (averaged).

**NUM**

Crack to be displayed:

- 0** Plot all cracks (default).
- 1** Plot only the first crack.
- 2** Plot only the second crack.
- 3** Plot only the third crack.

**Notes**

**PLCRACK** displays circles at locations of cracking or crushing in concrete elements. Cracking is shown with a circle outline in the plane of the crack, and crushing is shown with an octahedron outline. If the crack has opened and then closed, the circle outline will have an X through it. Each integration point can crack in up to three different planes. The first crack at an integration point is shown with a red circle outline, the second crack with a green outline, and the third crack with a blue outline.

Symbols shown at the element centroid (*LOC* = 1) are based on the status of all of the element's integration points. If any integration point in the element has crushed, the crushed (octahedron) symbol is shown at the centroid. If any integration point has cracked or cracked and closed, the cracked symbol is shown at the element centroid. If at least five integration points have cracked and closed, the cracked and closed symbol is shown at the element centroid. Finally, if more than one integration point has cracked, the circle outline at the element centroid shows the average orientation of all cracked planes for that element.

Portions of this command are not supported by PowerGraphics [/GRAPHICS,POWER].

**Menu Paths**

**Main Menu>General Postproc>Plot Results>ConcPlot>Crack/Crush**

**PLDISP**, *KUND***Displays the displaced structure.**

POST1: Results

MP ME ST PR PRN DS DSS &lt;&gt; &lt;&gt; &lt;&gt; DY PP &lt;&gt; EME MFS

*KUND*

Undisplaced shape key:

- 0** Display only displaced structure.
- 1** Overlay displaced display with similar undisplaced display (appearance is system-dependent).
- 2** Same as 1 except overlay with undisplaced edge display (appearance is system-dependent).

**Notes**

Displays the displaced structure for the selected elements.

For information on true scale plots, refer to the description of the **/DSCALE** command [**/DSCALE**,,1.0].**Menu Paths**

**Main Menu>General Postproc>Plot Results>Deformed Shape**  
**Utility Menu>Plot>Results>Deformed Shape**  
**Utility Menu>PlotCtrls>Animate>Deformed Shape**

**PLESOL**, *Item, Comp, KUND, Fact***Displays the solution results as discontinuous element contours.**

POST1: Results

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

*Item*Label identifying the item. Valid item labels are shown in [Table 240: PLESOL - Valid Item and Component Labels \(p. 1269\)](#) below. Some items also require a component label.*Comp*Component of the item (if required). Valid component labels are shown in [Table 240: PLESOL - Valid Item and Component Labels \(p. 1269\)](#) below.*KUND*

Undisplaced shape key:

- 0** Do not overlay undeformed structure display
- 1** Overlay displaced contour plot with undeformed display (appearance is system-dependent)



## 2

Overlay displaced contour plot with undeformed edge display (appearance is system-dependent)

**Fact**

Scale factor for 2-D display of contact items (defaults to 1). A negative scaling factor may be used to invert the display.

**Notes**

Displays the solution results as element contours discontinuous across element boundaries for the selected elements. For example, **PLESOL,S,X** displays the X component of stress S (that is, the SX stress component). Various element results depend on the calculation method and the selected results location (**AVPRIN**, **RSYS**, and **ESEL**). Contours are determined by linear interpolation within each element, unaffected by the surrounding elements (i.e., no nodal averaging is performed). The discontinuity between contours of adjacent elements is an indication of the gradient across elements. Component results are displayed in the active results coordinate system [**RSYS**] (default is the global Cartesian). See the **ETABLE** and **PLETAB** commands for displaying items not available through this command (such as line element results).

For PowerGraphics displays [**GRAPHICS**,POWER], results are plotted only for the model exterior surface. The items marked with [1 (p. 1273)] in *Table 240: PLESOL - Valid Item and Component Labels* (p. 1269) are not supported by PowerGraphics.

**Table 240 PLESOL - Valid Item and Component Labels**

Item	Comp	Description
Valid item and component labels for element results are:		
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.

Item	Comp	Description
EPSW		Swelling strain.
EPTO	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total mechanical strain.
"	INT	Total mechanical strain intensity.
"	EQV	Total mechanical equivalent strain.
EPTT	X, Y, Z, XY, YZ, XZ	Total mechanical <i>and</i> thermal strain (EPEL + EPPL + EPCR + EPTH).
"	1, 2, 3	Principal total mechanical <i>and</i> thermal strain.
"	INT	Total mechanical <i>and</i> thermal strain intensity.
"	EQV	Total mechanical <i>and</i> thermal equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
CDM	DMG	Damage variable
"	LM	Maximum previous strain energy for virgin material
FAIL	MAX	Maximum of all active failure criteria defined at the current location (See the <b>FCTYP</b> command for details.) [1][3]
"	EMAX	Maximum Strain Failure Criterion. [1][3]
"	SMAX	Maximum Stress Failure Criterion. [1][3]
"	TWSI	Tsai-Wu Strength Index Failure Criterion. [1][3]
"	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion. [1][3]
"	HFIB	Hashin Fiber Failure Criterion. [1][3][5]
"	HMAT	Hashin Matrix Failure Criterion. [1][3][5]
"	PFIB	Puck Fiber Failure Criterion. [1][3][5]
"	PMAT	Puck Matrix Failure Criterion. [1][3][5]
"	USR1, USR2, ..., USR9	User-defined failure criteria. [1][3][4][5]
FCMX	LAY	Layer number where the maximum of all active failure criteria over the entire element occurs. [1][3]
"	FC	Number of the maximum-failure criterion over the entire element [1][3] : 1 - EMAX 2 - SMAX 3 - TWSI 4 - TWSR 5 - HFIB6 - HMAT7 - PFIB8 - PMAT9~17 - USR1~USR9

Item	Comp	Description
"	VAL	Value of the maximum failure criterion over the entire element. [1][3]
SVAR	1, 2, 3, ... N	State variable. [1]
GKS	X, XY, XZ	Gasket component stress.
GKD	X, XY, XZ	Gasket component total closure.
GKDI	X, XY, XZ	Gasket component total inelastic closure.
GKTH	X, XY, XZ	Gasket component thermal closure.
SS	X, XY, XZ	Interface traction (stress)
SD	X, XY, XZ	Interface separation
CONT	STAT	Contact status[2]:  3-closed and sticking 2-closed and sliding 1-open but near contact 0-open and not near contact
"	PENE	Contact penetration
"	PRES	Contact pressure
"	SFRIC	Contact friction stress
"	STOT	Contact total stress (pressure plus friction)
"	SLIDE	Contact sliding distance
"	GAP	Contact gap distance
"	FLUX	Total heat flux at contact surface
"	CNOS	Total number of contact status changes during substep.
"	FPRS	Fluid penetration pressure
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component electromagnetic force or vector sum [1].
P	X, Y, Z, SUM	Pointing vector component or sum [1].
SERR[6]		Structural error energy [1].
SDSG[6]		Absolute value of the maximum variation of any nodal stress component [1].
TERR[6]		Thermal error energy [1].
TDSG[6]		Absolute value of the maximum variation of any nodal thermal gradient component [1].
F	X, Y, Z	X, Y, or Z structural force [1][7].

Item	Comp	Description
M	X, Y, Z	X, Y, or Z structural moment [1].
HEAT		Heat flow [1].
FLOW		Fluid flow [1].
AMPS		Current flow [1]. Use <b>FORCE</b> for type.
CHRG		Charge [1]. Use <b>FORCE</b> for type.
FLUX		Magnetic flux [1].
CSG	X, Y, Z	X, Y, or Z magnetic current segment component [1].
SENE		"Stiffness" energy or thermal heat dissipation. Same as TENE[1].
STEN		Elemental energy dissipation due to <b>stabilization</b> [1].
TENE		Thermal heat dissipation or "stiffness" energy. Same as SENE [1].
KENE		Kinetic energy [1].
JHEAT		Element Joule heat generation [1].
JS	X, Y, Z, SUM	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM). [1].
JT	X, Y, Z, SUM	Total measurable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM). [1].
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [1].
MRE		Magnetic Reynolds number [1].
VOLU		Volume of volume element [1].
CENT	X, Y, Z	Centroid X, Y, or Z location (based on shape function) in the active coordinate system [1].
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
SMISC	<i>snum</i>	Element summable miscellaneous data value at sequence number <i>snum</i> (shown in the Output Data section of each element description found in the <a href="#">Element Reference</a> ).
NMISC	<i>snum</i>	Element non-summable miscellaneous data value at sequence number <i>snum</i> (shown in the Output Data section of each element description found in the <a href="#">Element Reference</a> ).
TOPO		Densities used for topological optimization.
CAP	C0,X0,K0,ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; l1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2:

Item	Comp	Description
		shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.
EDPC	CSIG,CSTR	Material EDP creep model only (not including the cap model): Equivalent creep stress; equivalent creep strain.
FICT	TEMP	Fictive temperature.
ESIG	X,Y,Z,XY,YZ,ZX	Components of Biot's effective stress.
"	1, 2, 3	Principal stresses of Biot's effective stress.
"	INT	Stress intensity of Biot's effective stress.
"	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
"	GPOR	Porosity due to void growth.
"	NPOR	Porosity due to void nucleation.
FFLX	X,Y,Z	Fluid flow flux in poromechanics.

1. Not supported by PowerGraphics
2. For MPC-based contact definitions, the value of STAT can be negative. This indicates that one or more contact constraints were intentionally removed to prevent overconstraint. STAT = -3 is used for MPC bonded contact; STAT = -2 is used for MPC no-separation contact.
3. Works only if failure criteria information is provided. (For more information, see the documentation for the **FC** and **TB** commands.)
4. Works only if user-defined failure criteria routine is provided.
5. Must be added via the **FCTYP** command first.
6. Some element- and material-type limitations apply. (For more information, see the documentation for the **PRERR** command.)
7. Do not use the **PLESOL** command to obtain contact forces for contact elements. The force values reported by this command may not be accurate for these elements. Instead, use the **ETABLE** command to obtain contact force values.

## Menu Paths

**Main Menu>General Postproc>Plot Results>Contour Plot>Element Solu**  
**Utility Menu>Plot>Results>Contour Plot>Elem Solution**

**PLETAB**, *Itlab*, *Avglab*

**Displays element table items.**

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Itlab*

User-defined label, as specified with the **ETABLE** command, of item to be displayed.

**AvgLab**

Averaging operation:

**NOAV**

Do not average element items at common nodes (default).

**AVG**

Average the element items at common nodes.

## Notes

Displays items stored in the table defined with the **ETABLE** command for the selected elements. For display purposes, items are assumed to be constant over the element and assigned to each of its nodes. Contour display lines (lines of constant value) are determined by linear interpolation within each element from the nodal values. These nodal values have the option of being averaged (values are averaged at a node whenever two or more elements connect to the same node) or not averaged (discontinuous). The discontinuity between contour lines of adjacent elements is an indication of the gradient across elements.

Portions of this command are not supported by PowerGraphics [/GRAPHICS,POWER].

## Menu Paths

**Main Menu>General Postproc>Element Table>Plot Elem Table**  
**Main Menu>General Postproc>Plot Results>Contour Plot>Elem Table**  
**Utility Menu>Plot>Results>Contour Plot>Elem Table Data**

---

**PLFAR**, *Lab*, *Opt*, *PHI1*, *PHI2*, *NPH1*, *THETA1*, *THETA2*, *NTHETA*, *VAL1*, *VAL2*

**Plots electric far fields and far field parameters.**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**Lab**

Plot Parameter:

**EF**

Electric Field

**RCS**

Radar cross section

**RCSN**

Normalized radar cross section

**ANT**

Antenna parameters

**Opt**

As shown below, data entered in the Opt field will vary, depending on the print parameter.

**Lab**

As shown below, data entered in the Lab field will vary, depending on the print option (Opt).

**Table 241 Valid Lab Data Labels**

<i>Lab</i>	<i>Opt</i>
EF	<p>SUM — the maximum total rE-field (default)</p> <p>X — the maximum rE-field in x-direction</p> <p>Y — the maximum rE-field in y-direction</p> <p>Z — the maximum rE-field in z-direction</p> <p>PHI — the maximum rE-field in <math>\Phi</math>-direction</p> <p>THETA — the maximum rE-field in <math>\Theta</math>-direction</p> <p>R — the maximum rE-field in r-direction</p> <p>LHCP — the maximum left-hand circularly polarized component</p> <p>RHCP — the maximum right-hand circularly polarized component</p> <p>ARCP — the circular polarization axial ratio</p> <p>LW3X — the maximum dominant component for an x-polarized aperture with Ludwig's third definition of cross definition</p> <p>LW3Y — the maximum dominant component for an y-polarized aperture with Ludwig's third definition of cross definition</p> <p>ARCP — the linear polarization axial ratio</p>
RCS and RCSN	<p>TOTAL — Radar echo area (default)</p> <p>PP — <math>\Phi</math>-<math>\Phi</math> polarization</p> <p>PT — <math>\Phi</math>-<math>\Theta</math> polarization</p> <p>TP — <math>\Theta</math>-<math>\Phi</math> polarization</p> <p>TT — <math>\Theta</math>-<math>\Theta</math> polarization</p> <p>HH — H-H (horizontal – horizontal) polarization</p> <p>HV — H-V (Horizontal – Vertical) polarization</p> <p>VH — V-H (Vertical – Horizontal) polarization</p> <p>VV — V-V (Vertical – Vertical) polarization</p>

<i>Lab</i>	<i>Opt</i>
	TE2D — 2-D TE incident plane wave TM2D — 2-D TM incident plane wave
ANT	EPCT – radiation pattern (default) EPPL – radiation pattern in polar coordinate system plot UMAX – the maximum radiation intensity DGCT – directivity in Cartesian coordinate system plot DGPL – directivity in polar coordinate system plot DGMX – peak directivity PGCT – gain in Cartesian coordinate system plot PGPL – gain in polar coordinate system plot PGMX – peak gain RGCT –realized gain in Cartesian coordinate system plot RGPL –realized gain in polar coordinate system plot RGMX – peak realized gain EFF – efficiency PRAD – radiated power

The following PHI1, PHI2, NPFI, THETA1, THETA2, and NTHETA arguments are used only with Lab = EF, RCS, RCSN and ANT and Opt = EPCT, EPPL, DGCT, DGPL, PGCT, PGPL, RGCT and RGPL.

**PHI1, PHI2**

Starting and ending  $\phi$  angles (degrees) in the spherical coordinate system. Defaults to 0.

**NPFI**

Number of divisions between the starting and ending  $\phi$  angles for data computations. Defaults to 0.

**THETA1, THETA2**

Starting and ending  $\theta$  angles (degrees) in the spherical coordinate system. Defaults to 0.

**NTHETA**

Number of divisions between the starting and ending  $\theta$  angles for data computations. Defaults to 0.

**VAL1**

Used only with Lab = ANT and Opt = PGCT, PGPL, PGMX, RGCT, RGPL, RGMX, EFF or Lab = RCS or RCSN and Opt = TE2D, TM2D. For Lab = ANT and Opt = PGCT, PGMX, RGCT, RGMX, EFF, VAL1 = the incident power, Pinc, in watts. For Lab = RCS or RCSN and Opt = TE2D or TM2D, VAL1 = thickness of the 3-D model in the z direction, defaults to 1.



**VAL2**

Used only with *Lab* = ANT and *Opt* = PGCT, PGPL, PGMX, EFF. VAL2 = the  $|s_{11}|$  of the lossless port with a single propagation mode to calculate the power delivered to the radiating antenna structure, i.e.

$$P_{\text{del}} = P_{\text{inc}}(1 - |s_{11}|^2). \text{ Defaults to 0.}$$

**Notes**

**PLFAR** plots electric far fields and far field parameters as determined by the equivalent source principle. Use this command to plot electric far field, radar cross section, antenna parameters.

See [Spherical Coordinates in Hi-Frequency Electromagnetic Analysis Guide](#).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

This command cannot be accessed from a menu.

**PLFSS**, *Filename*, *Ext*, *Opt*, *Lab1*, *Lab2*

**Plots reflection and transmission parameters of a frequency selective surface solution.**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**Filename**

Name of the results file generated by the **SPFSS** command.

**Ext**

Filename extension (8 character maximum).

**Opt**

Format of plotted parameter:

**MAG**

Magnitude (default)

**ANG**

Angle

**DB**

Magnitude in dB units

**Lab1, Lab2**

Labels for reflection and transmission parameters:

**REFL**

Reflection coefficient.

**TRAN**

Transmission coefficient.

**PREFL**

Power reflection coefficient.

**PTRAN**

Power transmission coefficient.

**IL**

Insertion loss (dB only).

**RL**

Return loss (dB only).

## Notes

Plots reflection and transmission parameters results of the **SPFSS** command.

If *Lab2* is the same as *Lab1* or *blank*, only one plot is generated.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

---

### **PLF2D**, *NCONT*, *OLAY*, *ANUM*, *WIN*

**Generates a contour line plot of equipotentials.**

[POST1: Magnetics Calculations](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

***NCONT***

Number of contour lines to display. Issue in multiples of 9 (i.e., 9, 18, 27, etc.). Default is 27 contour lines.

***OLAY***

Overlay:

**0**

Overlay edge outlines by material number.

**1**

Overlay edge outlines by real constant number.

***ANUM***

Highest material or real constant attribute number. Command will cycle through *ANUM* element display overlays. Defaults to 10.

***WIN***

Window number to which command applies. Defaults to 1.

## Notes

**PLF2D** invokes an ANSYS macro which plots equipotentials of the degree of freedom AZ. These equipotential lines are parallel to flux lines and thus give a good representation of flux patterns. In the axisymmetric case, the display is actually  $r \cdot AZ$  where "r" is the node radius. The macro overlays (*OLAY*) edge outlines by material number or real constant number (*ANUM*) and allows user control over the number of contour lines to display (*NCONT*).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>General Postproc>Plot Results>Contour Plot>2D Flux Lines**  
**Utility Menu>Plot>Results>Flux Lines**

---

**PLLS**, *LabI*, *LabJ*, *Fact*, *KUND*

**Displays element table items as contoured areas along elements.**

POST1: Element Table

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### *LabI*

Label of element table item [**ETABLE**] for node I magnitude.

### *LabJ*

Label of element table item for node J magnitude.

### *Fact*

Scale factor for display (defaults to 1). A negative scaling factor may be used to invert the display.

### *KUND*

Undisplaced shape key:

**0**

Display selected items on undeformed shape.

**1**

Display selected items on deformed shape.

## Notes

Displays selected items (e.g., shears and moments) as a contoured area (trapezoid) display along line elements and 2-D axisymmetric shell elements (e.g., shear and moment diagrams). Three sides of the trapezoid are formed by the element (one side) and lines at nodes I and J of length proportional to the item magnitude and displayed normal to the element and the viewing direction (the two parallel sides).

Portions of this command are not supported by PowerGraphics [**/GRAPHICS,POWER**].

## Menu Paths

**Main Menu>General Postproc>Plot Results>Contour Plot>Line Elem Res**

---

**PLNEAR**, *Lab*, *Opt*, *KCN*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*

**Plots the electric field in the near zone exterior to the equivalent source surface.**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

***Lab***

Print the maximum E-file:

**SPHERE**

on the spherical structure

**PATH**

along the path

***Opt***

SUM — the maximum total E-field (default)

X — the maximum E-field in x-direction

Y — the maximum E-field in y-direction

Z — the maximum E-field in z-direction

PHI — the maximum E-field in  $\Phi$ -direction

THETA — the maximum E-field in  $\Theta$ -direction

R — the maximum E-field in r-direction

LHCP — the maximum left-hand circularly polarized component

RHCP — the maximum right-hand circularly polarized component

ARCP — the circular polarization axial ratio

LW3X — the maximum dominant component for an x-polarized aperture with Ludwig's third definition of cross definition

LW3Y — the maximum dominant component for an y-polarized aperture with Ludwig's third definition of cross definition

ARCP — the linear polarization axial ratio

***KCN***

*KCN* is the coordinate system reference number. It may be 0 (Cartesian) or any previously defined local coordinate system number (>10). Defaults to 0.

***VAL1*, *VAL2*, *VAL3*, . . . , *VAL7***

For *LAB* = SPHERE:

***VAL1***

the radius of spherical surface in spherical coordinate system.

***VAL2***

starting  $\phi$  angle (degree) in the spherical coordinate system. Defaults to 0.

***VAL3***

ending  $\phi$  angle (degree) in the spherical coordinate system. Defaults to 0.

***VAL4***

Number of divisions between the starting and ending  $\phi$  angles for data computations. Defaults to 0.

***VAL5***

starting  $\theta$  angle (degree) in the spherical coordinate system. Defaults to 0.

**VAL6**

ending  $\theta$  angle (degree) in the spherical coordinate system. Defaults to 0.

**VAL7**

Number of divisions between the starting and ending  $\theta$  angles for data computations. Defaults to 0.

For  $Lab = \mathbf{PATH, PLNEAR}$  computes the electric field for the path data points for the path currently defined by the **PATH** and **PPATH** commands.

## Notes

**PLNEAR** uses the equivalent source principle to calculate the electric field in the near zone exterior to the equivalent source surface (flagged with the Maxwell surface flag in the preprocessor) for one of the following locations:

- A spherical surface in the KAN coordinate system
- A path defined by the **PATH** and **PPATH** commands

To plot the electromagnetic field results for a path, use the **PLPAGM** or **PLPATH** commands.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**PLNSOL**, *Item*, *Comp*, *KUND*, *Fact*, *FileID*

**Displays results as continuous contours.**

POST1:Results

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Item**

Label identifying the item. Valid item labels are shown in [Table 242: PLNSOL - Valid Item and Component Labels](#) (p. 1282) below. Some items also require a component label.

**Comp**

Component of the item (if required). Valid component labels are shown in [Table 242: PLNSOL - Valid Item and Component Labels](#) (p. 1282) below.

**KUND**

Undisplaced shape key:

**0**

Do not overlay undeformed structure display

**1**

Overlay displaced contour plot with undeformed display (appearance is system-dependent)

**2**

Overlay displaced contour plot with undeformed edge display (appearance is system-dependent)

**Fact**

Scale factor for 2-D display for contact items. Default value is 1. A negative scaling factor may be used to invert the display.

**FileID**

The file index number (obtained via the **NLDIAG,NRRE,ON** command). Valid only for *Item* = NRRE.

**Notes**

Displays the solution results as continuous contours across element boundaries for the selected nodes and elements. For example, **PLNSOL,S,X** displays the X component of stress S (that is, the SX stress component). Various element results depend upon the recalculation method and the selected results location [**AVPRIN**, **RSYS**, **LAYER**, **SHELL**, and **NSEL**]. Contours are determined by linear interpolation within each element from the nodal values, which are averaged at a node whenever two or more elements connect to the same node (except for FMAG, which is summed at the node).

For PowerGraphics displays [**/GRAPHICS,POWER**], results are plotted only for the model exterior surface. The items marked with [2 (p. 1286)] are not supported by PowerGraphics. To plot midside nodes, you must first issue **/EFACET,2**.

**Table 242 PLNSOL - Valid Item and Component Labels**

Item	Comp	Description
<b>Valid item and component labels for nodal degree of freedom results are:</b>		
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum
TEMP[1]		Temperature
PRES		Pressure
VOLT		Electric potential
MAG		Magnetic scalar potential
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum in a fluid analysis, or X, Y, or Z velocity or vector sum in an ANSYS LS-DYNA analysis.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum in an electromagnetic analysis, or X, Y, or Z acceleration or vector sum in an ANSYS LS-DYNA analysis.
VEL	X, Y, Z, SUM	X, Y, or Z velocity or vector sum in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
ACC	X, Y, Z, SUM	X, Y, or Z acceleration or vector sum in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
OMG	X, Y, Z, SUM	X, Y, or Z rotational velocity or vector sum in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
DMG	X, Y, Z, SUM	X, Y, or Z rotational acceleration or vector sum in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
ENKE		Turbulent kinetic energy (FLOTRAN)
ENDS		Turbulent energy dissipation (FLOTRAN)

Item	Comp	Description
SP0 $n$		Mass fraction of species $n$ , where $n = 1$ to 6 (FLOTRAN). If a species is given a user-defined name [ <b>MSSPEC</b> ], use that name instead of $n$ .
WARP		Warping
NRRE	FX, FY, FZ, FNRM, MX, MY, MZ, MNRM	Plot the Newton-Raphson residuals from the file you obtained via the <b>NLDIAG,NRRE,ON</b> command. The FNRM and MNRM labels are computed as the square root of the sum of the squares of the residual component forces or moments (FX,FY,FZ, MX, MY, MZ). [5]  When KUND = 0, use the absolute value of the residual from the files (default).

**Valid item and component labels for element results are:**

S	X, Y, Z, XY, YZ, XZ	Component stress
"	1, 2, 3	Principal stress
"	INT	Stress intensity
"	EQV	Equivalent stress
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain
"	1, 2, 3	Principal elastic strain
"	INT	Elastic strain intensity
"	EQV	Elastic equivalent strain
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain
"	1, 2, 3	Principal thermal strain
"	INT	Thermal strain intensity
"	EQV	Thermal equivalent strain
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain
"	1, 2, 3	Principal plastic strain
"	INT	Plastic strain intensity
"	EQV	Plastic equivalent strain
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain
"	1, 2, 3	Principal creep strain
"	INT	Creep strain intensity
"	EQV	Creep equivalent strain
EPSW		Swelling strain
EPTO	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (EPEL + EPPL + EPCR)
"	1, 2, 3	Principal total mechanical strain
"	INT	Total mechanical strain intensity
"	EQV	Total mechanical equivalent strain
EPTT	X, Y, Z, XY, YZ, XZ	Component total mechanical <i>and</i> thermal strain (EPEL + EPPL + EPCR + EPTH)
"	1, 2, 3	Principal total mechanical <i>and</i> thermal strain

Item	Comp	Description
"	INT	Total mechanical <i>and</i> thermal strain intensity
"	EQV	Total mechanical <i>and</i> thermal equivalent strain
ESIG	X,Y,Z,XY,YZ,ZX	Components of Biot's effective stress.
"	1, 2, 3	Principal stresses of Biot's effective stress.
"	INT	Stress intensity of Biot's effective stress.
"	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
"	GPOR	Porosity due to void growth.
"	NPOR	Porosity due to void nucleation.
NL	SEPL	Equivalent stress (from stress-strain curve)
"	SRAT	Stress state ratio
"	HPRES	Hydrostatic pressure
"	EPEQ	Accumulated equivalent plastic strain
"	CREQ	Accumulated equivalent creep strain
"	PSV	Plastic state variable
"	PLWK	Plastic work/volume
SEND	ELASTIC	Elastic strain energy density
"	PLASTIC	Plastic strain energy density
"	CREEP	Creep strain energy density
CDM	DMG	Damage variable
"	LM	Maximum previous strain energy for virgin material
FAIL	MAX	Maximum of all active failure criteria defined at the current location. (See the <b>FCTYP</b> command for details.) [2][6]
"	EMAX	Maximum Strain Failure Criterion [2][6]
"	SMAX	Maximum Stress Failure Criterion [2][6]
"	TWSI	Tsai-Wu Strength Index Failure Criterion [2][6]
"	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion [2][6]
"	HFIB	Hashin Fiber Failure Criterion. [2][6][8]
"	HMAT	Hashin Matrix Failure Criterion. [2][6][8]
"	PFIB	Puck Fiber Failure Criterion. [2][6][8]
"	PMAT	Puck Matrix Failure Criterion. [2][6][8]
"	USR1, USR2, ..., USR9	User-defined failure criteria [2][6][7][8]
SVAR	1, 2, 3, ... N	State variable [2]
GKS	X, XY, XZ	Gasket component stress
GKD	X, XY, XZ	Gasket component total closure
GKDI	X, XY, XZ	Gasket component total inelastic closure
GKTH	X, XY, XZ	Gasket component thermal closure
SS	X, XY, XZ	Interface traction (stress)



Item	Comp	Description
SD	X, XY, XZ	Interface separation
FICT	TEMP	Fictive temperature
<b>For contact results PowerGraphics is applicable for 3-D models only.</b>		
CONT[3]	STAT	Contact status[4]: 3-closed and sticking 2-closed and sliding 1-open but near contact 0-open and not near contact
"	PENE	Contact penetration
"	PRES	Contact pressure
"	SFRIC	Contact friction stress
"	STOT	Contact total stress (pressure plus friction)
"	SLIDE	Contact sliding distance
"	GAP	Contact gap distance
"	FLUX	Total heat flux at contact surface
"	CNOS	Total number of contact status changes during substep.
"	FPRS	Fluid penetration pressure
TG	X, Y, Z, SUM	Component thermal gradient or vector sum
TF	X, Y, Z, SUM	Component thermal flux or vector sum
PG	X, Y, Z, SUM	Component pressure gradient or vector sum
EF	X, Y, Z, SUM	Component electric field or vector sum
D	X, Y, Z, SUM	Component electric flux density or vector sum
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum
B	X, Y, Z, SUM	Component magnetic flux density or vector sum
FMAG	X, Y, Z, SUM	Component electromagnetic force or vector sum [2].
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [2].
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
TOPO		Densities used for topological optimization.
<b>Valid item labels for FLOTRAN nodal results are:</b>		
TTOT		Total temperature
HFLU		Heat flux
HFLM		Heat transfer (film) coefficient
COND		Fluid laminar conductivity
PCOE		Pressure coefficient
PTOT		Total (stagnation) pressure
MACH		Mach number

Item	Comp	Description
STRM		Stream function (2-D applications only)
DENS		Fluid density
VISC		Fluid laminar viscosity
SPHT		Specific heat [2]
EVIS		Fluid effective viscosity
CMUV		Turbulent viscosity coefficient
ECON		Fluid effective conductivity
YPLU		Y+, a turbulent law of the wall parameter
TAUW		Shear stress at the wall
SFTS		Surface tension coefficient
LMD $n$		Laminar mass diffusion coefficient for species $n$ , where $n$ = 1 to 6
EMD $n$		Effective mass diffusion coefficient for species $n$ , where $n$ = 1 to 6
RDFL		Radiation heat flux [2]

1. For [SHELL131](#) and [SHELL132](#) elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . . , TTOP instead of TEMP to view the individual temperature degree of freedom. When other thermal elements are included in the model, they should be unselected to avoid plotting undefined information. To view all temperatures in the same plot, set [/ESHAPE,1](#) and [/GRAPHICS,POWER](#) and issue **PLNSOL,TEMP**.
2. Not supported by PowerGraphics
3. For the CONT items for elements [CONTA171](#) through [CONTA177](#), the reported data is averaged across the element. To obtain a more meaningful STAT value, use the **PLESOL** command.
4. For MPC-based contact definitions, the value of STAT can be negative. This indicates that one or more contact constraints were intentionally removed to prevent overconstraint. STAT = -3 is used for MPC bonded contact; STAT = -2 is used for MPC no-separation contact.
5. When plotting Newton-Raphson residual items ( $Item = NRRE$ ) from a file on the deformed geometry, the displacements are based on the current set of results in the database. These displacements may not correspond to the loadstep and substep in the `.nrxxxxxx` file. (For more information about `.nrxxxxxx` files and nonlinear diagnostics postprocessing, see the description of the **NLDPOST** command and [Performing Nonlinear Diagnostics](#).)
6. Works only if failure criteria information is provided. (For more information, see the documentation for the **FC** and **TB** commands.)
7. Works only if user failure criteria routine is provided.
8. Must be added via the **FCTYP** command first.

## Menu Paths

**Main Menu>Drop Test>Animate Results**

**Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu**

**Utility Menu>Plot>Results>Contour Plot>Nodal Solution**

**Utility Menu>PlotCtrls>Animate>Animate Over Results**

**Utility Menu>PlotCtrls>Animate>Animate Over Time**

**Utility Menu>PlotCtrls>Animate>Deformed Results**  
**Utility Menu>PlotCtrls>Animate>Dynamic Results**  
**Utility Menu>PlotCtrls>Animate>Isosurfaces**  
**Utility Menu>PlotCtrls>Animate>Mode Shape**  
**Utility Menu>PlotCtrls>Animate>Q-Slice Contours**  
**Utility Menu>PlotCtrls>Animate>Time-harmonic**

**/PLOPTS**, *Label, KEY*

**Controls graphics options on subsequent displays.**

GRAPHICS:Labeling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Label*

Apply display items as selected from the following labels:

#### **INFO**

Controls the display of the legend (ON or OFF) and allows the choice of preset or Multi-legend placement. Control is provided by the *KEY* values. (Defaults to *KEY*=3 when the GUI is on. Defaults to *KEY*= 2 otherwise.)

#### **LEG1**

Header portion of legend column (defaults to ON).

#### **LEG2**

View portion of legend column (defaults to ON (except off with contour displays)).

#### **LEG3**

View the contour section of the legend column (defaults to ON).

#### **FRAME**

Frame border lines around windows (defaults to ON).

#### **TITLE**

Title (bottom left text) (defaults to ON).

#### **MINM**

Min-Max symbols on contour displays (defaults to ON).

#### **LOGO**

ANSYS logo (defaults to OFF (displayed as text at top of legend column)). If *KEY* = ON, the text is removed from legend column but the logo symbol is displayed in whichever active window is either in the uppermost right corner and on top, or if there is no window in that location, then in the window to the furthest right of the screen. Version information remains in the legend column.

#### **WINS**

Controls whether graphics windows automatically stretch or shrink to adjust to screen size as the legend column is turned off or on [**/PLOPTS,INFO**] (defaults to ON). If *WINS* is on and the legend column is changed from off to on, all windows are shrunk regardless of what their correct size is.

#### **WP**

Working plane (defaults to OFF). The working plane is drawn as part of the display (*not* just an overlaid image as in **WPSTYL**). This option is best used in combination with a hidden-line technique [**/TYPE**].

## DATE

Controls the display of the date and time in your legend. Subsequent *KEY* values control the display as follows:

- Off or 0 - No date or time displays are included in your legend.
- 1 - Only the date is shown.
- 2 (default) - both the date and time are shown.

## FILE

Controls the display of the ANSYS jobname in your legend. Subsequent *KEY* values control the display as follows:

- Off or 0 (default) - The ANSYS jobname is NOT included in your legend.
- On or 1 - The ANSYS jobname is included in your legend.

## KEY

Switch:

### OFF or 0

Do not apply this display item. For *Label* = DATE, no time or date are displayed.

### ON or 1

Apply this display item. For *Label* = DATE, show only the date.

### AUTO or 2

For *Label* = INFO, initiate Auto-legend mode. If the display has contours, the legend is ON; if the display has no contours, the legend is OFF. For *Label* = DATE, display both the date and time.

### 3

For *Label* = INFO, switch to Multi-legend mode. See the **/UDOC** command for the available legend configurations.

## Command Default

See individual label defaults.

The Multi-legend mode (**/PLOPTS,INFO,3**) is the default for contour legend displays.

## Notes

Use **/PLOPTS,STAT** to display settings. Use **/PLOPTS,DEFA** to reset all specifications back to their defaults.

When you perform multiple results displays, contours on the legend column may be truncated. To avoid this, specify **/PLOPTS,LEG1,0**.

The Multi-legend mode provides a number of legend data item priority and placement options. These options are accessed via the GUI at **Utility Menu > PlotCtrls > Window Controls > Window Options**. The **/UDOC** command provides command line options for this capability.

This command is valid in any processor.

This command is not available for Academic Research or Teaching level products

## Menu Paths

**Utility Menu > PlotCtrls > Window Controls > Reset Window Options**

---

**Utility Menu>PlotCtrls>Window Controls>Window Options**


---

**PLOB**

**Displays the orbital motion of a rotating structure**

POST1:Results

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

**Notes**

When a structure is rotating and the Coriolis or gyroscopic effect is taken into account (**CORIOLIS**), nodes lying on the rotation axis generally exhibit an elliptical orbital motion. The **PLOB** command displays the orbit of each rotating node as well as the deformed shape at time  $t = 0$  (the real part of the solution).

To *print* the characteristics of the orbital path traversed by each node, issue the **PROB** command.

The **PLOB** command is valid for line elements (such as **BEAM188**, **BEAM189**, **PIPE288**, and **PIPE289**).

Your model must also involve a rotational velocity (**OMEGA** or **CMOMEGA**) with Coriolis enabled in a *stationary* reference frame (**CORIOLIS**,,,,*RefFrame* = ON).

A **SET** command should be issued after **PLOB** to ensure proper output for subsequent postprocessing commands.

The coordinate system for displaying nodal results must be global Cartesian (**RSYS**,*KCN* = 0).

**Menu Paths**

**Main Menu>General Postproc>Rotor Dynamics>Plot orbit**

---

**PLOT**, *NSTRT*, *NEND*, *NINC*

**Forms a display.**

DISPLAY:Action

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NSTRT*, *NEND*, *NINC***

Display plots sequentially from number *NSTRT* to *NEND* in steps of *NINC*. *NSTRT* defaults to the next plot. *NEND* defaults to *NSTRT*. *NINC* defaults to 1. If *NSTRT* = ALL, display all plots from the beginning of the file. If *NEND* = ALL, display to the end of the file.

**Notes**

Output will be to the terminal or to a file, depending on the driver [**/SHOWDISP**]. The INTERLEAF and DUMP drivers produce an output file for each plot named INTL $nn$  and DUMP $nn$ , with  $nn$  sequentially ranging from 00 to 99. A blank line after the **PLOT** command causes the next plot to be formed.

## Menu Paths

It is part of the **DISPLAY** command.

---

## PLOTTING

Specifies "Plotting settings" as the subsequent status topic.

[POST26: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>TimeHist Postproc>Plot**

---

**PLPAGM**, *Item*, *Gscale*, *Nopt*

Displays path items along the path geometry.

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Item*

The path data item to be displayed on the currently active path (defined by the **PATH** command). Valid path items are those defined with the **PDEF** or **PLNEAR** commands.

### *Gscale*

Scale factor for the offset from the path for the path data item displays. Defaults to 1.0.

### *Nopt*

Determines how data is displayed:

#### **(blank)**

Do not display nodes, and scale the display based on the currently selected node set (default).

#### **NODE**

Display path item data along with the currently selected set of nodes. The display geometry is scaled to the selected node set.

## Notes

You can use the *Gscale* argument to scale the contour display offset from the path for clarity. You need to type all six characters to issue this command.

## Menu Paths

Main Menu>General Postproc>Path Operations>Plot Path Item>On Geometry  
Main Menu>General Postproc>Plot Results>Plot Path Item>On Geometry

**PLPATH**, *Lab1, Lab2, Lab3, Lab4, Lab5, Lab6*

Displays path items on a graph.

POST1: Path Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Lab1, Lab2, Lab3, . . . , Lab6*

Labels identifying the path items to be displayed. Up to six items may be drawn per frame. Predefined path geometry items XG, YG, ZG, and S [**PDEF**] may also be displayed.

## Notes

The path must have been defined by the **PATH** and **PPATH** commands. Path items and their labels must have been defined with the **PDEF**, **PVECT**, **PCALC**, **PDOT**, **PCROSS**, or **PLNEAR** commands. Path items may also be printed with the **PRPATH** command. Graph scaling may be controlled with the **/XRANGE**, **/YRANGE**, and **PRANGE** commands. You need to type all six characters to issue this command.

## Menu Paths

Main Menu>General Postproc>Path Operations>Plot Path Item>On Graph  
Main Menu>General Postproc>Plot Results>Plot Path Item>On Graph  
Utility Menu>Plot>Results>Path Plot

**PLSCH**, *Fname, Ext, Lab, Port*

Converts and plots scattering, admittance, or impedance parameters on a Smith chart.

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**Fname**

File name and directory path of a Touchstone file (248 characters maximum, including the characters needed for the directory path). A Touchstone file may be created in ANSYS by performing a frequency sweep using the SPSWP command macro, or may it be supplied from another source. An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

**Ext**

Filename extension (8 character maximum).

**Lab**

Output parameter:

**S**

Scattering parameter (default)

**Y**  
Admittance parameter

**Z**  
Impedance parameter

**Port**  
Port number. Defaults to 1.

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>General Postproc>Plot Results>Smith Chart**

---

**PLSECT**, *Item*, *Comp*, *RHO*, *KBR*

**Displays membrane and membrane-plus-bending linearized stresses.**

POST1: Path Operations

MP ME ST PR PRN DS DSS <> <> <> DY PP <> EME MFS

### *Item*

Label identifying the item to be processed. Valid item labels are shown in [Table 243: PLSECT - Valid Item and Component Labels \(p. 1293\)](#) below. Items also require a component label.

### *Comp*

Component of the item. Valid component labels are shown in [Table 243: PLSECT - Valid Item and Component Labels \(p. 1293\)](#) below.

### *RHO*

In-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If zero (or blank), a plane or 3-D structure is assumed. If nonzero, an axisymmetric structure is assumed. Use a very large number (or -1) for an axisymmetric straight section.

### *KBR*

Through-thickness bending stresses key for an axisymmetric analysis ( $RHO \neq 0$ ):

**0**  
Include the thickness-direction bending stresses.

**1**  
Ignore the thickness-direction bending stresses.

**2**  
Include the thickness-direction bending stress using the same formula as the Y (axial direction ) bending stress. Also use the same formula for the shear stress.

## Notes

Calculates and displays the membrane and membrane-plus-bending linearized stresses (as described for the **PRSECT** command) along a path section [**PATH**] as a graph. The path section is defined by two points specified with the **PPATH** command. For linearized stress calculations, the path must be defined with nodes. The path



must be entirely within the selected elements (that is, there must not be any element gaps along the path). The total stress (equivalent to the **PLPATH** display) is also displayed. This command always uses 48 divisions along the path, regardless of the number of divisions defined by **PATH**.

Portions of this command are not supported by PowerGraphics [/GRAPHICS,POWER].

**Table 243 PLSECT - Valid Item and Component Labels**

Item	Comp	Description
Valid item and component labels for element results are:		
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.

## Menu Paths

**Main Menu>General Postproc>Path Operations>Linearized Strs**  
**Main Menu>General Postproc>Plot Results>Plot Path Item>Lineariz Strs**

**PLSYZ**, *Fname*, *Ext*, *Lab*, *Opt*, *VAL\_I1*, *VAL\_J1*, *VAL\_I2*, *VAL\_J2*, *VAL\_I3*, *VAL\_J3*, *VAL\_I4*, *VAL\_J4*

**Converts and plots network parameters versus frequency or plots losses versus frequency.**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

### ***Fname***

File name and directory path of a Touchstone file (248 characters maximum, including the characters needed for the directory path). A Touchstone file may be created in ANSYS by performing a frequency sweep using the SPSWP command macro, or may it be supplied from another source. An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name. *Fname* is only used for network parameters (*Lab* = S, Y, or Z).

### ***Ext***

Extension of Touchstone file (.snp where n is the number of ports). *Ext* is only used for network parameters (*Lab* = S, Y, or Z).

### ***Lab***

- S**  
Scattering parameter (default)
- Y**  
Admittance parameter
- Z**  
Impedance parameter
- REFL**  
Reflection loss
- INSL**  
Insertion loss

**ISOL**

Isolation loss

**VSWR**

Voltage standing wave ratio

**Opt**For  $Lab = S, Y, \text{ or } Z$ , format of Touchstone file:**MAG**

Magnitude (default)

**ANG**

Angle

**DB**Magnitude in dB units. Only available for scattering parameters ( $Lab = S$ ).**REAL**

Real part of the parameter

**IMAG**

Imaginary part of the parameter

For  $Lab = \text{REFL, INSL, or ISOL}$ , DB (magnitude in dB units) is the only available option. For  $Lab = \text{VSWR}$ , MAG (magnitude) is the only available option.

**VAL\_I1**For  $Lab = S, Y, \text{ or } Z$ , index I of first parameter plotted. For example, SIJ. Defaults to 1.For  $Lab = \text{REFL, INSL, ISOL, or VSWR}$ , output port number.**VAL\_J1**For  $Lab = S, Y, \text{ or } Z$ , index J of first parameter plotted. For example, SIJ. Defaults to 1.For  $Lab = \text{REFL, INSL, ISOL, or VSWR}$ , input port number.**VAL\_I2, VAL\_J2, VAL\_I3, VAL\_J3, VAL\_I4, VAL\_J4**For  $Lab = S, Y, \text{ or } Z$ , index I and J of second, third, and fourth parameters plotted. For example, SIJ. No default.For  $Lab = \text{REFL, INSL, ISOL, or VSWR}$ ,  $VAL\_I2$  through  $VAL\_J4$  are not used.

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>General Postproc>Plot Results>S,Y,Z Parameters**

---

**PLTD**, *Fname*, *Ext*, *Lab*, *Opt*, *Vkey*, *NFFT*, *TSTART*, *TRISE*, *PORTI1*, *PORTJ1*, *PORTI2*, *PORTJ2*, *PORTI3*, *PORTJ3*, *PORTI4*, *PORTJ4*

**Displays TDR/TDT waveforms, an impedance profile, or a total waveform.**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

***Fname***

A Touchstone file containing the S-parameters of a n-port system.

***Ext***

Extension of the n-port Touchstone file.

***Lab***

Display type:

**SPAR**

Display TDR and TDT waveforms.

**IMPD**

Display impedance profile.

**TDWF**

Display total waveform.

***Opt***

Excitation source:

**STEP**

Step-like source (default).

**IMPU**

Gaussian impulse.

***Vkey***

Control key:

**Off**

No display of the source waveform (default).

**On**

Display the source waveform.

***NFFT***

Number of interpolation points for Fast Fourier Transformation. It must be an integer power of 2. *NFFT* must be greater than or equal to 256 and less than or equal to 4096. Defaults to 512.

***TSTART***

Source beginning time in seconds. Defaults to 0.

***TRISE***

Source rise time. For a step-like source, it is the time from 10% of the rise to 90% of the rise. For a Gaussian impulse, it is the time from 50% of the rise to 50% of the rise. The rise time must be greater than or equal to 5 time steps. If the *TRISE* input is less than 5 time steps, it is changed to 5 time steps. Defaults to 5 time steps.

***PORTI1***

Driven port number. Defaults to 1.

**PORTJ1**

Output port number. Defaults to 1.

**PORTI2**

Driven port number. No default.

**PORTJ2**

Output port number. No default.

**PORTI3**

Driven port number. No default.

**PORTJ3**

Output port number. No default.

**PORTI4**

Driven port number. No default.

**PORTJ4**

Output port number. No default.

## Notes

The time step is given by  $dt = 1/(2f_{max})$  where  $f_{max}$  is the maximum frequency.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## PLTIME, TMIN, TMAX

**Defines the time range for which data are to be displayed.**

[POST26: Display](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**TMIN**

Minimum time (defaults to the first point stored).

**TMAX**

Maximum time (defaults to the last point stored).

## Command Default

Use the previously defined range [[TIMERANGE](#)].

## Notes

Defines the time (or frequency) range (within the range stored) for which data are to be displayed. Time is always displayed in the Z-axis direction for 3-D graph displays. If  $XVAR = 1$ , time is also displayed in the X-axis direction and this control also sets the abscissa scale range.

## Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

**PLTLINE**, *Filename, Ext, Lab, Modenum*

Plots port transmission line data generated by the HFPCSWP or SPSWP macros.

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

### *Filename*

A file name containing the transmission line results.

### *Ext*

Filename extension (8 character maximum).

### *Lab*

Data type:

#### **BETA**

Propagating constant.

#### **ZC**

Characteristic impedance.

#### **EREFF**

Effective dielectric constant.

### *Modenum*

#### **ALL**

Plot data for all modes (default).

#### **N**

Plot data for the Nth mode.

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

This command cannot be accessed from a menu.

**PLTRAC**, *Analopt, Item, Comp, TRPNum, Name, MXLOOP, TOLER, OPTION, ESCL, MSCL*

Displays a particle flow or charged particle trace on an element display.

POST1: Trace Points

MP <> <> <> <> <> <> FL EM <> <> PP <> <> <>

### *Analopt*

Analysis option

**FLUID**

Particle trace in fluid flow (default)

**ELEC**

Particle trace in electric field

**MAGN**

Particle trace in magnetic field

**EMAG**

Particle trace in presence of both electric and magnetic fields

**Item**

Label identifying the item to be contoured. Valid item labels are shown in [Table 244: PLTRAC - Valid Item and Component Labels](#) (p. 1299) below. Some items also require a component label. If *Item* is blank, display only the path trajectory.

**Comp**

Component of the item (if required). Valid component labels are shown in [Table 244: PLTRAC - Valid Item and Component Labels](#) (p. 1299) below.

**TRPNum**

Trace point number for storing trajectory data for use with **PATH** logic. Defaults to 0 (no trajectory path data is stored for further processing with **PATH** logic).

**Name**

Name of prefix of array variable. Defaults to TRAC. *NamePOIN* stores trajectory path points for trace point number *TRPNum*. If *AnaIopt* = ELEC, MAGN, or EMAG, two additional array parameters, *NameDATA* and *NameLABL*, store trajectory path data and labels for the same *TRPNum*.

**MXLOOP**

Maximum number of loops traced by a particle. Defaults to 25 for *Opt* = FLUID; otherwise, defaults to 1000.

**TOLER**

Length tolerance used for particle trajectory geometry calculation. Valid only for *AnaIopt* = ELEC, MAGN, or EMAG. If particle trace appears to terminate inside an element, adjusting the length tolerance may be necessary. Defaults to  $1.0 \times 10^{-8}$ .

**OPTION**

Flow trace option:

**0**

Use the undeformed mesh for computing the flow trace.

**1**

Use the deformed mesh for computing the flow trace.

**ESCL**

Electric field scale factor. Setting this scale factor affects only the tracing, not the field solution results. A negative factor corresponds to the opposite vector direction. Valid only for *AnaIopt* = ELEC or EMAG. Defaults to 1.

**MSCL**

Magnetic field scale factor. Setting this scale factor affects only the tracing, not the field solution results. A negative factor corresponds to the opposite vector direction. Valid only for *AnaIopt* = MAGN or EMAG. Defaults to 1.

## Notes

For a specified item, the variation of the item is displayed along the particle trace as a color-contoured ribbon. The **TRPOIN** command must be used to define a point on the trajectory path. Multiple traces may be displayed simultaneously by defining multiple trace points. Issue the **TRPLIS** command to list the current tracing points. Issue the **TRPDEL** command to delete tracing points defined earlier. Use the **PAPUT** command with the POIN option to retrieve the particle trajectory points as path points.

The model must be 3-D for the ELEC, MAGN, and EMAG analysis options.

Three array parameters are created at the time of the particle trace: TRACPOIN, TRACDATA and TRACLABL. These array parameters can be used to put the particle velocity and the elapsed time into path form. The procedure to put the arrays into a path named PATHNAME is as follows:

```
*get , npts , PARM , TRACPOIN , DIM , x
PATH , PATHNAME , npts , 9 , 1
PAPUT , TRACPOIN , POINTS
PAPUT , TRACDATA , TABLES
PAPUT , TRACLABL , LABELS
PRPATH , S , T_TRACE , VX_TRACE , VY_TRACE , VZ_TRACE , VS_TRACE
```

Not used if *AnaIopt* = FLUID. If working in the GUI, use the "All information" option to retrieve information from all three arrays at once.

If *OPTION* is set to 1, the deformed mesh is based on the displacement degrees of freedom UX, UY, and UZ, which must be available in the load step.

**Table 244 PLTRAC - Valid Item and Component Labels**

Item	Comp	Description
<b>Valid item and component labels for nodal degree of freedom results are:</b>		
TEMP		Temperature.
PRES		Pressure.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.
<b>Valid item and component labels for FLOTRAN nodal results are:</b>		
TTOT		Total temperature.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
SPHT		Specific heat.
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.

Item	Comp	Description
ECON		Fluid effective conductivity.
<b>Valid item labels for <i>Ana1opt</i> = ELEC nodal results are:</b>		
VOLT		Electric potential.
<b>Valid item labels for <i>Ana1opt</i> = MAGN or EMAG nodal results are:</b>		
None		Color contour displayed.

See the [Basic Analysis Guide](#) for more information on particle flow and charged particle traces. See [Animation](#) in the [Basic Analysis Guide](#) for information on particle trace animation.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>General Postproc>Plot Results>Particle Trace**

**Main Menu>General Postproc>Plot Results>Plot Flow Tra**

**Utility Menu>Plot>Results>Flow Trace**

**Utility Menu>PlotCtrls>Animate>Particle Flow**

---

**PLVAR**, *NVAR1*, *NVAR2*, *NVAR3*, *NVAR4*, *NVAR5*, *NVAR6*, *NVAR7*, *NVAR8*, *NVAR9*, *NVAR10*

**Displays up to ten variables in the form of a graph.**

[POST26:Display](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NVAR1*, *NVAR2*, *NVAR3*, . . . , *NVAR10*

Variables to be displayed, defined either by the reference number or a unique thirty-two character name. If duplicate names are used the command will plot the data for the lowest-numbered variable with that name.

## Notes

Variables are displayed vs. variable *N* on the **XVAR** command. The string value will be a predefined, unique name. For complex variables, the amplitude is displayed by default [**PLCPLX**]. Each **PLVAR** command produces a new frame. See the **/GRTYP** command for displaying multiple variables in a single frame with separate Y-axes.

## Menu Paths

**Main Menu>TimeHist Postpro>Graph Variables**

---



**PLVAROPT**, *Lab1, Lab2, Lab3, Lab4, Lab5, Lab6, Lab7, Lab8, Lab9, Lab10*

**Displays up to ten parameters in the form of a graph.**

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Lab1, Lab2, Lab3, . . . , Lab10*

Names of the parameters to be displayed on this frame.

### Command Default

No display.

### Notes

Displays up to ten parameters in the form of a graph. Parameters are displayed vs. *Lab* on the **XVAROPT** command (defaults to set number) in the order corresponding to an ascending order of the **XVAROPT** parameter. See the **/GRTYP** command for displaying multiple parameters in a single frame with separate Y-axes. Each **PLVAROPT** command produces a new frame.

### Menu Paths

Main Menu>Design Opt>Design Sets>Graphs/Tables

**PLVECT**, *Item, Lab2, Lab3, LabP, Mode, Loc, Edge, KUND*

**Displays results as vectors.**

POST1: Results

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Item**

Predefined vector item (from [Table 245: PLVECT - Valid Item Labels \(p. 1303\)](#) below) or a label identifying the i-component of a user-defined vector.

**Lab2**

Label identifying the j-component of a user-defined vector. In most cases, this value must be blank if *Item* is selected from [Table 245: PLVECT - Valid Item Labels \(p. 1303\)](#). Individual principal stresses (*Item* = S) or principal strains (*Item* = EP<sub>xx</sub>) may be plotted by specifying the value as 1, 2, or 3.

**Lab3**

Label identifying the k-component of a user-defined vector. Must be blank if *Item* is selected from list below or for 2-D user defined vector.

**LabP**

Label assigned to resultant vector for display labeling (defaults to *Item*).

**Mode**

Vector or raster mode override key:

**(blank)**

Use the setting of *KEY* on the **/DEVICE** command.

**RAST**

Use raster mode for **PLVECT** displays.

**VECT**

Use vector mode for **PLVECT** displays.

**Loc**

Vector location for display of field element results:

**ELEM**

Display at element centroid (default).

**NODE**

Display at element nodes.

Nodal results quantities will only be displayed at nodes, not at element centroids.

**Edge**

Edge display override key:

**(blank)**

Use the setting of Key on the **/EDGE** command.

**OFF**

Deactivate the edge display.

**ON**

Activate the edge display.

**KUND**

Undisplaced shape key:

**0**

Display vectors on undeformed mesh or geometry.

**1**

Display vectors on deformed mesh or geometry.

## Notes

Displays various solution results as vectors (arrows) for the selected nodes and/or elements (elements must contain at least three nodes that are not colinear). For example, **PLVECT,U** displays the displacement vector for all selected nodes. For section displays [**/TYPE**], the vectors are shown only on the section face (i.e., cutting plane). The **PLVECT** display of principal strains and stresses (*Item* = S, EPTO, EPEL, EPPL, EPCR, or EPTH) on a "cut" of the model (**/TYPE,,1,5,7,8, or 9**) is not supported. The resulting plot displays the vectors on *all* selected elements, not on just the sliced surface. See the **/VSCALE** command to scale vector lengths. Vector magnitudes may be shown as a contour display with the **PLNSOL** command. Various results also depend upon the recalculation method and the selected results location [**LAYER, SHELL, and NSEL**].

Items may be selected from a set of recognized vector labels (*Item*) or a vector may be defined from up to three scalar labels (*Item,Lab2,Lab3*). Scalar labels may be user-defined with the **ETABLE** command. The vectors appear on an element display as arrows showing the relative magnitude of the vector and its direction. The predefined items will be shown either at the node or at the element centroid, depending on what item is being displayed and depending on the *Loc* setting. User defined **ETABLE** items will be shown at the element centroid, regardless of the *Loc* setting. Stress vectors appear as arrows at the element centroid, with the arrowheads pointing away from each other for tension and toward each other for compression.

For PowerGraphics, vector arrow displays are generated in Global Cartesian (**RSYS** = 0). All subsequent displays will revert to your original coordinate system.

When vector mode is active (*Mode* = VECT), use the Z-buffered display type [**/TYPE,,6**] to maximize speed of **PLVECT** plots (other hidden display types may make plotting slow). For PowerGraphics [**/GRAPHICS,POWER**], the items marked with [1 (p. 1304)] are not supported by PowerGraphics.

It is possible to plot principal stresses (*Item* = S) or principal strains (*Item* = EPxx) individually. To do so, specify a *Lab2* value of 1, 2, or 3. For example, the following are valid commands:

```
PLVECT,S,1,,,VECT,ELEM,ON,0
PLVECT,EPEL,3,,,VECT,NODE,ON,0
```

**Table 245 PLVECT - Valid Item Labels**

Item	Description
<b>Valid item labels for nodal degree of freedom vector results are:</b>	
U	Structural displacement vector.
ROT	Structural rotation vector.
V	Velocity vector.
A	Magnetic vector potential vector.
FFLX	Fluid flux in poromechanics.
<b>Valid item labels for structural element results are:</b>	
S	Principal stresses[1].
EPTO	Principal total strain (EPEL + EPPL + EPCR)[1].
EPEL	Principal elastic strains[1].
EPPL	Principal plastic strains[1].
EPCR	Principal creep strains[1].
EPTH	Principal thermal strains[1].
<b>Valid item labels for field element results are:</b>	
TG	Thermal gradient vector.
TF	Thermal flux vector.
PG	Pressure gradient vector.
EF	Electric field vector.
D	Electric flux density vector.
H	Magnetic field intensity vector. If Lab2 is blank, then Item is interpreted as one of the predefined labels. Otherwise, Item is interpreted as a user-defined <b>ET</b> label and ANSYS will request a nonblank Lab2/Lab3 according to the dimension of the problem.
B	Magnetic flux density vector.
FMAG	Electromagnetic force vector.
P	Poynting vector.
JS	Source current density vector for low-frequency magnetic analyses. Total current density vector (sum of conduction

Item	Description
	and displacement current densities) in low frequency electric analyses.
JT	Total measureable current density vector in low-frequency electromagnetic analyses. (Conduction current density vector in a low-frequency electric analysis.)
JC	Conduction current density vector for elements that support conduction current calculation.
1.	Not supported by PowerGraphics

## Menu Paths

**Main Menu>General Postproc>Plot Results>Vector Plot>Predefined**  
**Main Menu>General Postproc>Plot Results>Vector Plot>User-defined**  
**Utility Menu>Plot>Results>Vector Plot**  
**Utility Menu>PlotCtrls>Animate>Q-Slice Vectors**

## PLVFRC, *CONT*

Displays volume fractions in a volume of fluid (VOF) analysis.

[PREP7: FLOTRAN Miscellaneous](#)

MP <> <> <> <> <> <> FL <> <> <> PP <> <> <>

### *CONT*

Contour setting:

- 0**  
Smooth plot with contour levels at 0.5 and 1.0 (default).
- 1**  
Smooth plot with current settings of contour levels.

## Notes

The **PLVFRC** command macro places the volume fraction results into an element table and then plots the element table data (i.e., **ETABLE**,VFRC,NMISC,1 and then **PLETAB**,VFRC,1). If *CONT* = 0 (default), the command macro issues the /CVAL,ALL,0.5,1.0 command before the **ETABLE** command. This means **PLETAB** uses contour levels of 0.5 and 1.0 to indicate the free surface and fluid regions. If *CONT* = 1, **PLETAB** uses the current contour settings.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Drop Test>Animate Results**  
**Main Menu>General Postproc>Plot Results>Contour Plot>Volume Fraction**  
**Utility Menu>PlotCtrls>Animate>Animate Over Results**  
**Utility Menu>PlotCtrls>Animate>Animate Over Time**  
**Utility Menu>PlotCtrls>Animate>Time-harmonic**

---

**Utility Menu>PlotCtrls>Multi-Plot Contrls**


---

**PLWAVE**, *Ex, Ey, Ez, AngX, AngZ*

Specifies a free-space time-harmonic incident plane electromagnetic wave.

SOLUTION: Misc Loads

MP <> <> <> <> <> <> <> <> <> <> PP <> <> <>

***Ex***

Electric field amplitude in x direction.

***Ey***

Electric field amplitude in y direction.

***Ez***

Electric field amplitude in z direction.

***AngX***

Angle between incident wave vector and X-axis (Phi).

***AngZ***

Angle between incident wave vector and Z-axis (Theta).

**Notes**

Defines an incident plane wave for the entire solution domain. See [Spherical Coordinates](#) in the *High-Frequency Electromagnetic Analysis Guide*.

This command cannot be used for the analysis of periodic structures (see the **HFPORT** command).

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>Planewav0>Define Wave**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>Planewav0>Wave Status**

**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>Planewav0>Define Wave**

**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>Planewav0>Wave Status**

---

**/PMACRO**

Specifies that macro contents be written to the session log file.

APDL: Macro Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

This command forces the contents of a macro or other input file to be written to Jobname.LOG. It is valid only within a macro or input file, and should be placed at the top of the file. **/PMACRO** should be included in any macro or input file that calls GUI functions.

## Menu Paths

This command cannot be accessed from a menu.

### PMP, FORM, DISCON

Creates mapping of the path geometry by defining path interpolation division points.

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### FORM

Defines the mapping method:

##### UNIFORM

Maps uniform divisions (specified on the *nDiv* argument of the **PATH** command) between specified points. This is the default.

##### ACCURATE

Map geometry using a small division at the beginning and end of each segment. This gives you accurate derivatives, integrals, tangents, and normals for curves which do not have continuous slopes at the specified points. To create nonuniform divisions, the *nDiv* argument of the **PATH** command must be greater than 2.

#### DISCON

Sets mapping for discontinuities in the field. The divisions are modified to put a point just before and just after the discontinuity. The valid label is MAT, for a material discontinuity. No discontinuity is the default. Discontinuity mapping involves the NOAV option on the **PDEF** command.

## Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>Path Options

Main Menu>Preprocessor>Path Operations>Define Path>Path Options

### PMGTRAN, Fname, FREQ, Fcnam1, Fcnam2, Pcnam1, Pcnam2, Ecnam1, Ccnam1

Summarizes electromagnetic results from a transient analysis.

[POST26: Special Purpose](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> <> <>

#### Fname

File name (8 characters maximum) to which tabular data and plot files will be written. Must be enclosed in single quotes when the command is manually typed in. Defaults to MG\_TRNS. The data file extension is .OUT and the plot file extension is .PLT.

#### FREQ

Frequency of solution output. Defaults to 1. Every *FREQ*th solution on the results file is output.

#### Fcnam1, Fcnam2

Names of element components for force calculation. Must be enclosed in single quotes when the command is manually typed in.

***Pcnam1, Pcnam2***

Names of element components for power loss calculation. Must be enclosed in single quotes when the command is manually typed in.

***Ecnam1, Ccnam1***

Names of element components for energy and total current calculations, respectively. Must be enclosed in single quotes when the command is manually typed in.

## Notes

**PMGTRAN** invokes an ANSYS macro which calculates and summarizes electromagnetic results from a transient analysis. The results are summarized by element components and listed on the screen as well as written to a file (*Fname*.OUT). Also, graph plots of results as a function of time are created and written to a file (*Fname*.PLT) for use in the DISPLAY program.

Two components may be selected for the summary of electromagnetic forces (see **FMAGSUM**), two for power loss, and one each for stored energy (see **SENERGY**) and total current (see **CURR2D**). See the referenced commands for other restrictions.

**PMGTRAN** is restricted to MKSA units.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>TimeHist Postpro>Elec&Mag>Magnetics

### **PMLOPT**, *ESYS, Lab, Xminus, Xplus, Yminus, Yplus, Zminus, Zplus*

**Defines perfectly matched layers (PMLs) for a high-frequency analysis.**

[PREP7: Perfectly Matched Layers](#)

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

***ESYS***

Element coordinate system number. *ESYS* may be 0 (global Cartesian) or any previously defined local Cartesian coordinate system number (>10). Defaults to 0.

***Lab***

Label defining the number of dimensions:

**ONE**

A one-dimensional PML region.

**THREE**

A three-dimensional PML region (default).

***Xminus***

For harmonic analysis, normal reflection coefficient in negative X direction of *ESYS*. Defaults to 1.E-3 (equivalent to -60 dB).

***Xplus***

For harmonic analysis, normal reflection coefficient in positive X direction of *ESYS*. Defaults to 1.E-3 (equivalent to -60 dB).

***Yminus***

For harmonic analysis, normal reflection coefficient in negative Y direction of ESYS. Defaults to 1.E-3 (equivalent to -60 dB).

***Yplus***

For harmonic analysis, normal reflection coefficient in positive Y direction of ESYS. Defaults to 1.E-3 (equivalent to -60 dB).

***Zminus***

For harmonic analysis, normal reflection coefficient in negative Z direction of ESYS. Defaults to 1.E-3 (equivalent to -60 dB).

***Zplus***

For harmonic analysis, normal reflection coefficient in positive Z direction of ESYS. Defaults to 1.E-3 (equivalent to -60 dB).

## Notes

**PMLOPT** defines perfectly matched layers (PML) for a high-frequency analysis. Each PML region must have a uniquely defined element coordinate system. Normal reflection coefficient values for a harmonic analysis must be less than 1.

Issue **PMLOPT,STAT** to list the current normal reflection coefficient or attenuation factor settings for a PML region. Issue **PMLOPT,CLEAR** to clear all normal reflection coefficient settings and restore them to the defaults. Issue **PMLOPT,ESYS,CLEAR** to clear all normal reflection coefficient settings for this element coordinate system and restore them to the defaults.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PMLOpt>Clear**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PMLOpt>Define**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PMLOpt>Status**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PMLOpt>All**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PMLOpt>On ESYS**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PMLOpt>Clear**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PMLOpt>Define**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PMLOpt>Status**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PMLOpt>All**  
**Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PMLOpt>On ESYS**

---

## **PMLSIZE**, *FREQB*, *FREQE*, *DMIN*, *DMAX*, *THICK*, *ANGLE*

**Determines number of PML layers.**

[PREP7: Perfectly Matched Layers](#)

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

***FREQB***

Minimum operating frequency

***FREQE***

Maximum operating frequency



**DMIN**

Minimum distance from radiation source to PML interface.

**DMAX**

Maximum distance from radiation source to PML interface.

**THICK**

Thickness of PML region. Defaults to 0.

**ANGLE**

Incident angle of wave to the PML interface. Defaults to 0.

## Notes

**PMLSIZE** determines the number of PML layers for acceptable numerical accuracy.

**PMLSIZE** must be issued before any meshing commands. If the thickness of the PML region is known, it determines an element edge length (h) and issues `ESIZE,h`. If the thickness of the PML region is unknown, it determines the number of layers (n) and issues `ESIZE,,n`.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Size Cntrl>PML**

**/PMORE**, --, X5, Y5, X6, Y6, X7, Y7, X8, Y8

**Creates an annotation polygon (GUI).**

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

--

Unused field.

**X5**

X location for vertex 5 of polygon (-1.0 < X < 2.0).

**Y5**

Y location for vertex 5 of polygon (-1.0 < Y < 1.0).

**X6**

X location for vertex 6 of polygon (-1.0 < X < 2.0).

**Y6**

Y location for vertex 6 of polygon (-1.0 < Y < 1.0).

**X7**

X location for vertex 7 of polygon (-1.0 < X < 2.0).

**Y7**

Y location for vertex 7 of polygon (-1.0 < Y < 1.0).

**X8**

X location for vertex 8 of polygon (-1.0 < X < 2.0).

**Y8**

Y location for vertex 8 of polygon (-1.0 < Y < 1.0).

## Notes

Defines the 5th through 8th vertices of an annotation polygon [**/POLYGON**]. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

**PNGR**, *Kywrđ*, *OPT*, *VAL*

Provides PNG file export for ANSYS displays.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Kywrđ*

Specifies various PNG file export options.

#### **COMP**

If *Kywrđ* = COMP, then *OPT* is either ON or OFF (blank is interpreted as OFF). This option allows you to turn PNG file compression ON or OFF. If *OPT* = ON, then The *VAL* field is read to determine the degree of compression. See the *VALUE* argument for acceptable compression values.

#### **ORIENT**

If *Kywrđ* = ORIENT, then *OPT* will determine the orientation of the entire plot. *OPT* can be either Horizontal (default) or Vertical.

#### **COLOR**

If *Kywrđ* = COLOR, then *OPT* will determine the color depth of the saved file. *OPT* can be 0, 1, or 2, corresponding to Black and White, Grayscale, and Color (default), respectively.

#### **TMOD**

If *Kywrđ* = TMOD, then *OPT* will determine the text method. *OPT* can be either 1 or 0, corresponding to bitmap text (default) or line stroke text, respectively.

#### **DEFAULT**

If *Kywrđ* = DEFAULT, then all of the default values, for all of the *Kywrđ* parameters listed above, are active.

#### **STAT**

Shows the current status of PNG file export.

### *OPT*

*OPT* can have the following names or values, depending on the value for *Kywrđ* (see above).

#### **ON, OFF**

If *Kywrđ* = COMP, the values On and Off control the use of compression. The degree of compression is determined by *VAL*

**Horizontal, Vertical**

If *Kywrđ* = ORIENT, the terms Horizontal or Vertical determine the orientation of the plot.

**0, 1, 2**

If *Kywrđ* = COLOR, the numbers 0, 1, and 2 correspond to Black and White, Grayscale and Color, respectively.

**1, 0**

If *Kywrđ* = TMOD, the values 1 and 0 determine whether bitmap (1) or stroke text (0) fonts will be used

**VAL**

VAL is active only when *Kywrđ* = COMP, and determines the degree of compression applied to the exported file (see above).

**1**

Apply the default, optimum value for compression. This value represents the best combination of speed and compression. It varies according to the release level of the ZLIB compression package.

**1-9**

Use this value to specify a specific compression level. 1 is the lowest compression level (fastest) and 9 is the highest compression level (slowest).

**Menu Paths**

**Utility Menu>PlotCtrls>Redirect Plots**

**Utility Menu>PlotCtrls>Hard Copy**

**/PNUM**, *Label*, *KEY***Controls entity numbering/coloring on plots.**

**GRAPHICS:Labeling**

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Label**

Type of numbering/coloring:

**NODE**

Node numbers on node and element plots.

**ELEM**

Element numbers and colors on element plots.

**SEC**

Section numbers and colors on element and solid model plots (see *"Notes"* (p. 1312)).

**MAT**

Material set numbers and colors on element and solid model plots (see *"Notes"* (p. 1312)).

**TYPE**

Element type reference numbers and colors on element and solid model plots (see *"Notes"* (p. 1312)).

**REAL**

Real constant set numbers and colors on element and solid model plots (see *"Notes"* (p. 1312)).

**ESYS**

Element coordinate system numbers on element and solid model plots (see *"Notes"* (p. 1312)).

**PART**

Element part numbers and colors on element plots (applicable to ANSYS LS-DYNA only).

**LOC**

Location numbers/colors of the element in matrix assembly order on element plots.

---

**Note**

LOC and ELEM numbers will be the same unless the model has been reordered.

**KP**

Keypoint numbers on solid model plots.

**LINE**

Line numbers on solid model plots (both numbers and colors on line plots).

**AREA**

Area numbers on solid model plots (both numbers and colors on area plots).

**VOLU**

Volume numbers on solid model plots (both numbers and colors on volume plots).

**SVAL**

Stress (or any contour) values on postprocessing plots, and surface load values and colors on model plots (when surface load symbols are on [/PSF]). For tabular boundary conditions, the table-evaluated values will be displayed on node, element, or contour displays in POST1 when load symbols (/PBF, /PSF, /PBC) are on and TABNAM is OFF.

**TABNAM**

Table names for tabular boundary conditions. If this label is turned on, the table name appears next to the appropriate symbol, arrow, face outline, or contour as dictated by the /PSF, /PBC, and /PBF commands.

**STAT**

Shows current settings for /PNUM.

**DEFA**

Resets all /PNUM specifications back to default.

**KEY**

Switch:

**0**

Turns OFF numbers/colors for specified label.

**1**

Turns ON numbers/colors for specified label.

## Notes

This command specifies entity numbering and coloring for subsequent plots.

The MAT, TYPE, REAL, and ESYS labels activate both the numbering and coloring of the corresponding attributes for **E**PLOT, **K**PLOT, **L**PLOT, **A**PLOT, and **V**PLOT. The ELEM, MAT, TYPE, REAL, ESYS, PART (ANSYS LS-

DYNA only), and LOC labels are mutually exclusive, i.e., only one can be specified at a time. Also, turning on a LINE, AREA, or VOLU label will turn off the MAT, TYPE, REAL, and PART labels.

PowerGraphics [/GRAPHICS,POWER] displays for/PNUM can be problematic. /PNUM,ELEM will display erratically depending on other display command specifications, while /PNUM,LOC and /PNUM,ESYS are not supported.

Element and volume numbers are not visible for 3-D elements and volumes when Z-buffering is turned on (/TYPE,,[6,7, or 8]).

Use /PSTATUS or /PNUM,STAT to show settings. Use /PNUM,DEFA to reset all specifications back to default. Use the /NUMBER command to control whether numbers and colors are displayed together.

This command is valid in any processor

## Menu Paths

Utility Menu>PlotCtrls>Numbering

---

## POINT

Specifies "Point flow tracing settings" as the subsequent status topic.

POST1:Status

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

## Notes

This is a status [STAT] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>General Postproc>Trace Points

---

## POLY

Creates a polygonal area based on working plane coordinate pairs.

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Defines a polygonal area on the working plane. The area will be defined with NPT keypoints and NPT lines, where NPT (must be at least 3) is the number of coordinate pairs defined with the **PTXY** command. See the **RPOLY** and **RPR4** commands for other ways to create polygons.

## Menu Paths

This command cannot be accessed from a menu.

---

**/POLYGON**, *NVERT*, *X1*, *Y1*, *X2*, *Y2*, *X3*, *Y3*, *X4*, *Y4*

Creates annotation polygons (GUI).

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NVERT*

Number of vertices of polygon ( $3 \leq NVERT \leq 8$ ). Use **/PMORE** for polygons with more than 4 vertices.

### *X1*

X location for vertex 1 of polygon ( $-1.0 < X < 2.0$ ).

### *Y1*

Y location for vertex 1 of polygon ( $-1.0 < Y < 1.0$ ).

### *X2*

X location for vertex 2 of polygon ( $-1.0 < X < 2.0$ ).

### *Y2*

Y location for vertex 2 of polygon ( $-1.0 < Y < 1.0$ ).

### *X3*

X location for vertex 3 of polygon ( $-1.0 < X < 2.0$ ).

### *Y3*

Y location for vertex 3 of polygon ( $-1.0 < Y < 1.0$ ).

### *X4*

X location for vertex 4 of polygon ( $-1.0 < X < 2.0$ ).

### *Y4*

Y location for vertex 4 of polygon ( $-1.0 < Y < 1.0$ ).

## Notes

Creates annotation polygons to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if annotation

is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All polygons are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** and the **/PSPEC** command to set the attributes of the polygon. Use the **/PMORE** command to define the 5th through 8th vertices of the polygon.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

---

## /POST1

Enters the database results postprocessor.

SESSION: Processor Entry  
POST1: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Enters the general database results postprocessor (POST1). All load symbols (**/PBC**, **/PSF**, or **/PBF**) are automatically turned off with this command.

This command is valid only at the Begin Level.

## Menu Paths

Main Menu>General Postproc

---

## /POST26

Enters the time-history results postprocessor.

SESSION: Processor Entry  
POST26: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Enters the time-history results postprocessor (POST26).

This command is valid only at the Begin Level.

## Menu Paths

Main Menu>TimeHist Postpro

---

**POUTRES**, *Item1, Item2, Item3, Item4, Item5, Item6, Item7, Item8, Item9, Item10, Item11, Item12, Item13, Item14, Item15, Item16, Item17, Item18, Item19*

**Controls the nodal DOF and computed element results graphics data that is written to the PGR file.**

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Item1, Item2, Item3, ..., Item19*

Data written to the PGR file. Acceptable values are as follows:

**ALL**

All solution items are written to the PGR file.

**NONE**

No solution results are written to the PGR file (default)

**NSOL**

Nodal DOF solution (always written)

**S**

Stress

**NL**

Structural nonlinear data

**CONT**

Contact data

**EPTO**

Total strain

**EPEL**

Elastic Strain

**EPHT**

Thermal strain

**EPPL**

Plastic Strain

**EPCR**

Creep strain

**TG**

Thermal gradient

**TF**

Thermal flux

**EF**

Electric field

**D**

Electric flux density

**HF**

Magnetic field intensity

**B**

Magnetic flux density



**FMAG**

Electromagnetic forces

**PG**

Pressure gradient

**BFE**

Body temperatures

**TOPO**

Densities for topological optimization

**Command Default**

No solution results are written to the PGR file.

**Notes**

PowerGraphics data using the CONT option is not available for 2-D models.

This command is also valid in POST1.

**Menu Paths**

**Main Menu>General Postproc>Write PGR File**

---

**POWERH**

**Calculates the rms power loss in a conductor or lossy dielectric.**

[POST1: Magnetics Calculations](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> <> <>

**Notes**

**POWERH** invokes an ANSYS macro which calculates the time-averaged (rms) power loss in a conductor or lossy dielectric material from a harmonic analysis. The power loss is stored in the parameter PAVG. Conductor losses include solid conductors and surface conductors approximated by impedance or shielding boundary conditions. The power loss density for solid conductors or dielectrics is stored in the element table with the label PLOSSD and may be listed [**PRETAB**] or displayed [**PLETAB**]. PLOSSD does not include surface losses. The elements of the conducting region must be selected before this command is issued. **POWERH** is valid for 2-D and 3-D analyses.

**Menu Paths**

**Main Menu>General Postproc>Elec&Mag Calc>Element Based>Power Loss**

---

**PPATH**, *POINT*, *NODE*, *X*, *Y*, *Z*, *CS*

**Defines a path by picking or defining nodes, or locations on the currently active working plane, or by entering specific coordinate locations.**

POST1: Path Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**POINT**

The point number. It must be greater than zero and less than or equal to the *nPts* value specified on the **PATH** command if graphical picking is not being used.

**NODE**

The node number defining this point. If blank, use the *X*, *Y*, *Z* coordinates to define the point. A valid node number will override *X*, *Y*, *Z* coordinate arguments.

**X, Y, Z**

The location of the point in the global Cartesian coordinate system. Use these arguments only if you omit the *NODE* argument.

**CS**

The coordinate system for interpolation of the path between the previous point and this point. Omit this argument if you wish to use the currently active (**CSYS**) coordinate system. If the coordinate system of two adjacent points is different, the *CS* value of the latter point will be used.

## Notes

For linearized stress calculations, the path must be defined with nodes.

This command is designed and works best in interactive (GUI) mode, using the menu paths listed below. For command line operations, issue **PPATH,P** to define your path by picking nodes.

For information on displaying paths you have defined, see [Defining Data to be Retrieved](#) in the *Basic Analysis Guide*.

## Menu Paths

**Main Menu>General Postproc>Path Operations>Define Path>By Location**  
**Main Menu>General Postproc>Path Operations>Define Path>By Nodes**  
**Main Menu>General Postproc>Path Operations>Define Path>Modify Path**  
**Main Menu>General Postproc>Path Operations>Define Path>On Working Plane**  
**Main Menu>Preprocessor>Path Operations>Define Path>By Location**  
**Main Menu>Preprocessor>Path Operations>Define Path>By Nodes**  
**Main Menu>Preprocessor>Path Operations>Define Path>Modify Path**  
**Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane**

---

**PRANGE**, *LINC*, *VMIN*, *VMAX*, *XVAR*

**Determines the path range.**

POST1: Path Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**LINC**, **VMIN**, **VMAX**

Set the range for listing or displaying the table locations between a minimum value (*VMIN*) and a maximum value (*VMAX*) of the path distance with a location increment of *LINC* (defaults to 1). The first location begins at *VMIN*.

**XVAR**

Path variable item to be used as the x-axis plot variable. Any valid path variable may be used (**PDEF** command). Default variable is the path distance, *S*.

**Command Default**

Include every interpolation point and entire path distance.

**Notes**

Determines the path distance range for use with the **PRPATH** and **PLPATH** commands.

**Menu Paths**

Main Menu>General Postproc>Path Operations>Plot Path Item>Path Range

**PRCAMP**, *Option*, *SLOPE*, *UNIT*, *FREQB*, *Cname*, *STABVAL*, *KeyALLFreq*

**Prints Campbell diagram data for applications involving rotating structure dynamics.**

POST1: Special Purpose

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**Option**

Flag to activate or deactivate sorting of forward or backward whirl frequencies:

**0 (OFF or NO)**

No sorting.

**1 (ON or YES)**

Sort. This value is the default.

**SLOPE**

The slope of the line to be printed. This value must be greater than 0. The line represents the number of excitations per revolution of the rotor. For example, *SLOPE* = 1 represents one excitation per revolution, usually resulting from unbalance. The default is no line.

**UNIT**

Specifies the unit of measurement for rotational angular velocities:

**RDS**

Rotational angular velocities in radians per second (rad/s). This value is the default.

**RPM**

Rotational angular velocities in revolutions per minute (RPMs).

**FREQB**

The beginning, or lower end, of the frequency range of interest. The default is zero.

**Cname**

The rotating component name.

**STABVAL**

Flag to plot the stability values:

**0 (OFF or NO)**

Plot the frequencies (the imaginary parts of the eigenvalues in Hz). This value is the default.

**1 (ON or YES)**

Plot the stability values (the real parts of the eigenvalues in Hz).

**2**

Plot the logarithmic decrements.

For more information about complex eigenvalues and corresponding logarithmic decrements, see [Complex Eigensolutions](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

**KeyALLFreq**

Key to specify if all frequencies above FREQB are printed out:

**0 (OFF or NO)**

A maximum of 10 frequencies are printed out. They correspond to the frequencies displayed via the **PLCAMP** command. This value is the default.

**1 (ON or YES)**

All frequencies are printed out.

## Notes

To take the gyroscopic effect into account when printing a Campbell diagram, first issue the **CORIOLIS** command in the SOLUTION module.

The **PRCAMP** command is valid only for modal analyses (QR damped [**MODOPT**,**QRDAMP**] or damped [**MODOPT**,**DAMP**] methods only). The command works over *two or more* load step results generated with an ascending order of rotational velocity (**OMEGA** or **CMOMEGA**).

Complex eigenvalues are necessary (**MODOPT**,**QRDAMP**,*,,,Cpxmod* = ON), and you must specify the number of modes to expand (**MXPAND**) at each load step.

In some cases where modes are not in the same order from one load step to the other, sorting (*Option* = 1) the frequencies can help to obtain a correct print. Sorting is based on the comparison between complex mode shapes calculated at two successive load steps.

At each load step, ANSYS compares the mode shape to the loads at other loads steps to determine whirl direction at the load step. If applicable, a label appears (on the rows of output data) representing each whirl mode (BW for backward whirl and FW for forward whirl).

If you specify a non-zero slope (*SLOPE* > 0), the command prints the critical speeds corresponding to the intersection points of the frequency curves and the added line. In the case of a named component (*Cname*), critical speeds relate to the rotational velocity of the component.

ANSYS considers the *SLOPE* value only if the frequencies are plotted (*STABVAL* = OFF).

At each load step, ANSYS checks for instability (based on the sign of the real part of the eigenvalue). The label "U" appears on the printout for each unstable frequency.

The rotational velocities of a named component (*Cname*) are printed out along with the natural frequencies.

In general, ANSYS, recommends printing a Campbell diagram only when your analysis is performed in a *stationary* reference frame (**CORIOLIS**,,,,*RefFrame* = ON).

To print the Campbell diagram for a prestressed structure, first issue a **CAMPBELL,ON** command in the static portion of the analysis. For a usage example of the companion command **PLCAMP** (used for *plotting* a Campbell diagram), see [Sample Campbell Diagram Analysis](#).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Rotor Dynamics>Print Campbell

### PRCINT, *ID*, *Node*, *Dtype*

Lists the fracture parameter (**CINT**) results data.

[POST1:Results](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**ID**

Crack ID number.

**Node**

Crack tip node number (Default = ALL). Used only for 3-D.

**Dtype**

Data type to output:

**JINT**

J-integral

**IIN1**

Interaction integral 1

**IIN2**

Interaction integral 2

**IIN3**

Interaction integral 3

**K1**

Mode 1 stress-intensity factor

**K2**

Mode 2 stress-intensity factor

**K3**

Mode 3 stress-intensity factor

- G1**  
Mode 1 energy release rate
- G2**  
Mode 2 energy release rate
- G3**  
Mode 3 energy release rate
- GT**  
Total energy release rate

## Notes

When a crack tip node is defined, the values associated with the specified node are listed.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## PRCPLX, KEY

Defines the output form for complex variables.

[POST26: Listing](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### KEY

Output form key:

- 0**  
Real and imaginary parts.
- 1**  
Amplitude and phase angle. Stored real and imaginary data are converted to amplitude and phase angle upon output. Data remain stored as real and imaginary parts.

## Notes

Defines the output form for complex variables. Used only with harmonic analyses ([ANTYPE,HARMIC](#)).

All results data are stored in the form of real and imaginary components and converted to amplitude and/or phase angle as specified via the **PRCPLX** command. The conversion is not valid for derived results (such as principal stress/strain, equivalent stress/strain and USUM).

## Menu Paths

**Main Menu>TimeHist Postpro>Settings>List**

---

**PRED**, *Sskey*, *--*, *Lskey*

**Activates a predictor in a nonlinear analysis.**

SOLUTION: Nonlinear Options

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

***Sskey***

Substep predictor key:

**OFF**

No prediction occurs.

**ON**

Use a predictor on all substeps after the first. This behavior is the default, with certain exceptions; see "*Command Default*" (p. 1323) for details.

--

Unused field.

***Lskey***

Load step predictor:

**OFF**

No prediction across load steps occurs. This is the default behavior.

**ON**

Use a predictor also on the first substep of the load step. (*Sskey* = ON is required.)

***Command Default***

The default command behavior is to use prediction (*Sskey* = ON). However, prediction does not occur (*Sskey* = OFF) if one or more of these conditions exist:

- You are using the SOLID65 element.
- You are mapping (MAPSOLVE) variables to a new mesh during rezoning. (Prediction does not occur for any MAPSOLVE substeps, nor for the first substep afterwards.)

**Notes**

Activates a predictor in a nonlinear analysis on the degree-of-freedom solution for the first equilibrium iteration of each substep.

The default values given for this command assume SOLCONTROL,ON (the default). See the description of the SOLCONTROL command for a complete listing of the defaults set via SOLCONTROL,ON and SOLCONTROL,OFF.

When using the arc-length method (ARCLen, ARCTRM), you cannot issue the DOF solution predictor command (PRED), the automatic time stepping command (AUTOTS), or the line search command (LNSRCH). If you activate the arc-length method after you set PRED, AUTOTS, or LNSRCH, a warning message appears. If you elect to proceed with the arc-length method activation, ANSYS disables your DOF predictor, automatic time stepping, and line search settings.

When using step-applied loads, such as TUNIF, BFUNIF, etc., or other types of non-monotonic loads, the predictor may adversely affect the convergence. If the solution is discontinuous, the predictor may need to be turned off.

When performing a nonlinear analysis involving large rotations, the predictor may require using smaller substeps.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Predictor**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear**

**Main Menu>Solution>Load Step Opts>Nonlinear>Predictor**

---

**PREENERGY**, *EnergyType*, *Cname1*, *Cname2*, *Cname3*, *Cname4*, *Cname5*, *Cname6*

**Prints the total energies of a model or the energies of the specified components.**

[POST1:Results](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *EnergyType*

Type of energies to be printed:

#### **ALL**

All energies are printed: potential, kinetic, artificial hourglass/drill stiffness, and artificial stabilization energy when applicable. This is the default.

#### **SENE**

Potential energy.

#### **KENE**

Kinetic energy.

### *Cname1*, *Cname2*, *Cname3*,...

Component names for energies of the components printout.

If *Cname1* is blank, the total energies are listed.

If *Cname1* = ALL, the energies are listed for all selected components.

If *Cname1* is neither blank nor ALL, it is the name of an existing component. The energies are listed for up to 6 selected components named in *Cname1* to *Cname6*.

## Notes

The **PREENERGY** command prints out either the total energies of the entire model or the energies of the components depending on the *Cname1* specification.

Only existing components based on elements (defined with the **CM** command) are supported when component energies are listed.

This command applies to structural elements only.



## Menu Paths

This command cannot be accessed from a menu.

### /PREP7

Enters the model creation preprocessor.

SESSION: Processor Entry  
PREP7: Database

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Enters the general input data preprocessor (PREP7).

This command is valid only at the Begin Level.

## Menu Paths

Main Menu>Preprocessor

## PRERR

Prints SEPC and TEPC.

POST1: Listing

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

## Notes

Prints the percent error in structural energy norm (SEPC) and the thermal energy norm percent error (TEPC). Approximations of mesh discretization error associated with a solution are calculated for analyses having structural or thermal degrees of freedom.

The structural approximation is based on the energy error (which is similar in concept to the strain energy) and represents the error associated with the discrepancy between the calculated stress field and the globally continuous stress field (see [POST1 - Error Approximation Technique](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*). This discrepancy is due to the assumption in the elements that only the displacements are continuous at the nodes. The stress field is calculated from the displacements and should also be continuous, but generally is not.)

Thermal analyses may use any solid and shell thermal element having only temperature degrees of freedom. The thermal approximation is based on the total heat flow dissipation and represents the error associated with the discrepancy between the calculated nodal thermal flux within an element and a continuous global thermal flux. This continuous thermal flux is calculated with the normal nodal averaging procedure.

The volume (result label VOLU) is used to calculate the energy error per element (result label SERR for the structural energy error and TERR for the thermal energy error). These energy errors, along with the appropriate energy, are then used to calculate the percent error in energy norm (SEPC for structural and TEPC for thermal).

These percentages can be listed by the **PRERR** command, retrieved by the **\*GET** command (with labels SEPC and TEPC) for further calculations, and shown on the displacement display (**PLDISP**), as applicable.

For structural analyses, the maximum absolute value of nodal stress variation of any stress component for any node of an element (result item SDSG) is also calculated. Similarly, for thermal gradient components, TDSG is calculated. Minimum and maximum result bounds considering the possible effect of discretization error will be shown on contour displays (**PLNSOL**). For shell elements, the top surface location is used to produce a meaningful percentage value. SERR, TERR, SEPC, TEPC, SDSG, and TDSG will be updated whenever the nodal stresses or fluxes are recalculated.

If the energy error is a significant portion of the total energy, then the analysis should be repeated using a finer mesh to obtain a more accurate solution. The energy error is relative from problem to problem but will converge to a zero energy error as the mesh is refined. An automated adaptive meshing procedure using this energy error is described with the **ADAPT** macro.

**The following element- and material-type limitations apply:**

- Valid with most 2-D solid, 3-D solid, axisymmetric solid, or 3-D shell elements.
- The following element types are *not* valid: [SHELL28](#), [SHELL41](#), and [SOLID65](#).
- The model should have only structural or thermal degrees of freedom.
- The analysis must be linear (for both material and geometry).
- Multi-material (for example, composite) elements are not valid.
- Transition regions from one material to another are not valid (that is, the entire model should consist of one material).
- Anisotropic materials ([TB,ANEL](#)) are not considered.

## Menu Paths

**Main Menu>General Postproc>List Results>Percent Error**  
**Utility Menu>List>Results>Percent Error**

## **PRESOL**, *Item*, *Comp*

**Prints the solution results for elements.**

[POST1:Results](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Item*

Label identifying the item. Valid item labels are shown in [Table 246: PRESOL - Valid Item and Component Labels](#) (p. 1327) below. Some items also require a component label.

### *Comp*

Component of the item (if required). Valid component labels are shown in [Table 246: PRESOL - Valid Item and Component Labels](#) (p. 1327) below.

## Notes

Prints the solution results for the selected elements in the sorted sequence. For example, **PRESOL,S** prints the stress items *SX, SY, SZ, SXY, SYZ, and SXZ* for the node locations of the element. Component results are in the global Cartesian coordinate directions unless transformed (**RSYS**).

Shell elements print values at the top, then bottom of the element (or layer). If **KEYOPT(8) = 2** (for **SHELL181, SHELL208, SHELL209, SHELL281, or ELBOW290**), the results are printed in the order **TOP, BOT** and then **MID** of each element, (or layer). The **MID** value will be the actual value as written to the results file.

Items are listed as columns of a table versus element number. An exception occurs for item **ELEM** which uses an element format (all applicable line element results are listed per element) instead of a tabular format.

The **FORCE** command can be used to define which component of the nodal load is to be used (static, damping, inertia, or total). See the **ETABLE** and **PRETAB** commands for printing items not available through this command (such as line element results).

For PowerGraphics [**/GRAPHICS,POWER**], results are listed only for the element surface. The items marked with [1 (p. 1330)] are not supported by PowerGraphics.

**Table 246 PRESOL - Valid Item and Component Labels**

Item	Comp	Description
Valid item and component labels for element results are:		
S	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) stresses.
"	PRIN	Principal stresses (1, 2, 3), stress intensity (INT), and equivalent stress (EQV)
EPEL	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) elastic strains.
"	PRIN	Principal elastic strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV)
EPTH	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) thermal strains.
"	PRIN	Principal thermal strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV)
EPPL	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) plastic strains.
"	PRIN	Principal plastic strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV)
EPCR	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) creep strains.
"	PRIN	Principal creep strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV)
EPSW		Swelling strain.
EPTO	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) total mechanical strains (EPEL + EPPL + EPCR).
"	PRIN	Principal total mechanical strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV)
EPTT	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) total mechanical <i>and</i> thermal strains (EPEL + EPPL + EPCR + EPTH).

Item	Comp	Description
"	PRIN	Principal total mechanical and thermal strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV)
NL		Nonlinear items (SEPL, SRAT, HPRES, EPEQ, CREQ, PSV, PL-WK).
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
CDM	DMG	Damage variable
"	LM	Maximum previous strain energy for virgin material
FAIL		Failure criteria [ <a href="#">1</a> (p. 1330)][ <a href="#">2</a> (p. 1330)].
		<b>Default components:</b> Maximum of all failure criteria defined at current location (MAX), maximum strain (EMAX), maximum stress (SMAX), Tsai-Wu Strength Index (TWSI), inverse of Tsai-Wu Strength Ratio Index (TWSR).
		<b>Other available components:</b> Hashin Fiber Failure (HFIB), Hashin Matrix Failure (HMAT), Puck Fiber Failure (PFIB), Puck Matrix Failure (PMAT), and any user-defined failure criteria (USR1 through USR9) [ <a href="#">4</a> (p. 1330)].
		Issue the <b>FCTYP</b> command to activate or remove failure criteria.
FCMX		Maximum failure criterion over the entire element [ <a href="#">1</a> ][ <a href="#">2</a> ]. Components: Layer number where the maximum occurs (LAY), name of the maximum failure criterion (FC), and value of the maximum failure criterion (VAL).
SVAR	1,2,3, ... N	State variable.
GKS		Gasket component (X, XY, XZ) stress.
GKD		Gasket component (X, XY, XZ) total closure.
GKDI		Gasket component (X, XY, XZ) total inelastic closure.
GKTH		Gasket component (X, XY, XZ) thermal closure.
CONT		Contact items (STAT, PENE, PRES, SFRIC, STOT, SLIDE, GAP, FLUX, CNOS, FPRS). See component descriptions in <b>PLESOL</b> .
TG		Component (X, Y, Z) thermal gradients and vector sum (SUM).
TF		Component (X, Y, Z) thermal fluxes and vector sum (SUM).
PG		Component (X, Y, Z) pressure gradients and vector sum (SUM).
EF		Component (X, Y, Z) electric fields and vector sum (SUM).
D		Component (X, Y, Z) electric flux densities and vector sum (SUM).
H		Component (X, Y, Z) magnetic field intensities and vector sum (SUM).

Item	Comp	Description
B		Component (X, Y, Z) magnetic flux densities and vector sum (SUM).
FMAG		Component (X, Y, Z) electromagnetic forces and vector sum (SUM) [1].
P		Poynting vector components (X, Y, Z) and sum (SUM) [1].
F		Component (X, Y, Z) structural forces. Use <b>FORCE</b> for type [1][5].
M		Component (X, Y, Z) structural moments. Use <b>FORCE</b> for type [1].
HEAT		Heat flow. Use <b>FORCE</b> for type [1].
FLOW		Fluid flow. Use <b>FORCE</b> for type.
AMPS		Current flow [1]. Use <b>FORCE</b> for type.
CHRG		Charge [1]. Use <b>FORCE</b> for type.
FLUX		Magnetic flux [1].
CSG		Component (X, Y, Z) magnetic current segments. Use <b>FORCE</b> for type [1].
FORC		All available force items (F to CSG above). (10 maximum) [1].
BFE		Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
ELEM		All applicable element results (available only for structural line elements) [1].
SERR[3]		Structural error energy [1].
SDSG[3]		Absolute value of the maximum variation of any nodal stress component [1].
TERR[3]		Thermal error energy [1].
TDSG[3]		Absolute value of the maximum variation of any nodal thermal gradient component [1].
SENE		"Stiffness" energy or thermal heat dissipation. Same as TENE [1].
STEN		Elemental energy dissipation due to <b>stabilization</b> .
TENE		Thermal heat dissipation or "stiffness" energy. Same as SENE [1].
KENE		Kinetic energy [1].
JHEAT		Element Joule heat generation (coupled-field calculation) [1].
JS		Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM). [1].
JT		Total measurable current density in low-frequency electro-magnetic analyses. (Conduction current density in a low-

Item	Comp	Description
JC		frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM). [1].
JC		Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [1].
MRE		Magnetic Reynolds number [1].
VOLU		Volume of volume element [1].
CENT		Centroid X, Y, or Z location (based on shape function) in the active coordinate system [1].
LOCI		Integration point location
SMISC	<i>snum</i>	Element summable miscellaneous data value at sequence number <i>snum</i> (Shown in the Output Data section of each element description. See Chapter 4 of the <a href="#">Element Reference</a> ) [1].
NMISC	<i>snum</i>	Element nonsummable miscellaneous data value at sequence number <i>snum</i> (Shown in the Output Data section of each element description. See Chapter 4 of the <a href="#">Element Reference</a> ) [1].
TOPO		Densities used for topological optimization.
CAP	C0,X0,K0,ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; l1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2: shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.
EDPC	CSIG,CSTR	Material EDP creep model only (not including the cap model): equivalent creep stress; equivalent creep strain.
FICT	TEMP	Fictive temperature.
ESIG	X,Y,Z,XY,YZ,ZX	Components of Biot's effective stress.
"	1, 2, 3	Principal stresses of Biot's effective stress.
"	INT	Stress intensity of Biot's effective stress.
"	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
"	GPOR	Porosity due to void growth.
"	NPOR	Porosity due to void nucleation.
FFLX	X,Y,Z	Fluid flow flux in poromechanics.

1. Not supported by PowerGraphics
2. Works only if failure criteria information is provided. (For more information, see the documentation for the **FC** and **TB** commands.)
3. Some element- and material-type limitations apply. (For more information, see the documentation for the **PRERR** command.)
4. Works only if user failure criteria routine is provided.

5. Do not use the **PRESOL** command to obtain contact forces for contact elements. The force values reported by this command may not be accurate for these elements. Instead, use the **ETABLE** command to obtain contact force values.

## Menu Paths

**Main Menu>General Postproc>List Results>Element Solution**  
**Utility Menu>List>Results>Element Solution**

**PRETAB**, *Lab1, Lab2, Lab3, Lab4, Lab5, Lab6, Lab7, Lab8, Lab9*

Prints the element table items.

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Lab1, Lab2, Lab3, . . . , Lab9*

Print selected items. Valid labels are (blank) or any label as specified with the **ETABLE** command. Convenience labels may be used for *Lab1* to select groups of labels (10 labels maximum): GRP1 for first 10 stored items; GRP2 for items 11 to 20; GRP3 for items 21 to 30; GRP4 for items 31 to 40; GRP5 for items 41 to 50. Enter **ETABLE,STAT** command to list stored item order. If all labels are blank, print first 10 stored items (GRP1).

## Notes

Prints the items stored in the table defined with the **ETABLE** command. Item values will be listed for the selected elements in the sorted sequence [**ESORT**]. The **FORCE** command can be used to define which component of the nodal load is to be used (static, damping, inertia, or total).

Portions of this command are not supported by PowerGraphics [**/GRAPHICS,POWER**].

## Menu Paths

**Main Menu>General Postproc>Element Table>List Elem Table**  
**Main Menu>General Postproc>List Results>Elem Table Data**  
**Utility Menu>List>Results>Element Table Data**

**PRFAR**, *Lab, Opt, PHI1, PHI2, NPH1, THETA1, THETA2, NTHETA, VAL1, VAL2*

Prints electric far fields and far field parameters.

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

*Lab*

Print Parameter:

**EF**

Electric Field

**RCS**

Radar cross section

**RCSN**

Normalized radar cross section

**ANT**

Antenna parameters

**Opt**

As shown below, data entered in the Opt field will vary, depending on the print parameter.

**Lab**

As shown below, data entered in the Lab field will vary, depending on the print option (Opt).

**Table 247 Valid Lab Data Labels**

<i>Lab</i>	<i>Opt</i>
EF	SUM — the maximum total E-field (default) X — the maximum E-field in x-direction Y — the maximum E-field in y-direction Z — the maximum E-field in z-direction PHI — the maximum E-field in $\Phi$ -direction THETA — the maximum E-field in $\Theta$ -direction R — the maximum E-field in r-direction LHCP — the maximum left-hand circularly polarized component RHCP — the maximum right-hand circularly polarized component ARCP — the circular polarization axial ratio LW3X — the maximum dominant component for an x-polarized aperture with Ludwig's third definition of cross definition LW3Y — the maximum dominant component for an y-polarized aperture with Ludwig's third definition of cross definition ARCP — the linear polarization axial ratio
RCS and RCSN	TOTAL — Radar echo area (default) PP — $\Phi$ - $\Phi$ polarization PT — $\Phi$ - $\Theta$ polarization TP — $\Theta$ - $\Phi$ polarization TT — $\Theta$ - $\Theta$ polarization



<i>Lab</i>	<i>Opt</i>
	HH — H-H (horizontal – horizontal) polarization HV — H-V (Horizontal – Vertical) polarization VH — V-H (Vertical – Horizontal) polarization VV — V-V (Vertical – Vertical) polarization TE2D — 2-D TE incident plane wave TM2D — 2-D TM incident plane wave
ANT	EPCT – radiation pattern (default) UMAX – the maximum radiation intensity DGCT – directivity DGMX – peak directivity PGCT – gain PGMX – peak gain RGCT – realized gain RGMX – peak realized gain EFF – efficiency PRAD – radiated power

The following PHI1, PHI2, NPFI, THETA1, THETA2, and NTHETA arguments are used only with Lab = EF, RCS, RCSN and ANT and Opt = EPCT, DGCT, PGCT and RGCT .

**PHI1, PHI2**

Starting and ending  $\phi$  angles (degrees) in the spherical coordinate system. Defaults to 0.

**NPFI**

Number of divisions between the starting and ending  $\phi$  angles for data computations. Defaults to 0.

**THETA1, THETA2**

Starting and ending  $\theta$  angles (degrees) in the spherical coordinate system. Defaults to 0.

**NTHETA**

Number of divisions between the starting and ending  $\theta$  angles for data computations. Defaults to 0.

**VAL1**

Used only with *Lab* = ANT and *Opt* = PGCT, PGMX, RGCT, RGMX, EFF or *Lab* = RCS or RCSN and *Opt* = TE2D, TM2D. For *Lab* = ANT and *Opt* = PGCT, PGMX, RGCT, RGMX, EFF, *VAL1* = the incident power, *Pinc*, in watts. For *Lab* = RCS or RCSN and *Opt* = TE2D or TM2D, *VAL1* = thickness of the 3-D model in the z direction, defaults to 1.

**VAL2**

Used only with *Lab* = ANT and *Opt* = PGCT, PGMX, EFF. VAL2 = the  $|s_{11}|$  of the lossless port with a single propagation mode to calculate the power delivered to the radiating antenna structure, i.e.

$$P_{\text{del}} = P_{\text{inc}}(1 - |s_{11}|^2). \text{ Defaults to 0.}$$

**Notes**

**PRFAR** prints electric far fields and far field parameters as determined by the equivalent source principle. Use this command to print electric far field, radar cross section, antenna parameters.

See [Spherical Coordinates in Hi-Frequency Electromagnetic Analysis Guide](#).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

This command cannot be accessed from a menu.

**PRI2**, P51X, Z1, Z2

Creates a polygonal area or a prism volume by vertices (GUI).

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

Creates a polygonal area or a prism volume using the vertices as input. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if graphical picking is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

For polygons, the **PRI2** command will appear in the log file as **PRI2,P51X,0,0,0,0** preceded by **FITEM** commands that define the vertices (in global Cartesian coordinates). For prisms, the **PRI2** command will appear in the log file as **PRI2,P51X** preceded by **FITEM** commands that define the vertices and the Z-end of the prism.

See the **RPOLY**, **POLY**, **RPRISM**, **PRISM**, and **RPR4** commands for other ways to create polygons and prisms.

**Menu Paths**

Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Vertices

Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Vertices

## PRIM

Specifies "Solid model primitives" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This is a status [[STAT](#)] topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the [STAT](#) command should immediately follow this command.

### Menu Paths

**Utility Menu>List>Status>Preprocessor>Solid Model**

---

## PRINT

Specifies "Print settings" as the subsequent status topic.

[POST1: Status](#)

[POST26: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This is a status [[STAT](#)] topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the [STAT](#) command should immediately follow this command.

### Menu Paths

**Utility Menu>List>Status>General Postproc>List Results**

**Utility Menu>List>Status>TimeHist Postproc>List**

---

**\*PRINT**, *Matrix, Fname*

**Prints the matrix values to a file.**

[APDL: Matrix Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### **Matrix**

Name of matrix or vector to print. Must be specified.

### **Fname**

File name. If blank, matrix is written to the output file.

## Notes

The matrix may be a dense matrix (**\*DMAT**), a sparse matrix (**\*SMAT**), or a vector (**\*VEC**). Only the non-zero entries of the matrix are printed.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**PRISM**, *Z1, Z2*

**Creates a prism volume based on working plane coordinate pairs.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Z1, Z2*

Working plane Z coordinates of the top and bottom of the prism.

## Notes

Defines a prism volume based on the working plane. The top and bottom areas will each be defined with NPT keypoints and NPT lines, where NPT (must be at least 3) is the number of coordinate pairs defined with **PTXY** command. Also, a line will be defined between each point pair on the top and bottom face. See the **RPRISM** and **RPR4** commands for other ways to create prisms.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## PRITER

Prints solution summary data.

[POST1: Listing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Prints solution summary data (such as time step size, number of equilibrium iterations, convergence values, etc.) from a static or full transient analysis. All other analyses print zeros for the data.

## Menu Paths

**Main Menu>General Postproc>List Results>Iteration Summry**  
**Utility Menu>List>Results>Iteration Summry**

**PRJSOL**, *Item*, *Comp*

Prints joint element output.

[POST1: Results](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Item*

Label identifying the item. Some items also require a component label.

#### **DISP**

Relative displacements.

#### **ROT**

Relative rotations.

#### **VEL**

Relative linear velocities.

#### **OMG**

Relative angular velocities.

#### **ACC**

Relative linear accelerations.

#### **DMG**

Relative angular accelerations.

#### **SMISC**

Summable miscellaneous quantities.

### *Comp*

Component of the item (if required). For *Item* = DISP, ROT, VEL, OMG, ACC, and DMG, enter the direction label, X, Y, or Z. For *Item* = SMISC, enter a valid number.

## Notes

Prints element output for the **MPC184** joint element. The joint element quantities printed are the values for the free or unconstrained relative degrees of freedom.

This command is valid in POST1 only.

## Menu Paths

**This command cannot be accessed from a menu.**

**PRNEAR**, *Lab*, *Opt*, *KCN*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*

**Prints the electric field in the near zone exterior to the equivalent source surface.**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

### *Lab*

Print the maximum E-file:

#### **POINT**

at the point (x,y,z)

#### **SPHERE**

on the spherical structure

#### **PATH**

along the path

### *Opt*

SUM — the maximum total E-field (default)

X — the maximum E-field in x-direction

Y — the maximum E-field in y-direction

Z — the maximum E-field in z-direction

PHI — the maximum E-field in  $\Phi$ -direction

THETA — the maximum E-field in  $\Theta$ -direction

R — the maximum E-field in r-direction

LHCP — the maximum left-hand circularly polarized component

RHCP — the maximum right-hand circularly polarized component

ARCP — the circular polarization axial ratio

LW3X — the maximum dominant component for an x-polarized aperture with Ludwig's third definition of cross definition

LW3Y — the maximum dominant component for an y-polarized aperture with Ludwig's third definition of cross definition

ARCP — the linear polarization axial ratio

### *KCN*

KCN is the coordinate system reference number. It may be 0 (Cartesian) or any previously defined local coordinate system number (>10). Defaults to 0.

**VAL1, VAL2, VAL3, . . . , VAL7**

For *Lab* = POINT:

#### **VAL1**

x coordinate value

#### **VAL2**

y coordinate value

**VAL3**

z coordinate value

For  $L_{AB} = \text{SPHERE}$ :

**VAL1**

the radius of spherical surface in spherical coordinate system.

**VAL2**

starting  $\phi$  angle (degree) in the spherical coordinate system. Defaults to 0.

**VAL3**

ending  $\phi$  angle (degree) in the spherical coordinate system. Defaults to 0.

**VAL4**

Number of divisions between the starting and ending  $\phi$  angles for data computations. Defaults to 0.

**VAL5**

starting  $\theta$  angle (degree) in the spherical coordinate system. Defaults to 0.

**VAL6**

ending  $\theta$  angle (degree) in the spherical coordinate system. Defaults to 0.

**VAL7**

Number of divisions between the starting and ending  $\theta$  angles for data computations. Defaults to 0.

For  $L_{ab} = \text{PATH}$ , **PRNEAR** computes the electric field for the path data points for the path currently defined by the **PATH** and **PPATH** commands.

## Notes

**PRNEAR** uses the equivalent source principle to calculate the electric field in the near zone exterior to the equivalent source surface (flagged with the Maxwell surface flag in the preprocessor) for one of the following locations:

- A point X, Y, Z in the KAN coordinate system
- A spherical surface in the KAN coordinate system
- A path defined by the **PATH** and **PPATH** commands

To list the electromagnetic field results for a path, use the **PRPATH** command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

**PRNLD**, *Lab, TOL, Item***Prints the summed element nodal loads.**

POST1: Results

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; EM EH DY PP &lt;&gt; EME MFS

**Lab**

Nodal reaction load type. If blank, use the first ten of all available labels. Valid labels are: Structural force labels: FX, FY or FZ (forces); F (includes FX, FY and FZ); MX, MY or MZ (moments); M (includes MX, MY and MZ). Thermal force labels: HEAT, HBOT, HE2, HE3, . . . , HTOP (heat flow). Fluid force labels: FLOW (fluid flow); VFX, VFY and VFZ (fluid "forces"); VF (includes VFX, VFY and VFZ). Electric force labels: AMPS (current flow); CHRG (charge); CURT (current); VLTG (voltage drop). Magnetic force labels: FLUX (magnetic flux); CSGX, CSGY, CSGZ (magnetic current segments); CSG (includes CSGX, CSGY and CSGZ).

**TOL**

Tolerance value about zero within which loads are not printed. Defaults to 1.0E-9 times the absolute value of the maximum load on the selected nodes. If zero, print all nodal loads.

**ITEM**

Selected set of nodes.

**(blank)**

Prints the summed element nodal loads for all selected nodes (default), excluding contact elements.

**CONT**

Prints the summed element nodal loads for contact nodes only.

**BOTH**

Prints the summed element nodal loads for all selected nodes, including contact nodes.

**Notes**

Prints the summed element nodal loads (forces, moments, heat flows, flux, etc.) for the selected nodes in the sorted sequence. Results are in the global Cartesian coordinate directions unless transformed [**RSYS**]. Zero values (within a tolerance range) are not printed. Loads applied to a constrained degree of freedom are not included. The **FORCE** command can be used to define which component of the nodal load is to be used (static, damping, inertia, or total).

By default, **PRNLD** excludes elements **TARGE169** - **CONTA177**. **PRNLD,,,CONT** will only account for nodal forces on selected contact elements (**CONTA171** - **CONTA177**). **PRNLD,,,BOTH** will account for nodal forces on all selected nodes, including contact nodes.

**Using PRNLD in a Spectrum or PSD Analysis (ANTYPE, SPECTR)**

When using **PRNLD** in a spectrum analysis after the combination file has been input (**/INPUT,,MCOM**), or in a PSD analysis when postprocessing 1-sigma results (loadstep 3, 4, or 5), the following message will display in the printout header:

```
(Spectrum analysis summation is used)
```

This message means that the summation of the element nodal forces is performed prior to the combination of those forces. In this case, **RSYS** does not apply. The forces are in the nodal coordinate systems, and the vector sum is always printed in the global coordinate system.



Because modal displacements cannot be used to calculate contact element nodal forces, *ITEM* does not apply to spectrum and PSD analyses.

## Menu Paths

**Main Menu>General Postproc>List Results>Nodal Loads**  
**Utility Menu>List>Results>Nodal Loads**

## PRNSOL, *Item*, *Comp*

Prints the nodal solution results.

POST1: Results

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Item*

Label identifying the item. Valid item labels are shown in [Table 248: PRNSOL - Valid Item and Component Labels](#) (p. 1342) below. Some items also require a component label.

### *Comp*

Component of the item (if required). Valid component labels are shown in [Table 248: PRNSOL - Valid Item and Component Labels](#) (p. 1342) below. Defaults to COMP.

## Notes

Prints the nodal solution results for the selected nodes in the sorted sequence. For example, **PRNSOL,U,X** prints the X component of displacement vector U (that is, the UX degree of freedom). Component results are in the global Cartesian coordinate directions unless transformed (**RSYS**). Various element results also depend upon the recalculation method and the selected results location (**AVPRIN**, **RSYS**, **LAYER**, **SHELL**, and **NSEL**). If the **LAYER** command is issued, then the resulting output is listed in full graphics mode (**/GRAPHICS,FULL**). You can use the **FORCE** command to define which component of the nodal load (static, damping, inertia, or total) should be used.

PowerGraphics can affect your nodal solution listings. For PowerGraphics (**/GRAPHICS,POWER**), results are listed only for the model exterior surfaces.

When shell element types are present, results are output on a surface-by-surface basis. For shell elements, such as **SHELL181** or **SHELL281**, and for **ELBOW290**, printed output is for both the top and bottom surfaces. For solid elements such as **SOLID185**, the output is averaged for each surface and printed as follows:

- For a node at a vertex, three lines are output (one printed line for each surface).
- For a node on an edge, two lines are output (one printed line for each surface).
- For nodes on a face, one value is output.
- For nodes interior to the volume, no printed values are output.

If a node is common to more than one element, or if a geometric discontinuity exists, several conflicting listings may result. For example, a corner node incorporating results from solid elements and shell elements could yield as many as nine different results; the printed output would be averages at the top and bottom for the three shell surfaces plus averages at the three surfaces for the solid, for a total of nine lines of output. ANSYS does not average result listings across geometric discontinuities when shell element types are present. It is important to analyze the listings at discontinuities to ascertain the significance of each set of data.

The printed output for full graphics (**/GRAPHICS,FULL**) follows the standard ANSYS convention of averaging results at the node. For shell elements, the default for display is TOP so that the results for the top of the shell are averaged with the other elements attached to that node.

If an **NSORT, ESORT** or **/ESHAPE** command is issued with PowerGraphics activated, then the **PRNSOL** listings will be the same as in full graphics mode (**/GRAPHICS,FULL**). The items marked with [2 (p. 1345)] are not supported by PowerGraphics. To print midside nodes, you must first issue an **/EFACET,2** command.

**Table 248 PRNSOL - Valid Item and Component Labels**

Item	Comp	Description
<b>Valid item and component labels for nodal degree of freedom results are:</b>		
U	X, Y, Z	X, Y, or Z structural displacement.
"	COMP	X, Y, and Z structural displacements and vector sum.
ROT	X, Y, Z	X, Y, or Z structural rotation.
"	COMP	X, Y, and Z structural rotations and vector sum.
TEMP [1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity in a fluid analysis, or X, Y, or Z velocity in an ANSYS LS-DYNA analysis.
"	COMP	X, Y, and Z fluid velocity and vector sum in a fluid analysis, or X, Y, and Z velocity and vector sum in an ANSYS LS-DYNA analysis.
A	X, Y, Z	X, Y, or Z magnetic vector potential in an electromagnetic analysis, or X, Y, or Z acceleration in an ANSYS LS-DYNA analysis.
"	COMP	X, Y, and Z magnetic vector potential and vector sum in an electromagnetic analysis, or X, Y, and Z acceleration and vector sum in an ANSYS LS- DYNA analysis.
VEL	X, Y, Z	X, Y, or Z velocity in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
"	COMP	X, Y, and Z velocity and vector sum in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
ACC	X, Y, Z	X, Y, or Z acceleration in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
"	COMP	X, Y, and Z acceleration and vector sum in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
OMG	X, Y, Z	X, Y, or Z rotational velocity in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
"	COMP	X, Y, and Z rotational velocity and vector sum in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).

Item	Comp	Description
DMG	X, Y, Z	X, Y, or Z rotational acceleration in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
"	COMP	X, Y, and Z rotational acceleration and vector sum in a structural transient dynamic analysis ( <b>ANTYPE,TRANS</b> ).
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.
SP0 <sub>n</sub>		Mass fraction of species <i>n</i> , where <i>n</i> = 1 to 6 (FLOTRAN). If a species is given a user-defined name [ <b>MSSPEC</b> ], use that name instead of SP0 <sub>n</sub> [2].
DOF		All available degree of freedom labels (10 maximum).
FICT	TEMP	Fictive temperature.
<b>Valid item and component labels for element results are:</b>		
S	COMP	X, Y, Z, XY, YZ, and XZ component stresses.
"	PRIN	S1, S2, S3 principal stresses, SINT stress intensity, and SEQV equivalent stress.
EPEL	COMP	Component elastic strains.
"	PRIN	Principal elastic strains, strain intensity, and equivalent strain.
"	FAIL	Maximum Strain Failure Criteria. [2][4]
EPTH	COMP	Component thermal strains.
"	PRIN	Principal thermal strains, strain intensity, and equivalent strain.
EPPL	COMP	Component plastic strains.
"	PRIN	Principal plastic strains, strain intensity, and equivalent strain.
EPCR	COMP	Component creep strains.
"	PRIN	Principal creep strains, strain intensity, and equivalent strain.
EPSW		Swelling strain.
EPTO	COMP	Component total mechanical strains (EPEL + EPPL + EPCR).
"	PRIN	Principal total mechanical strains, strain intensity, and equivalent strain.
EPTT	COMP	Component total mechanical <i>and</i> thermal strains (EPEL + EPPL + EPCR + EPTH).
"	PRIN	Principal total mechanical <i>and</i> thermal strains, strain intensity, and equivalent strain.
NL		Nonlinear items (SEPL, SRAT, HPRES, EPEQ, CREQ, PSV, PL-WK).
SEND		Elastic, plastic, and creep strain energy density.
CDM	DMG	Damage variable

Item	Comp	Description
"	LM	Maximum previous strain energy for virgin material
FAIL		Failure criteria [2 (p. 1345)][4 (p. 1346)].  <b>Default components:</b> Maximum of all failure criteria defined at current location (MAX), maximum strain (EMAX), maximum stress (SMAX), Tsai-Wu Strength Index (TWSI), inverse of Tsai-Wu Strength Ratio Index (TWSR).  <b>Other available components:</b> Hashin Fiber Failure (HFIB), Hashin Matrix Failure (HMAT), Puck Fiber Failure (PFIB), Puck Matrix Failure (PMAT), and any user-defined failure criteria (USR1 through USR9) [5 (p. 1346)].  To activate or remove failure criteria, issue the <b>FCTYP</b> command.
SVAR	1, 2, 3, ... N	State variable.
GKS	COMP	X, XY, XZ component gasket stress.
GKD	COMP	X, XY, XZ component gasket total closure.
GKDI	COMP	X, XY, XZ component gasket total inelastic closure.
GKTH	COMP	X, XY, XZ component thermal closure.
SS	X, XY, XZ	Interface traction (stress).
SD	X, XY, XZ	Interface separation
CONT		Contact items (STAT[3], PENE, PRES, SFRIC, STOT, SLIDE, GAP, FLUX, CNOS, FPRS). See component descriptions in <b>PLNSOL</b> .
TG	COMP	Component thermal gradients and vector sum.
TF	COMP	Component thermal fluxes and vector sum.
PG	COMP	Component pressure gradients and vector sum.
EF	COMP	Component electric fields and vector sum.
D	COMP	Component electric flux densities and vector sum.
H	COMP	Component magnetic field intensities and vector sum.
B	COMP	Component magnetic flux densities and vector sum.
FMAG	COMP	Component electromagnetic forces and vector sum [2].
JC	COMP	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [2].
BFE		Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
TOPO		Densities used for topological optimization.
CAP	C0,X0,K0,ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; I1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2:

Item	Comp	Description
		shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.
EDPC	CSIG,CSTR	Material EDP creep model only (not including the cap model): Equivalent creep stress; equivalent creep strain.
ESIG	X,Y,Z,XY,YZ,ZX	Components of Biot's effective stress.
"	1, 2, 3	Principal stresses of Biot's effective stress.
"	INT	Stress intensity of Biot's effective stress.
"	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
"	GPOR	Porosity due to void growth.
"	NPOR	Porosity due to void nucleation.
FFLX	X,Y,Z	Fluid flow flux in poromechanics.

**Valid item and component labels for FLOTRAN nodal results are:**

TTOT	Total temperature.
HFLU	Heat flux.
HFLM	Heat transfer (film) coefficient.
COND	Fluid laminar conductivity.
PCOE	Pressure coefficient.
PTOT	Total (stagnation) pressure.
MACH	Mach number.
STRM	Stream function. (2-D applications only.)
DENS	Fluid density.
VISC	Fluid laminar viscosity.
SPHT	Specific heat.
EVIS	Fluid effective viscosity.
CMUV	Turbulent viscosity coefficient.
ECON	Fluid effective conductivity.
YPLU	$Y^+$ , a turbulent law of the wall parameter.
TAUW	Shear stress at the wall.
SFTS	Surface tension coefficient.
LMD $n$	Laminar mass diffusion coefficient for species $n$ , where $n = 1$ to 6.
EMD $n$	Effective mass diffusion coefficient for species $n$ , where $n = 1$ to 6.
RDFL	Radiation heat flux.

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . . , TTOP instead of TEMP.
2. Not supported by PowerGraphics

3. For the CONT items for elements [CONTA171](#) through [CONTA177](#), the reported data is averaged across the element. To obtain a more meaningful STAT value, use the **PRESOL** command.
4. Works only if failure criteria information is provided. (For more information, see the documentation for the **FC** and **TB** commands.)
5. Works only if user failure criteria routine is provided.

## Menu Paths

**Main Menu>General Postproc>List Results>Nodal Solution**  
**Main Menu>General Postproc>List Results>Sorted Listing>Sort Nodes**  
**Utility Menu>List>Results>Nodal Solution**

**PROD**, *IR, IA, IB, IC, Name, --, --, FACTA, FACTB, FACTC*

**Multiplies variables.**

POST26: Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**IR**

Arbitrary reference number assigned to the resulting variable (2 to NV [[NUMVAR](#)]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

**IA, IB, IC**

Reference numbers of the three variables to be operated on. If only two leave *IC* blank. If only one, leave *IB* blank also.

**Name**

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

**--, --**

Unused fields

**FACTA, FACTB, FACTC**

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

## Notes

Multiplies variables (up to three at once) according to the operation:

$$IR = (FACTA \times IA) \times (FACTB \times IB) \times (FACTC \times IC)$$

## Menu Paths

**Main Menu>TimeHist Postpro>Math Operations>Multiply**

## PRORB

Prints the orbital motion characteristics of a rotating structure

POST1: Results

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### Notes

When a structure is rotating and the Coriolis or gyroscopic effect is taken into account (**CORIOLIS**), nodes lying on the rotation axis generally exhibit an elliptical orbital motion. The **PRORB** command prints out the orbit characteristics A, B, PSI, PHI, YMAX and ZMAX of each rotating node, where

A is the semi-major axis.

B is the semi-minor axis.

PSI is the angle between local y axis and major axis.

PHI is the angle between initial position (t = 0) and major axis.

YMAX is the maximum displacement along local y axis.

ZMAX is the maximum displacement along local z axis.

Angles PSI and PHI are in degrees and within the range of -180 through +180. For more information about orbit definition, see [Orbits](#) in the *Advanced Analysis Techniques Guide*.

To *display* the characteristics of the orbital path traversed by each node, issue the **PLOBR** command.

The **PRORB** command is valid for line elements (such as [BEAM188](#), [BEAM189](#), [PIPE288](#), and [PIPE289](#)).

Your model must also involve a rotational velocity (**OMEGA** or **CMOMEGA**) with Coriolis enabled in a *stationary* reference frame (**CORIOLIS,,,,RefFrame = ON**).

A **SET** command should be issued after **PRORB** to ensure proper output for subsequent postprocessing commands.

The coordinate system for displaying nodal results must be global Cartesian (**RSYS,KCN = 0**).

### Menu Paths

Main Menu>General Postproc>Rotor Dynamics>Plot orbit

**PRPATH**, *Lab1, Lab2, Lab3, Lab4, Lab5, Lab6*

Prints path items along a geometry path.

POST1: Path Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Lab1, Lab2, Lab3, . . . , Lab6*

Labels identifying the path items to be printed. Up to six items may be printed at a time. Predefined path geometry items XG, YZ, ZG, and S [**PDEF**] may also be printed.

## Notes

Prints path items with respect to a geometry path (as defined by the **PATH** and **PPATH** commands). Path items and their labels must have been defined with the **PDEF**, **PVECT**, **PCALC**, **PDOT**, **PCROSS**, or **PRNEAR** commands. Path items may also be displayed with the **PLPATH** and **PLPAGM** commands. See the **PRANGE** command for range control of the path.

## Menu Paths

**Main Menu>General Postproc>List Results>Path Items**

**Main Menu>General Postproc>Path Operations>Plot Path Item>List Path Items**

**Utility Menu>List>Results>Path Data**

## PRRFOR, Lab

Used with the **FORCE** command. Prints the constrained node reaction solution.

[POST1: Results](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### Lab

Nodal reaction load type. If blank, use the first ten of all available labels. Valid labels are: Structural force labels: FX, FY or FZ (forces); F (FX, FY and FZ); MX, MY or MZ (moments); M (MX, MY and MZ). Thermal force labels: HEAT, HBOT, HE2, HE3, . . . , HTOP (heat flow). Fluid force labels: FLOW (fluid flow); VFX, VFY and VFZ (fluid "forces"); VF (VFX, VFY and VFZ). Electric force labels: AMPS (current flow); CHRGM (charge); CURT (current); VLTG (voltage drop). Magnetic force labels: FLUX (magnetic flux); CSGX, CSGY, CSGZ (magnetic current segments); CSG (CSGX, CSGY and CSGZ); CURT (current), VLTG (voltage drop).

## Notes

**PRRFOR** has the same functionality as the **PRRSOL** command; use **PRRFOR** instead of **PRRSOL** when a **FORCE** command has been issued.

**PRRFOR** cannot be used when a squaring operation has been carried out on a load case. **PRRSOL** values are correct in this case, because they are squared after summing the nodal forces.

## Using PRRFOR in a Spectrum or PSD Analysis (ANTYPE, SPECTR)

When using **PRRFOR** in a spectrum analysis after the combination file has been input (**/INPUT,,MCOM**), or in a PSD analysis when postprocessing 1-sigma results (loadstep 3, 4, or 5), the following message will display in the printout header:

```
(Spectrum analysis summation is used)
```

This message means that the summation of the element nodal forces is performed prior to the combination of those forces. In this case, **RSYS** does not apply, and the reaction forces are in the nodal coordinate systems.

## Menu Paths

**This command cannot be accessed from a menu.**



**PRRSOL**, *Lab*

**Prints the constrained node reaction solution.**

[POST1: Results](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

**Lab**

Nodal reaction load type. If blank, use the first ten of all available labels. Valid labels are: Structural force labels: FX, FY or FZ (forces); F (FX, FY and FZ); MX, MY or MZ (moments); M (MX, MY and MZ); BMOM (bimoments). Thermal force labels: HEAT, HBOT, HE2, HE3, . . . , HTOP (heat flow). Fluid force labels: FLOW (fluid flow); VFX, VFY and VFZ (fluid "forces"); VF (VFX, VFY and VFZ). Electric force labels: AMPS (current flow); CHRNG (charge); CURT (current); VLTG (voltage drop). Magnetic force labels: FLUX (magnetic flux); CSGX, CSGY, CSGZ (magnetic current segments); CSG (CSGX, CSGY and CSGZ); CURT (current), VLTG (voltage drop).

**Notes**

Prints the constrained node reaction solution for the selected nodes in the sorted sequence. For coupled nodes and nodes in constraint equations, the sum of all reactions in the coupled or constraint equation set appears at the primary node of the set. Results are in the global Cartesian coordinate directions unless transformed [[RSYS](#)]. **PRRSOL** is not valid if any load is applied to a constrained node in the direction of the constraint *and* any of the following is true: a) **LCOPER** has been used, or b) **LCASE** has been used to read from a load case file or c) the applied loads and constraints in the database are not the ones used to create the results data being processed.

Use **PRRFOR** instead of **PRRSOL** with the **FORCE** command.

**Menu Paths**

**Main Menu>General Postproc>List Results>Reaction Solu  
Utility Menu>List>Results>Reaction Solution**

**PRSECT**, *RHO, KBR*

**Calculates and prints linearized stresses along a section path.**

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

**RHO**

In-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If zero (or blank), a plane or 3-D structure is assumed. If nonzero, an axisymmetric structure is assumed. Use any large number (or -1) for an axisymmetric straight section.

**KBR**

Through-thickness bending stresses key for an axisymmetric analysis ( $RHO \neq 0$ ):

**0**

Include the thickness-direction bending stresses.

**1**

Ignore the thickness-direction bending stresses.

## 2

Include the thickness-direction bending stress using the same formula as the Y (axial direction ) bending stress. Also use the same formula for the shear stress.

## Notes

You may choose to linearize the stresses through a section and separate them into categories for various code calculations. **PRSECT** calculates and reports linearized stresses along a section path. The linearized stresses are also separated into membrane, bending, membrane plus bending, peak, and total stress categories.

First, define your section path using the **PATH** and **PPATH** (with the **NODE** option) commands. Your path must lie entirely within the selected set of elements (that is, there must be no element gaps along the path). **PATH** and **PPATH** are used only to retrieve the two end nodes. The path data is not retained. The section path is defined by the two end nodes, and by 47 intermediate points that are automatically determined by linear interpolation in the active display coordinate system [**DSYS**]. The number and location of the intermediate points are not affected by the number of divisions set by **PATH,,,,,nDiv**.

Your linearized component stress values are obtained by interpolating each element's average corner nodal values along the section path points within each path element. **PRSECT** reports the linearized component and principal stresses for each stress category at the beginning, mid-length, and end of the section path. **PRPATH** can be used to report the total stresses at the intermediate points.

Section paths may be through any set of solid (2-D plane, 2-D axisymmetric or 3-D) elements. However, section paths are usually defined to be through the thickness of the structure and normal to the inner and outer structure surfaces. Section paths (in-plane only) may also be defined for shell element structures. See the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details.

If the **RHO** option is set to indicate the axisymmetric option (non-zero), **PRSECT** reports the linearized stresses in the section coordinates (SX – along the path, SY – normal to the path, and SZ – hoop direction). If the **RHO** option is set to indicate the 2-D planar or 3-D option (zero or blank), **PRSECT** reports the linearized stresses in the active results coordinate system [**RSYS**]. If the **RHO** option is zero or blank and either **RSYS**, **SOLU** or **RSYS**, -1 are active, the linearized stresses are calculated and reported in the global Cartesian coordinate system. It is recommended that linearized stress calculations be performed in a rectangular coordinate system. Principle stresses are recalculated from the component stresses and are invariant with the coordinate system as long as SX is in the same direction at all points along the defined path. The **PLSECT** command displays the linearized stresses in the same coordinate system as reported by **PRSECT**.

Stress components through the section are linearized by a line integral method and separated into constant membrane stresses, bending stresses varying linearly between end points, and peak stresses (defined as the difference between the actual (total) stress and the membrane plus bending combination).

For nonaxisymmetric structures, the bending stresses are calculated such that the neutral axis is at the midpoint of the path. Axisymmetric results include the effects of both the radius of revolution (automatically determined from the node locations) and the in-plane average radius of curvature of the section surfaces (user input).

For axisymmetric cases, ANSYS calculates the linearized bending stress in the through-thickness direction as the difference between the total outer fiber stress and the membrane stress if  $KBR = 1$ . The calculation method may be conservative for locations with a highly nonlinear variation of stress in the through-thickness direction. Alternatively, you can specify  $KBR = 2$  to calculate the bending stress using the same method and formula as the Y (axial direction) bending stress. For more information, see the discussion of *axisymmetric cases* (specifically Equation 19–40) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

Portions of this command are not supported by PowerGraphics [/GRAPHICS,POWER].

## Menu Paths

**Main Menu>General Postproc>List Results>Linearized Strs**  
**Main Menu>General Postproc>Path Operations>List Linearized**  
**Utility Menu>List>Results>Linearized Stresses**

---

**PRSYZ**, *Fname*, *Ext*, *Lab*, *Opt*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*, *VAL10*, *VAL11*, *VAL12*, *VAL13*, *VAL14*, *VAL15*, *VAL16*

**Converts and lists scattering, admittance, or impedance parameters.**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

### ***Fname***

File name and directory path of a Touchstone file (248 characters maximum, including the characters needed for the directory path). A Touchstone file may be created in ANSYS by performing a frequency sweep using the SPSWP command macro, or may it be supplied from another source. An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

### ***Ext***

Extension of Touchstone file (.snp where n is the number of ports).

### ***Lab***

Output parameters:

**S**

Scattering parameters (default)

**Y**

Admittance parameters

**Z**

Impedance parameters

**SNRM**

Renormalized S-parameters

### ***Opt***

Format of Touchstone file:

**MA**

Magnitude and angle

**DB**

Magnitude and angle with magnitude in dB units. Only available for scattering parameters (*Lab* = S).

**RI**

Real and imaginary parts (default).

***VAL1*, *VAL2*, *VAL3*, . . . , *VAL16***

Port characteristic impedance (ohms). Defaults to 50 ohms. Only used for *Lab* = SNRM.

## Notes

For  $Lab = SNRM$ , the renormalized S-parameters are output to a new Touchstone file with the name  $Fname\_Sparm.snp$ . Up to 16 ports can be renormalized.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>General Postproc>List Results>Field Extension>S,Y,Z Parameters**

---

### **PRTIME**, $TMIN$ , $TMAX$

**Defines the time range for which data are to be listed.**

[POST26: Listing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### **$TMIN$**

Minimum time (defaults to the first point stored).

#### **$TMAX$**

Maximum time (defaults to the last point stored).

## Command Default

Use the previously defined range [**TIMERANGE**].

## Notes

Defines the time (or frequency) range (within the range stored) for which data are to be listed.

## Menu Paths

**Main Menu>TimeHist Postpro>Settings>List**

---

### **PRVAR**, $NVAR1$ , $NVAR2$ , $NVAR3$ , $NVAR4$ , $NVAR5$ , $NVAR6$

**Lists variables vs. time (or frequency).**

[POST26: Listing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### **$NVAR1$ , $NVAR2$ , $NVAR3$ , . . . , $NVAR6$**

Variables to be displayed, defined either by the reference number or a unique thirty-two character name. If duplicate names are used the command will print the data for the lowest-numbered variable with that name.

## Notes

Lists variables vs. time (or frequency). Up to six variables may be listed across the line. Time column output format can be changed using the **/FORMAT** command arguments *Ftype*, *NWIDTH*, and *DSIGNF*.

## Menu Paths

Main Menu>TimeHist Postpro>List Variables

**PRVAROPT**, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*

Lists up to ten optimization parameters.

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Lab1*, *Lab2*, *Lab3*, . . . , *Lab10*

Names of the parameters to be listed.

## Notes

Lists up to ten optimization parameters at a time. Parameters are listed vs. the **XVAROPT** parameter (defaults to set number) in the order corresponding to an ascending order of the **XVAROPT** parameter.

## Menu Paths

Main Menu>Design Opt>Design Sets>Graphs/Tables

**PRVECT**, *Item*, *Lab2*, *Lab3*, *LabP*

Prints results as vector magnitude and direction cosines.

POST1: Results

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Item*

Predefined vector item (from [Table 249: PRVECT - Valid Item and Component Labels \(p. 1354\)](#) below) or a label identifying the i-component of a user-defined vector.

*Lab2*

Label identifying the j-component of a user-defined vector. In most cases, this value must be blank if *Item* is selected from [Table 249: PRVECT - Valid Item and Component Labels \(p. 1354\)](#). Individual principal stresses (*Item* = S) or principal strains (*Item* = EP<sub>xx</sub>) may be printed by specifying the value as 1, 2, or 3.

*Lab3*

Label identifying the k-component of a user-defined vector. Must be blank if *Item* is selected from list below or for 2-D user defined vector.

*LabP*

Label assigned to resultant vector for printout labeling (defaults to *Item*).

## Notes

Prints various solution results as vector magnitude and direction cosines for the selected nodes and/or elements. For example, **PRVECT,U** prints the displacement magnitude and its direction cosines for all selected nodes. For nodal degree of freedom vector results, direction cosines are with respect to the results coordinate system RSYS. For element results, direction cosines are with respect to the global Cartesian system. Item components may be printed with the **PRNSOL** command. Various results also depend upon the recalculation method and the selected results location [**LAYER, SHELL, NSEL, and ESEL**]. Items may be selected from a set of recognized vector labels (*Item*) or a vector may be defined from up to three scalar labels (*Item,Lab2,Lab3*). Scalar labels may be user-defined with the **ETABLE** command.

Portions of this command are not supported by PowerGraphics [**/GRAPHICS,POWER**].

**Table 249 PRVECT - Valid Item and Component Labels**

Item	Comp	Description
Valid item labels for nodal degree of freedom vector results are:		
U		Structural displacement vector magnitude and direction cosines.
ROT		Structural rotation vector magnitude and direction cosines.
V		Velocity vector magnitude and direction cosines.
A		Magnetic vector potential vector magnitude and direction cosines.
Valid item labels for element results are:		
S		Principal stresses and direction cosines.
EPTO		Principal total strains (EPEL + EPPL + EPCR) and direction cosines.
EPEL		Principal elastic strains and direction cosines.
EPPL		Principal plastic strains and direction cosines.
EPCR		Principal creep strains and direction cosines.
EPTH		Principal thermal strains and direction cosines.
TG		Thermal gradient vector sum and direction cosines.
TF		Thermal flux vector sum and direction cosines.
PG		Pressure gradient vector sum and direction cosines.
EF		Electric field vector sum and direction cosines.
D		Electric flux density vector sum and direction cosines.
H		Magnetic field intensity vector sum and direction cosines. If <i>Lab2</i> is blank, then <i>Item</i> is interpreted as one of the predefined labels. Otherwise, <i>Item</i> is interpreted as a user-defined <b>ET</b> label and ANSYS will request a non-blank <i>Lab2/Lab3</i> according to the dimension of the problem.
B		Magnetic flux density vector sum and direction cosines.
FMAG		Electromagnetic force vector sum and direction cosines.
P		Poynting vector sum and direction cosines.

Item	Comp	Description
JS		Source current density vector sum and direction cosines for low-frequency magnetic analyses. Total current density vector sum and direction cosines (sum of conduction and displacement current densities) in low frequency electric analyses.
JT		Total measurable current density vector sum and direction cosines in low-frequency electromagnetic analyses. (Conduction current density vector sum and direction cosines in a low-frequency electric analysis.)
JC		Conduction current density vector sum and direction cosines for elements that support conduction current calculation.

## Menu Paths

**Main Menu>General Postproc>List Results>Vector Data**  
**Utility Menu>List>Results>Vector Data**

## PSCONTROL, *Option, Key*

Turn shared-memory parallel operations on or off during solution.

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Option*

Specify the operations for which you will enable/disable parallel behavior:

#### **ALL**

Turn parallel on/off for all areas.

#### **PREP**

Turn parallel on/off during preprocessing (**/PREP7**).

#### **SOLU**

Turn parallel on/off during solution (**/SOLU**).

#### **FORM**

Turn parallel on/off during element matrix generation.

#### **SOLV**

Turn parallel on/off during equation solver.

#### **RESU**

Turn parallel on/off during element results calculation.

#### **POST**

Turn parallel on/off during postprocessing (**/POST1** and **/POST26**).

#### **STAT**

List parallel operations that are enabled/disabled.

### *Key*

Option control key. Used for all *Option* values, except STAT.

**ON**

Turn parallel operation on.

**OFF**

Turn parallel operation off.

## Command Default

Leave all parallel operations enabled (ON) if multiple processors under *Shared-memory ANSYS* are used.

## Notes

Use this control on shared memory parallel operations, for example, when you encounter minor discrepancies in a nonlinear solution when using different numbers of processors. A parallel operation applied to the element matrix generation can produce a different nonlinear solution with a different number of processors. Although the nonlinear solution converges to the same nonlinear tolerance, the minor discrepancy created may not be desirable for consistency.

Turning parallel behavior on/off for the solution (*Option* = SOLU) supercedes the activation/deactivation of parallel behavior for element matrix generation (FORM), equation solver (SOLV), and element results calculation (RESU).

The SOLV option supports only the sparse direct, PCG, and AMG solvers [**EQSLV**,SPARSE (or PCG, or AMG)]. No other solvers are supported by this option.

This command applies only to shared-memory architecture. It does not apply to the Distributed ANSYS product.

This command does not apply to FLOTRAN analyses.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**PSCR**, *Kywrđ*, *KEY*

**Specifies various PostScript options.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

If *Kywrđ* = CMAP, command format is **PSCR**,CMAP,*INDEX*,*IRED*,*IGRN*,*IBLU*.

**INDEX**

Color map index (0 to 15, 128 to 255).

**IRED**

Red intensity (0 to 100).

**IGRN**

Green intensity (0 to 100).

**IBLU**

Blue intensity (0 to 100).



If *Kyword* = LWID, command format is **PSCR,LWID,KYLWID**.

**KYLWID**

Line width factor (1 to 99, initially 3).

If *Kyword* = COLOR, command format is **PSCR,COLOR,KEY**.

**KEY**

Color key:

**0**

Monochrome.

**1**

16 colors.

**2**

256+ colors.

**3**

Shaded monochrome.

If *Kyword* = TRANX, command format is **PSCR,TRANX,OFFSET**.

**OFFSET**

X translational offset (initially 592).

If *Kyword* = TRANY, command format is **PSCR,TRANY,OFFSET**.

**OFFSET**

Y translational offset (initially 25).

If *Kyword* = ROTATE, command format is **PSCR,ROTATE,ANGLE**.

**ANGLE**

Rotation angle in degrees (initially 90.0).

If *Kyword* = SCALE, command format is **PSCR,SCALE,SCALE**.

**SCALE**

Scale factor (initially 0.238).

If *Kyword* = TIFF, command format is **PSCR,TIFF,KEY**. Used to add a Tagged Image File Format (TIFF) bitmap preview image to the encapsulated PostScript file.

**KEY**

Preview image key:

**0**

Do not include TIFF bitmap preview image.

**1**

Include wireframe TIFF bitmap preview image.

**2**

Include shaded TIFF bitmap preview image.

If *Kyword* = EPSI, command format is **PSCR,EPSI,KEY**. Used to add an Encapsulated PostScript Interchange (EPSI) format bitmap preview image to the encapsulated PostScript file.

**KEY**

Preview image key:

- 0** Do not include EPSI bitmap preview image.
- 1** Include wireframe EPSI bitmap preview image.
- 2** Include shaded EPSI bitmap preview image.

If *Kywrđ* = PAPER, command format is **PSCR,PAPER,SIZE,ORIENT**.

**SIZE**

Standard paper sizes, A, B, C, D, E, A4, A3, A1, A0.

**ORIENT**

Paper orientation, either Landscape or Portrait.

If *Kywrđ* = HIRES, command format is **PSCR,HIRES,KEY**. Used to select the output resolution.

**KEY**

Output resolution key:

- 0** High resolution graphics off. The image is exported as a bitmap.
- 1** High resolution graphics on. The image will be exported in polygon mode (**/TYPE**, , 4 precise hidden) for maximum printer resolution.

## Notes

This command is available in both the ANSYS and DISPLAY programs. It is valid for postscript format files chosen in ANSYS with the **/SHOW,PSCR** command, or in DISPLAY with **/SHOWDISP,POSTSCRIPT**.

An output file is generated for each plot. The ANSYS file is named `JobnameNN.pscr`. In the DISPLAY program, this file is named `PSCRnn`. This file remains open for a subsequent **/NOERASE** plot, and will be incomplete until the program is closed (**/EXIT**), or until the next file is opened by the next **/ERASE** plot request.

Issuing **PSCR,STAT** will list paper size, orientation and resolution modes.

## Menu Paths

**Utility Menu>PlotCtrls>Redirect Plots>To PSCR File**

---

**PSDCOM**, *SIGNIF*, *COMODE*

Specifies the power spectral density mode combination method.

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
[Product Restrictions](#)

***SIGNIF***

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For PSD response (**SPOPT**, PSD), the significance level is defined as the modal covariance matrix term, divided by the maximum modal covariance matrix term. Any term whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of terms used. *SIGNIF* defaults to 0.0001. If *SIGNIF* is specified as 0.0, it is taken as 0.0.

***COMODE***

First *COMODE* number of modes to be actually combined. *COMODE* must always be less than or equal to *NMODE* (input quantity *NMODE* on the **SPOPT** command). *COMODE* defaults to *NMODE*. *COMODE* performs a second level of control for the first sequential *COMODE* number of modes to be combined. It uses the significance level threshold indicated by *SIGNIF* and operates only on the significant modes.

**Notes**

This command is also valid for PREP7. This command is valid only for **SPOPT**, PSD.

**Product Restrictions**

**PSDCOM** is not allowed in ANSYS Professional.

**Menu Paths**

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Mode Combine

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Mode Combine

**PSDFRQ**, *TBLNO1*, *TBLNO2*, *FREQ1*, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*

Defines the frequency points for the input spectrum tables PSDVAL vs. PSDFRQ for PSD analysis.

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***TBLNO1***

Input table number. When used with the **COVAL** or the **QDVAL** command, *TBLNO1* represents the row number of this table. Up to 20 tables may be defined.

***TBLNO2***

Input table number. *TBLNO2* is used only for the **COVAL** or the **QDVAL** commands and represents the column number of this table.

***FREQ1*, *FREQ2*, *FREQ3*, ..., *FREQ7***

Frequency points (cycles/time) for spectrum vs. frequency tables. *FREQ1* should be greater than zero, and values must be in ascending order. Log-log interpolation will be used between frequency points.

## Notes

The spectrum values may be input with the **PSDVAL**, **COVAL**, or **QDVAL** commands. A separate **PSDFRQ** command must be used for each table and cross table defined. Frequencies must be in ascending order.

Repeat **PSDFRQ** command for additional frequency points. Values are added after the last nonzero frequency. If all fields after **PSDFRQ** are blank, all input vs. frequency tables are erased. If *TBLNO1* is nonblank, all corresponding **PSDVAL** tables are erased. If both *TBLNO1* and *TBLNO2* are nonblank, all corresponding **COVAL** and **QDVAL** tables are erased.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Cospectral**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Erase Co & Quad**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Erase Tables**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>PSD vs Freq**  
**Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Cospectral**  
**Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Erase Co & Quad**  
**Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral**  
**Main Menu>Solution>Load Step Opts>Spectrum>PSD>Erase Tables**  
**Main Menu>Solution>Load Step Opts>Spectrum>PSD>PSD vs Freq**

## PSDGRAPH, *TBLNO1*, *TBLNO2*

### Displays input PSD curves

**SOLUTION: Spectrum Options**

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

#### *TBLNO1*

PSD table number to display.

#### *TBLNO2*

Second PSD table number to display. *TBLNO2* is used only in conjunction with the **COVAL** or the **QDVAL** commands.

## Notes

The input PSD tables are displayed in log-log format as dotted lines. The best-fit curves, used to perform the closed-form integration, are displayed as solid lines. If there is a significant discrepancy between the two, then you should add one or more intermediate points to the table to obtain a better fit.

If *TBLNO2* is zero, blank, or equal to *TBLNO1*, then the autospectra (**PSDVAL**) are displayed for *TBLNO1*. If *TBLNO2* is also specified, then the autospectra for *TBLNO1* and *TBLNO2* are displayed, along with the corresponding cospectra (**COVAL**) and quadspectra (**QDVAL**), if they are defined.

This command is valid in any processor.

## Menu Paths

Main Menu>Preprocessor>Loads>Spectrum>Graph PSD Tab

Main Menu>Solution>Spectrum>Graph PSD Tab

**PSDRES**, *Lab*, *RelKey*

Controls solution output written to the results file from a PSD analysis.

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *Lab*

Label identifying the solution output:

#### **DISP**

Displacement solution (default). One-sigma displacements, stresses, forces, etc. Written as load step 3 on `File.RST`.

#### **VELO**

Velocity solution. One-sigma velocities, "stress velocities," "force velocities," etc. Written as load step 4 of `File.RST`.

#### **ACEL**

Acceleration solution. One-sigma accelerations, "stress accelerations," "force accelerations," etc. Written as load step 5 on `File.RST`.

### *RelKey*

Key defining relative or absolute calculations:

#### **REL**

Calculations are relative to the base excitation (default).

#### **ABS**

Calculations are absolute.

#### **OFF**

No calculation of solution output identified by *Lab*.

## Command Default

Relative displacement solution, no velocity or acceleration solution for 1  $\sigma$  results.

## Notes

Controls the amount and form of solution output written to the results file from a PSD analysis. One-sigma values of the relative or absolute displacement solution, relative or absolute velocity solution, relative or absolute acceleration solution, or any combination may be included on the results file.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Calc Controls  
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Calc Controls

---

**PSDSPL**, *TBLNO*, *RMIN*, *RMAX*

Defines a partially correlated excitation in a PSD analysis.

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***TBLNO***

Input PSD table number defined with **PSDVAL** command.

***RMIN***

Minimum distance between excitation points which are partially correlated. Excited nodes closer than *RMIN* will be fully correlated.

***RMAX***

Maximum distance between excitation points which are partially correlated. Excited nodes farther apart than *RMAX* will be uncorrelated.

## Notes

Defines a partially correlated excitation in terms of a sphere of influence relating excitation point geometry (in a PSD analysis). If the distance between any two excitation points is less than *RMIN*, then the excitation is fully correlated. If the distance is greater than *RMAX*, then the excitation is uncorrelated. If the distance lies between *RMIN* and *RMAX*, then the excitation is partially correlated with the degree of correlation dependent on the separation distance between the points. This command is not available for a pressure PSD analysis.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Spatial Correlat  
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Spatial Correlat

---

**PSDUNIT**, *TBLNO*, *Type*, *GVALUE*

Defines the type of input PSD.

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***TBLNO***

Input table number.

***Type***

Label identifying the type of spectrum:

**DISP**

Displacement spectrum (in terms of displacement<sup>2</sup>/Hz ).

**VELO**

Velocity spectrum (in terms of velocity<sup>2</sup>/Hz ).

**ACEL**

Acceleration spectrum (in terms of acceleration<sup>2</sup>/Hz ).

**ACCG**

Acceleration spectrum (in terms of g<sup>2</sup>/Hz ).

**FORC**

Force spectrum (in terms of force<sup>2</sup>/Hz ).

**PRES**

Pressure spectrum (in terms of pressure<sup>2</sup>/Hz ).

**GVALUE**

Value of acceleration due to gravity in any arbitrary units for Type=ACCG. Default is 386.4 in/sec<sup>2</sup>.

**Command Default**

Acceleration (ACEL) spectrum (acceleration<sup>2</sup>/Hz).

**Notes**

Defines the type of PSD defined by the **PSDVAL**, **COVAL**, and **QDVAL** commands.

Force (FORC) and pressure (PRES) type spectra can be used only as a nodal excitation.

**GVALUE** is valid only when type ACCG is specified. A zero or negative value cannot be used. A parameter substitution can also be performed.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Settings**

**Main Menu>Solution>Load Step Opts>Spectrum>PSD>Settings**

**PSDVAL**, *TBLNO*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

**Defines PSD values.**

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**TBLNO**

Input table number being defined.

*SV1, SV2, SV3, . . . , SV7*

Spectral values corresponding to the frequency points [**PSDFRQ**]. Values are interpreted as defined with the **PSDUNIT** command.

## Notes

Defines PSD values to be associated with the previously defined frequency points.

Repeat **PSDVAL** command for additional values, up to the number of frequency points [**PSDFRQ**]. Values are added after the last nonzero value.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>PSD vs Freq**

**Main Menu>Solution>Load Step Opts>Spectrum>PSD>PSD vs Freq**

---

**PSDWAV**, *TBLNO, VX, VY, VZ*

Defines a wave propagation excitation in a PSD analysis.

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

*TBLNO*

Input PSD table number defined with **PSDVAL** command.

*VX*

Global Cartesian X-velocity of traveling wave.

*VY*

Global Cartesian Y-velocity of traveling wave.

*VZ*

Global Cartesian Z-velocity of traveling wave.

## Notes

Defines a traveling wave in a PSD analysis. This command is not available for a pressure PSD analysis.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Traveling Wave**

**Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Traveling Wave**

---



**/PSEARCH**, *Pname*

Specifies a directory to be searched for "unknown command" macro files.

[APDL: Macro Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Pname***

Path name (64 characters maximum, and must include the final delimiter) of the middle directory to be searched. Defaults to the user home directory. If *Pname* = OFF, search only the ANSYS and current working directories. If *Pname* = STAT, list the current middle directory and show the **ANSYS\_MACROLIB** setting.

**Command Default**

The middle directory searched is the user home directory.

**Notes**

Specifies the pathname of a directory for file searches when reading "unknown command" macro files. The search for the files is typically from the ANSYS directory, then from the user home directory, and then from the current working directory. This command allows the middle directory searched to be other than the user home directory.

This command is valid only at the Begin Level.

**Menu Paths**

Utility Menu>Macro>Macro Search Path

---

**PSEL**, *Type*, *Pname1*, *Pname2*, *Pname3*, *Pname4*, *Pname5*, *Pname6*, *Pname7*, *Pname8*, *Pname9*, *Pname10*

Selects a path or paths.

[POST1: Path Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Type***

Label identifying the type of select:

**S**

Select a new path.

**R**

Reselect a path from the current set of paths.

**A**

Additionally select a path and extend the current set of paths.

**U**

Unselect a path from the current set of paths.

**ALL**

Restore the full set of paths.

**NONE**

Unselect the full set of paths.

**INV**

Invert the current set of paths (selected becomes unselected and vice versa).

*Pname1, Pname2, Pname3, . . . , Pname10*

Name of existing path(s).

**Notes**

Selects a path or multiple paths, up to ten. Data are flagged as selected and unselected; no data are actually deleted from the database. There is no default for this command; you must specify a type and pathname.

**Menu Paths**

**Main Menu>General Postproc>Path Operations>Archive Path>Store>Paths in file**

**Main Menu>Preprocessor>Path Operations>Store>Paths in file**

**/PSF**, *Item, Comp, KEY, KSHELL, Color*

**Shows surface load symbols on model displays.**

[GRAPHICS:Labeling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Item, Comp***

Labels identifying the surface load to be shown; see [Table 250: /PSF - Valid Item and Component Labels](#) (p. 1367).

***KEY***

Key to turn surface load symbols on or off:

**0**

Off (default).

**1**

On, shown as face outlines. Line surface loads (**SFL**) on solid model plots are shown as arrows.

**2**

On, shown as arrows.

**3**

On, shown as color filled surfaces. Line and area surface loads (**SFL** and **SFA**) on solid model plots are shown as arrows.

***KSHELL***

Visibility key for shell elements.

**0**

Off (default), surface load symbols are displayed only on visible load faces.

**1**

On, surface load symbols are displayed even if load face is not visible.

***Color***

Visibility key for contour legend.

**ON**

The symbols (arrows or face outlines) will show up in color with the legend showing the corresponding color labels (default).

**OFF**

The contour legend will not be displayed. The symbols (arrows or face outlines) will show up in grey. The size of the arrows will be proportional to the applied load.

**Command Default**

No surface load symbols are displayed.

**Notes**

The **/PSF** command determines whether and how to show surface loads on subsequent model displays.

If surface loads are applied to solid model entities, only solid model plots show the load symbols; node and element plots do not show them unless the loads are transferred (**SFTRAN** or **SBCTRAN**). Similarly, solid model plots do not show the load symbols if surface loads are applied to nodes and elements. For node and element plots of shell element models, the surface load symbols are shown only if the load face is visible from the current viewing direction.

The effects of the **/PSF** command are not cumulative (that is, the command does not modify an existing setting from a previously issued **/PSF** command). Only the setting specified via the most recent **/PSF** command applies.

If you issue a **postprocessing (POST1)** plot command that produces result contours (such as **PLNSOL**), the **/PSF** command has no effect. This behavior prevents conflicting contours in the graphics window.

When using the Radiosity method (*Item* = RDSF and *Comp* = ENCL) with *Key* = 2, the radiation arrows point outward from any element face. When using **SURF154** with **KEYOPT(2) = 1**, set the *Item* to PRES and leave the Component Label blank.

**/PSF,STAT** displays current **/PSF** settings, and **/PSF,DEFA** resets them back to default.

Other useful commands are **/PNUM,SVAL,1** to show the values of the surface loads, **/VSCALE** to change arrow lengths, and **/PBC** and **/PBF** to activate other load symbols.

For beam elements, only the colors representing shear (GREEN) and normal (RED) pressures are displayed for the arrows. The color of these arrows does not correspond to the magnitudes in the contour legend. The length of these arrows does, however, correlate to the relative magnitude of the pressures.

For the **SURF159**, **SOLID272**, and **SOLID273** elements, the **/PSF** command is not available when displaying elements with shapes determined from the real constants or section definition (**/ESHAPE**).

This command is valid in any processor.

**Table 250 /PSF - Valid Item and Component Labels**

Item	Comp	Description
Valid item and component labels are:		
PRES	NORM	Applied structural pressure normal to the face (real component).

Item	Comp	Description
"	TANX	Applied structural pressure in the tangential-x direction (real component).
"	TANY	Applied structural pressure in the tangential-y direction (real component).
"	INRM	Applied structural pressure normal to the face (imaginary component).
"	ITNX	Applied structural pressure in the tangential-x direction (imaginary component).
"	ITNY	Applied structural pressure in the tangential-y direction (imaginary component).
CONV	HCOEF	Applied convection (film coefficient).
"	TBULK	Applied convection (bulk temperature).
RAD	EMIS	Applied radiation (emissivity).
"	TAMB	Applied radiation (ambient temperature).
RDSF	EMSS	Radiation emissivity.
"	ENCL	Enclosure number.
FSIN		Fluid-solid interface number.
VFRC		Volume fraction (VOF method).
HFLUX		Applied heat flux.
FSI		Acoustic fluid-structure interface flag.
IMPD		Applied acoustic impedance.
SHLD	COND	Applied conductivity.
"	MUR	Applied relative permeability.
MXWF		Maxwell force flag.
INF		Exterior surface flag.
CHRG		Applied electric surface charge density.
MCI		Magnetic circuit interface.

## Menu Paths

Utility Menu>PlotCtrls>Symbols

**PSMAT**, *Fname*, *Ext*, *Matrix*, *Color*

**Writes an assembled global matrix to a postscript format that graphically displays nonzero matrix values.**

[AUX2: Binary Files](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

This file name is used for the postscript file name. It defaults to the current Jobname if left blank.

### **Ext**

Filename extension (8 character maximum).

Defaults to `.PS` if left blank.

### **Matrix**

Specify which matrix to write to the output postscript file:

#### **STIFF**

Write stiffness matrix to output postscript file. Valid for all types of analyses that write a `.FULL` file.

#### **MASS**

Write mass matrix to output postscript file. Valid for buckling, substructure, and modal analyses. If the `.FULL` file was generated in a buckling analysis, then this label will write the stress stiffening matrix to the output postscript file.

#### **DAMP**

Write damping matrix to output postscript file. Only valid for damped modal analyses.

### **Color**

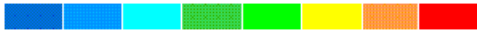
Specifies whether to display the grid in black and white or in color:

#### **BLACK**

Each nonzero coefficient is symbolized by a black square (default).

#### **COLOR**

Each nonzero coefficient is symbolized by a colored square. The color depends on the module of the coefficient; the range is from blue for the smallest values to red for the largest values. The color map is:



## **Notes**

This command is used to copy a matrix from the assembled global matrix file (`.FULL` file) as specified on the **FILE** command and write it in a postscript format to a new file named `Fname.Ext` (defaults to `Jobname.PS`). The matrix is symbolized by a grid in which the black or colored squares represent the nonzero coefficients of the matrix. The `.FULL` file must be available for this command to work properly.

If the matrix is large, it may be difficult to display the postscript file. In this case, use `Color = BLACK` to reduce the postscript file size.

The assembled global matrix file is created during solution depending on the analysis type, equation solver, and other solution options. By default, the assembled global matrix file is never deleted at the end of solution. For most analysis types, the Sparse direct solver, the ICCG solver, and the AMG solver (when available) will write a `.FULL` file. With the exception of the reduced (**MODOPT,REDUC**) and Variational Technology (**MODOPT,VT**) options, all mode extraction methods used for buckling and modal analyses will write a properly formatted `.FULL` file to be used with the **PSMAT** command.

When copying the stiffness matrix for transient and harmonic analyses, be aware that the element mass matrix values (and possibly element damping matrix values) are incorporated into the globally assembled stiffness matrix. Thus, the globally assembled stiffness matrix represents more than the stiffness of the model for these analysis types. Please refer to the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for more details.

The **PSMAT** command is not able to display a lumped mass matrix from a `.FULL` file generated by a harmonic analysis.

When copying a `.FULL` file, the rows and columns corresponding to specified constraints (e.g., **D** commands) are eliminated from the system of equations and therefore are not written to the `.PS` file. In addition, rows and columns corresponding to eliminated (slave) degrees of freedom from coupling and/or constraint equations (e.g., **CE**, **CP** commands) are eliminated from the system of equations and are not written to the `.PS` file. The DOFs that are eliminated from any coupling and/or constraint equations are determined internally by the solution code and may not match what you specified via the **CE/CP** (or similar) commands.

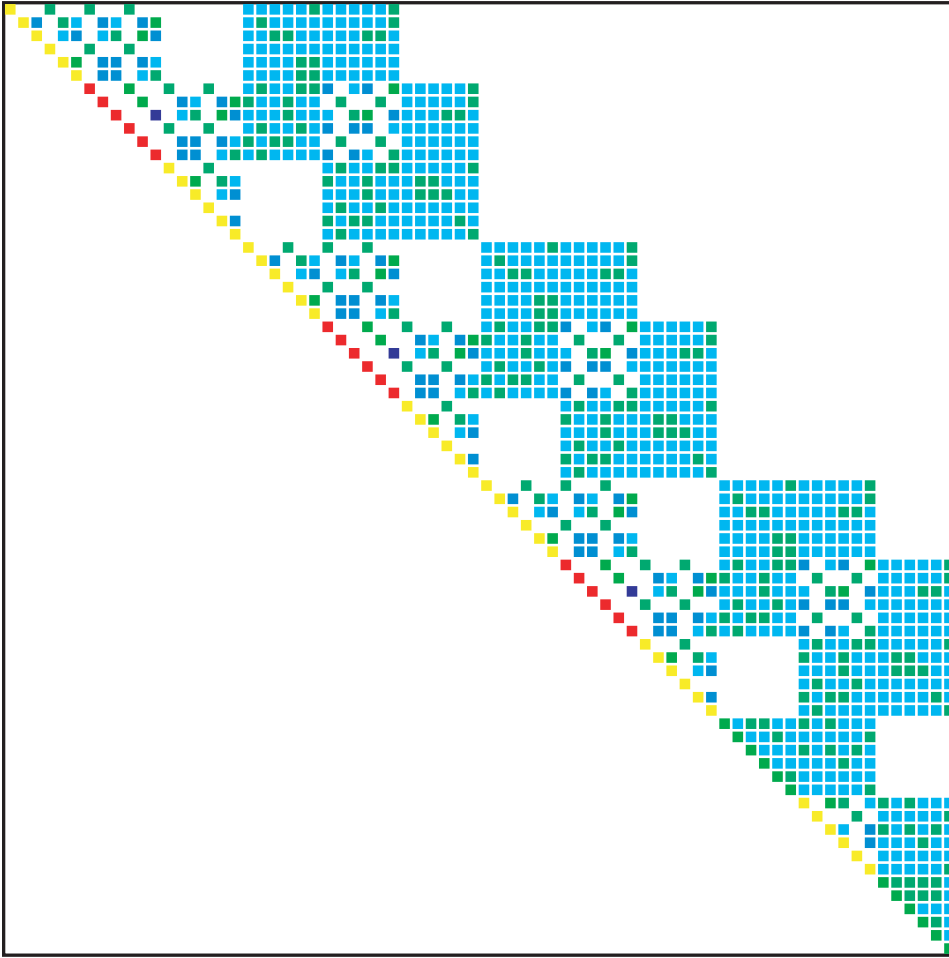
When copying a `.FULL` file, only the upper triangular part of the matrix will be written to the `.PS` file if the matrix is symmetric; the full matrix is written if the matrix is unsymmetric. Only matrix coefficients that are greater than zero will be written.

The **WRFULL** command, in conjunction with the **SOLVE** command, can be used to generate the assembled global matrix file and eliminate the equation solution process and results output process.

The following command sequence shows typical usage of this command.

```
/BATCH,LIST
/AUX2    ! Enter AUX2 processor
FILE,job1,full  ! FULL file containing stiffness matrix is job1.full
PSMAT,job1KColor,ps,STIFF,COLOR ! Create file job1KColor.ps in color
!
!                               postscript format for stiffness matrix
PSMAT,job1MBlack,,STIFF,BLACK ! Create file job1MBalck.ps in black/white
!
!                               postscript format for stiffness matrix
FINISH
```

Below is an example of an export of the stiffness matrix to a postscript format using the **COLOR** option.



K Matrix

## Menu Paths

This command cannot be accessed from a menu.

---

**PSMESH**, *SECID*, *Name*, *P0*, *Egroup*, *NUM*, *KCN*, *KDIR*, *VALUE*, *NDPLANE*, *PSTOL*, *PSTYPE*, *ECOMP*, *NCOMP*

### Create and mesh a pretension section

[PREP7: Meshing](#)

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

#### *SECID*

Unique section number. This number must not already be assigned to a section.

#### *Name*

Unique eight character descriptive name, if desired.

#### *P0*

Pretension node number. The node will be defined if it doesn't exist and the number defaults to the highest node number plus one.

**Egroup, NUM**

Element group on which **PSMESH** will operate. If *Egroup* = P, graphical picking is enabled and *NUM* is ignored (valid only in the GUI).

**L (or LINE)**

**PSMESH** operates on all elements in the line specified by *NUM*. New pretension nodes are associated with *NUM* or entities below it. Any subsequent **LCLEAR** operation of *NUM* deletes the pretension elements and nodes created by **PSMESH**.

**A (or AREA)**

**PSMESH** operates on all elements in the area specified by *NUM*. New pretension nodes are associated with *NUM* or entities below it. Any subsequent **ACLEAR** of *NUM* deletes the pretension elements and nodes created by **PSMESH**.

**V (or VOLU)**

**PSMESH** operates on all elements in the volume specified by *NUM*. New pretension nodes are associated with *NUM* or entities below it. Any subsequent **VCLEAR** of *NUM* deletes the pretension elements and nodes created by **PSMESH**.

**P**

**PSMESH** operates on elements selected through the subsequent picking operations, and *NUM* is ignored

**ALL**

The command operates on all selected elements, and *NUM* is ignored.

**KCN**

Coordinate system number for the separation surface and normal direction.

**KDIR**

Direction (x, y, or z) normal to separation surface in the *KCN* coordinate system.

If *KCN* is cartesian, the pretension section normal will be parallel to the *KDIR* axis regardless of the position of the pretension node.

If *KCN* is non-cartesian, the pretension section normal will be aligned with the *KDR* direction of system *KCN* at the position of the pretension node.

**VALUE**

Point along the *KDIR* axis at which to locate the separation surface. Ignored if *NDPLANE* is supplied.

**NDPLANE**

Existing node that **PSMESH** will use to locate the separation surface. If *NDPLANE* is supplied, the location of the separation surface is defined by the *KDIR* coordinate of *NDPLANE*.

**PSTOL**

Optional tolerance below *VALUE*. Allows nodes occurring precisely at or slightly below the separation to be identified properly as above the plane. Has the effect of shifting the plane down by *PSTOL*. The following expression represents the default value:

$$\frac{\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}}{1000}$$

where  $\Delta X$ ,  $\Delta Y$ , and  $\Delta Z$  are the dimensions of the model based on nodal locations (that is,  $\Delta X = X_{max} - X_{min}$ ).



**PSTYPE**

If specified, this value is the type number for pretension elements. (If not specified, ANSYS defines this value.) If already defined, it must be of type **PRETS179**.

**ECOMP**

If specified, the name of a component to be composed of new pretension elements and existing elements modified by the **PSMESH** command.

**NCOMP**

Name of a component to be composed of nodes on new pretension elements.

## Notes

The **PSMESH** command creates a pretension section normal to the pretension load direction by cutting the mesh along existing element boundaries at the point defined by *VALUE* or *NDPLANE* and inserting **PRETS179** elements. The **PSMESH** command verifies that *PSTYPE* is **PRETS179**; if it is not, the command finds the lowest available *ITYPE* that is **PRETS179**, or if necessary will create a new one.

When it is necessary to define the pretension node, ANSYS uses node *NDPLANE*. If the *NDPLANE* value is not specified, ANSYS defines the pretension node at:

- The centroid of geometric entity *NUM*, if *Egroup* = LINE, AREA or VOLU
- The centroid location of all selected elements, if *Egroup* = ALL or if graphical picking is used.

If the elements to which the pretension load is to be applied have already been meshed in two groups, **PSMESH** cannot be used to insert the pretension elements. The **EINTF** command must be used to insert the **PRETS179** elements between the two meshed groups.

The **PSMESH** operation copies any nodal temperatures you have defined on the split surface of the original mesh from the original nodes to the newly created coincident duplicate nodes. However, displacements, forces, and other boundary conditions are not copied.

By mathematical definition, the pretension surface must always be a flat plane. In a non-Cartesian coordinate system, the **PSMESH** command creates that plane at the indicated position, oriented with respect to the specified direction of the active system (in the same manner that the **NROTAT** command orients a nodal system with respect to a curved system). For example, assuming a  $X = 1$  and  $Y = 45$  in a cylindrical coordinate system with  $Z$  as the axis of rotation ( $KCN = 1$ ), a pretension surface normal to  $X$  tilts 45 degrees away from the global  $X$  axis.

The **PSMESH** command is valid for structural analyses only.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Elements in Area**

**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Elements in Line**

**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Elements in Volu**

**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Picked Elements**

**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Selected Element**

**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Area**

**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Line**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Volu**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Picked Elements**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Selected Element**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Area**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Line**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Volu**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Picked Elements**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Selected Element**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Elements in Area**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Elements in Line**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Elements in Volu**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Picked Elements**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Selected Element**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Area**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Line**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Volu**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Picked Elements**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Selected Element**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Area**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Line**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Volu**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Picked Elements**  
**Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Selected Element**

---

**PSOLVE**, *Lab, Rkey*

**Directs the program to perform a partial solution.**

**SOLUTION:** Analysis Options

MP ME ST PR PRN <> <> <> <> <> <> <> PP <> EME MFS

**Lab**

Valid labels defining the solution step. All characters are required:

**EIGDAMP**

Calculates the eigenvalues and eigenvectors using the damped eigensolver. Requires Jobname . FULL from **MODOPT**,UNSYM or **MODOPT**,DAMP options. Produces Jobname . MODE.

**EIGQRDA**

Calculates eigenvalues and eigenvectors using the QR damped eigensolver. Requires Jobname . EMAT from **MODOPT**,QRDAMP option. Produces Jobname . MODE.

**EIGEXP**

Expands the eigenvector solution. Requires Jobname . ESAV and Jobname . MODE. Produces Jobname . RST.

**EIGLANB**

Calculates the eigenvalues and eigenvectors using Block Lanczos. Requires Jobname . EMAT from **MODOPT**,LANB option. Produces Jobname . MODE.

**EIGLANPCG**

Calculates the eigenvalues and eigenvectors using PCG Lanczos. Requires Jobname . EMAT from **MODOPT**,LANPCG option. Produces Jobname . MODE.

**EIGSNODE**

Calculates the eigenvalues and eigenvectors using the Supernode method. Requires Jobname . EMAT from **MODOPT**,SNODE option. Produces Jobname . MODE. (See the **MODOPT** command for more information on the SNODE modal solver.)

**EIGREDUC**

Calculates the eigenvalues and eigenvectors using Householder. Requires Jobname . REDM. Produces Jobname . MODE.

**EIGUNSYM**

Calculates the eigenvalues and eigenvectors using the unsymmetric eigensolver. Requires Jobname . EMAT from **MODOPT**,UNSYM or **MODOPT**,DAMP options. Produces Jobname . MODE.

**ELFORM**

Creates the element matrices. Produces Jobname . EMAT and Jobname . ESAV.

**Note**

If you want to include prestress effects (**PSTRES**,ON) from a previous prestress analysis, the ELFORM option requires the Jobname . EMAT and Jobname . ESAV files generated by that analysis.

**ELPREP**

Modifies element matrices for solution and calculates inertia relief terms (**IRLF**). Requires Jobname . EMAT. Produces Jobname . ER0T.

**REDWRITE**

Writes the reduced matrix to a file. Requires `Jobname.REDM`. Produces `Jobname.SUB`.

**TRIANG**

Triangularizes the matrices completely. This option is required prior to performing a partial solution using the `EIGREDUC` option.

**Rkey**

Key for initial contact results:

**CNDI**

Write initial contact configuration to the results file. This option is only valid for `Lab = ELFORM`.

**Notes**

Directs the program to perform a partial solution (that is, one step of an analysis sequence). Predefined analysis types (**ANTYPE**) perform a defined subset of these solution steps in a predefined sequence. You can use the partial-solution procedure to repeat a certain step of an analysis or to restart an analysis.

Not all steps are valid for all analysis types. The order of the steps may vary depending on the result you desire. See the *Basic Analysis Guide* for a description of how to perform partial and predefined solutions.

An example of a prestressed modal analysis is given below. The `Jobname.EMAT` and `Jobname.ESAV` files from a prior static analysis must be available.

The example is provided for illustration purposes only, as the [linear perturbation analysis method](#) is recommended in place of this partial-solution method.

```
! Prestressed modal analysis
!
/SOLU
ANTYPE,MODAL      ! Modal analysis
UPCOORD,1.0,ON   ! Display mode shapes relative to deformed geometry
                  !   in the postprocessor.
PSTRES,ON        ! Prestress effects ON
MODOPT,LANB      ! Select eigensolver
PSOLVE,EIGLANB   ! Calculate the eigenvalues and eigenvectors.
                  ! EIGxxx label must be consistent with mode extraction method on MODOPT command.
FINISH
/SOLU             !Additional solution step for expansion.
EXPASS,ON
MXPAND,...       ! Specify the number of modes to expand, if desired.
PSOLVE,EIGEXP    ! Expand the eigenvector solution. (Required if you
                  !   want to review mode shapes in the postprocessor.)
FINISH
```

In a cyclic symmetry analysis, **PSOLVE,EIGLANB** or **PSOLVE,EIGLANPCG** performs the modal analysis at multiple load steps, one for each nodal-diameter specified via the **CYCOPT** command. In addition, the eigenvector solution is expanded at each nodal-diameter solution, eliminating the need for a separate expansion pass (**PSOLVE,EIGEXP**).

If issuing **PSOLVE,ELFORM** and **PSOLVE,ELPREP** using the Jacobi Conjugate Gradient solver, do so only *after* issuing **PSOLVE,CGSOL**; otherwise, unpredictable results may occur.

Although documented to work, using the **PSOLVE** commands with an iterative solver is not likely to decrease solution-processing time.

If Jobname .EMAT is required, run the prior analysis with **EMATWRITE,YES** to ensure that a Jobname .EMAT is generated.

In a prestressed modal analysis, issue a **PSOLVE,TRIANG** command immediately before a **PSOLVE,EIGREDUC** command to ensure that ANSYS creates a proper .FULL file. The **PSOLVE ,EIGUNSYM**; **PSOLVE,EIGLANB**; **PSOLVE,EIGDAMP**; **PSOLVE,EIGQRDA**; **PSOLVE,EIGLANPCG**; and **PSOLVE,EIGSNODE** commands do not require a preceding **PSOLVE,TRIANG** command and should *not* be preceded by a **PSOLVE,TRIANG** command.

**Distributed ANSYS Restriction** Only the EIGLANB, and EIGLANPCG options on this command are supported in Distributed ANSYS.

## Menu Paths

Main Menu>Solution>Solve>Partial Solu

**/PSPEC**, *PCOLOR*, *KFILL*, *KBORDR*

Creates annotation polygon attributes (GUI).

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *PCOLOR*

Polygon color ( $0 \leq PCOLOR \leq 15$ ):

- 0** Black.
- 1** Red-Magenta.
- 2** Magenta.
- 3** Blue-Magenta.
- 4** Blue.
- 5** Cyan-Blue.
- 6** Cyan.
- 7** Green-Cyan.
- 8** Green.
- 9** Yellow-Green.
- 10** Yellow.

**11**

Orange.

**12**

Red.

**13**

Dark Gray.

**14**

Light Gray.

**15**

White.

***KFILL***

Polygon fill key:

**0**

Hollow polygon.

**1**

Filled polygon.

***KBORDR***

Polygon border key:

**0**

No border.

**1**

Border.

## Notes

Creates annotation polygon attributes to control certain characteristics of the polygons created via the **/POLYGON**, **/PMORE**, **/PCIRCLE** and **/PWEDGE** commands. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Annotation>Create Annotation**

---

## **/PSTATUS**, *WN*

**Displays the global or window display specifications.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***WN***

Window number for status (defaults to global specifications).

## Notes

Displays the current global or window display specifications. Global display specifications are common to all windows (e.g. **/SHOW**, etc.). Window display specifications are specific to one window (e.g. **/VIEW**, **/TYPE**, etc.).

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Status>Graphics>General  
 Utility Menu>List>Status>Graphics>Window 1  
 Utility Menu>List>Status>Graphics>Window 2  
 Utility Menu>List>Status>Graphics>Window 3  
 Utility Menu>List>Status>Graphics>Window 4  
 Utility Menu>List>Status>Graphics>Window 5

## PSTRES, *Key*

Specifies whether prestress effects are calculated or included.

[SOLUTION: Nonlinear Options](#)

MP ME ST PR PRN DS DSS <> EM EH <> PP <> EME MFS

### Key

Prestress key:

#### OFF

Do not calculate (or include) prestress effects (default).

#### ON

Calculate (or include) prestress effects.

## Notes

The **PSTRES** command specifies whether or not prestress effects are to be calculated or included. The command should be issued *after* the **ANTYPE** command.

Prestress effects are calculated in a static or transient analysis for inclusion in a buckling, modal, harmonic (Method = FULL or REDUC), transient (Method = REDUC), or substructure generation analysis. If used in the solution processor (**/SOLU**), this command is valid only within the first load step.

If the prestress effects are to be calculated in a nonlinear static or transient analysis (for a [prestressed modal analysis of a large-deflection solution](#)), you can issue a **SSTIF,ON** command (rather than a **PSTRES,ON** command) in the static analysis.

If you apply thermal body forces during a static analysis to calculate prestress effects, do not delete the forces during any subsequent full harmonic response analyses. If you delete the thermal body forces, the thermal prestress effects will not be included in the harmonic response analysis. Temperature loads used to define the thermal prestress will also be used in the full harmonic response analysis as sinusoidally time-varying temperature loads.

A prestress effect applied with non-follower loads resists rigid body rotation of the model. For example, an unsupported beam with axial tensile forces applied to both ends will have two nonzero rotational rigid body modes.

If tabular loading (**\*DIM,,TABLE**) was used in the prestress static analysis step, the corresponding value of **TIME** will be used for tabular evaluations in the modal analysis.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic**  
**Main Menu>Solution>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Basic**

**/PSYMB**, *Label, KEY*

**Shows various symbols on displays.**

[GRAPHICS: Labeling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Label*

Show symbols as selected from the following labels:

#### **CS**

Local coordinate systems.

#### **NDIR**

Nodal coordinate systems (on rotated nodes only).

#### **ESYS**

Element coordinate systems (element displays only).

#### **LDIR**

Line directions (line displays only).

#### **LDIV**

Controls the display of element divisions on lines.

#### **ADIR**

Area direction symbol (for keypoint, line, area and volume plots).

#### **LAYR**

Layer orientations (relative to the projected element x-axis) or fiber orientations in smeared reinforcing elements. Used only within an element display. Use *KEY* to specify the layer number.

#### **PCON**

Convergence criterion symbols on nodes (for a p-method analysis).

#### **ECON**

Element mesh symbols on keypoints and lines.

#### **DOT**

Larger symbols displayed for node and keypoint locations. When *Label* = DOT, *KEY* = 1 by default.



**XNOD**

Extra node of surface or circuit elements.

**FBCS**

Force boundary condition scaling. Subsequent *KEY* value determines whether or not to scale the applied and derived forces/moments to the same maximum value.

**DEFA**

Resets the symbol keys so that ANSYS displays none of the symbols controlled by the **/PSYMB** command. The value of the *KEY* field is ignored.

**STAT**

Prints the status of the settings of the symbol keys controlled by the **/PSYMB** command. The *KEY* field is ignored.

**KEY**

Symbol key:

**-1**

Effective only if *Label* = LAYR and solid shape element display (**/ESHAPE**) is active. Orientation of all layers appears with the solid shape element display.

**0**

No symbol (default). If *Label* = LDIV, then *KEY* = 0 indicates that the displayed element divisions will correspond to the existing mesh (the word MESHED or EXISTING can also be substituted). Also, for *Label* = LDIV, if you execute any meshing command (such as **AMESH** or **VMESH**), *KEY* is set to 0 (MESHED) automatically. If *Label* = FBCS, then *KEY* = 0 indicates that boundary condition scaling will not be common. The applied and derived forces/moments will be scaled to their respective maximum values.

**1**

Include symbol. If *Label* = LDIV, then *KEY* = 1 indicates that the displayed line divisions will correspond to the value assigned by **LESIZE** (the word LESIZE can also be substituted). Also, for *Label* = LDIV, if you execute the **LESIZE** command, *KEY* is set to 1 (LESIZE) automatically. If *Label* = FBCS, then *KEY* = 1 indicates that boundary condition scaling will be common. The applied and derived forces/moments will be scaled to the same maximum value.

**N**

If *Label* = LAYR, then *N* is equal to the layer number. If *Label* = DOT, then *N* can be equal to 0,1,.....15, indicating the dot size. If *Label* = LDIV, then *KEY* = -1, indicates that no element divisions will be displayed (the word OFF can also be substituted).

**Notes**

Includes various symbols on the display. Triads are right-handed with x displayed as the longest leg. Where color is displayed, x is white, y is green, and z is blue. For beams, x is always along the length of the element. For lines, an arrow represents the direction of a line, from the beginning keypoint to the end keypoint. See **/PLOPTS** command for additional display options. Use **/PSTATUS** or **/PSYMB,STAT** to display settings. Use **/PSYMB,DEFA** to reset all specifications back to their defaults. The command **/PSYMB,ECON,1** causes the symbol "M" to be displayed on keypoints and lines associated with meshed entities. When you issue the command **/PSYMB,DOT,1**, a larger symbol is displayed for each node and keypoint location.

PowerGraphics (**/GRAPHICS,POWER**) does not support **/PSYMB,ESYS** and **/PSYMB,LAYR**.

If  $KEY = N$  and PowerGraphics is off, the centroid of the surface elements is connected to the extra node using a gray line. However, if PowerGraphics is on, the color of the line connecting the centroid to the extra node is the same as that for the elements themselves (as determined by **/PNUM**).

When  $Label = LAYR$ , the layer systems can be visualized with all current-technology layered elements and the smeared reinforcing element **REINF265**. To use **/PSYMB,LAYR** with **REINF265**, first set the vector-mode graphics option (**/DEVICE,VECTOR,1**).

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Areas**  
**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Lines**  
**Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>with Area**  
**Main Menu>Preprocessor>Trefftz Domain>TZ Symmetry**  
**Utility Menu>PlotCtrls>Symbols**

---

**PTR**, *LOC*, *BASE*

Dumps the record of a binary file.

[AUX2: Binary Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*LOC*, *BASE*

Dump the file record starting at pointer *LOC*. *BASE* is the base pointer, and would be used if *LOC* is a relative pointer.

## Notes

Dumps the record of the file named on the AUX2 **FILEAUX2** command according the format specified on the **FORM** command.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**PTXY**, *X1*, *Y1*, *X2*, *Y2*, *X3*, *Y3*, *X4*, *Y4*

Defines coordinate pairs for use in polygons and prisms.

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*X1*, *Y1*, *X2*, *Y2*, *X3*, *Y3*, *X4*, *Y4*

X and Y coordinate pairs on the working plane.

## Notes

Defines coordinate pairs for use in polygons and prisms [**POLY**, **RPRISM**]. The coordinates must be in the Cartesian coordinate system. The coordinate pairs must be input in a continuous order. **PTXY** may be repeated (up to 100 pairs) until the required pairs have been defined. The pairs will be saved until either the **POLY** or **PRISM** command is entered. Use **PTXY,STAT** to list the saved coordinate pairs. Use **PTXY,DELE** to delete all the saved coordinate pairs. See the **RPOLY**, **RPRISM**, and **RPR4** commands for other ways to create polygons and prisms.

## Menu Paths

**This command cannot be accessed from a menu.**

**PVECT**, *Oper*, *LabXR*, *LabYR*, *LabZR*

**Interpolates a set of items onto a path.**

POST1: Path Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Oper*

Valid operations for geometry operations along a path are:

#### **NORM**

Defines a unit normal vector at each interpolation point in the direction of the cross product of the tangent to the path and the active Z axis. Resulting vector components are in the active coordinate system (which must be Cartesian).

#### **TANG**

Defines a unit vector tangent to the path at each interpolation point. Vector components are in the active coordinate system (which must be Cartesian).

#### **RADI**

Defines the position vector of each interpolation point of the path from the center of the active coordinate system (which must be Cartesian).

### *LabXR*

Label (8 characters maximum) assigned to X-component of the resulting vector.

### *LabYR*

Label (8 characters maximum) assigned to Y-component of the resulting vector.

### *LabZR*

Label (8 characters maximum) assigned to Z-component of the resulting vector.

## Notes

Defines and interpolates a set of labeled path items along predefined path [**PATH**] and performs various geometric operations on these path items. A path item must be defined before it can be used with other path operations. Additional path items may be defined with the **PDEF**, **PCALC**, **PDOT**, and **PCROSS** commands. Path items may be listed or displayed with the **PLPATH**, **PLPAGM** and **PRPATH** commands. Path geometry items (XG, YG, ZG, S) are automatically interpolated (in the active CSYS) if not done so previously with the **PDEF** command.

## Menu Paths

Main Menu>General Postproc>Path Operations>Unit Vector

---

**/PWEDGE**, *XCENTR*, *YCENTR*, *XLRAD*, *ANGLE1*, *ANGLE2*

**Creates an annotation wedge (GUI).**

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***XCENTR***

Wedge X center location (-1.0 < X < 2.0).

***YCENTR***

Wedge Y center location (-1.0 < Y < 1.0).

***XLRAD***

Wedge radius length.

***ANGLE1***

Starting angle of wedge.

***ANGLE2***

Ending angle of wedge. The wedge is drawn counterclockwise from the starting angle, *ANGLE1*, to the ending angle, *ANGLE2*.

## Notes

Creates an annotation wedge to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (*Jobname.LOG*) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All wedges are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** and the **/PSPEC** command to set the attributes of the wedge.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

---

---

## Q Commands

**QDVAL**, *TBLNO1*, *TBLNO2*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

**Defines PSD quadspectral values.**

**SOLUTION: Spectrum Options**

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**TBLNO1**

First input PSD table number associated with this spectrum.

**TBLNO2**

Second input PSD table number associated with this spectrum.

**SV1**, **SV2**, **SV3**, . . . , **SV7**

PSD quadspectral values corresponding to the frequency points [**PSDFRQ**].

### Notes

Defines PSD quadspectral values to be associated with the previously defined frequency points. Repeat **QDVAL** command with the same table number for additional points. Unlike autospectra [**PSDVAL**], the quadspectra can be positive or negative. The quadspectral curve segment where there is a sign change is interpolated linearly (the rest of the curve segments use log-log interpolation). For better accuracy, choose as small a curve segment as possible wherever a sign change occurs.

Two table numbers are required since values are off-diagonal terms. This command is valid for **SPOPT**, PSD only.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral**

**Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral**

---

## QFACT

Calculates the quality factor for high-frequency electromagnetic resonators.

[POST1: Magnetics Calculations](#)

MP <> <> <> <> <> <> <> <> <> <> PP <> <> <>

## Notes

The **QFACT** command macro calculates the quality factor for high-frequency electromagnetic resonators. It returns the quality factor as a scalar parameter, **QFACT**. To compute the quality factor, the macro uses the stored energy, surface losses, and dielectric losses.

See magnetic macros for further details.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Cavity>Q-factor

---

## QRDOPT, ReuseKey

Specifies additional QRDAMP modal analysis option.

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN DS DSS <> <> EH <> PP <> EME MFS  
[Product Restrictions](#)

### ReuseKey

Reuse key for method=*QRDAMP* specified in **MODOPT** command.

#### ON

Reuse the Block Lanczos eigensolution from the previous load steps or from the previous solution.

#### OFF

Do not reuse (calculates Block Lanczos eigensolution at current load step). This is the default.

## Notes

If the *filename.modesym* file exists in the working directory and *ReuseKey* = ON, *filename.modesym* will be reused. If *filename.modesym* does not exist in the working directory, the Block Lanczos eigensolution will be calculated.

*ReuseKey* can be set to *ON* or *OFF* in each load step.

## Menu Paths

This command cannot be accessed from a menu.

---

**QSOPT**, *Opt*

**Specifies quasi static radiation options.**

[SOLUTION: Radiosity](#)

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

***Opt***

Quasi static option:

**OFF**

Do not run transient radiation problem to steady-state (default).

**ON**

Run transient radiation problem to steady-state.

**Notes**

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Quasi-Static**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Quasi-Static**

**QUAD**, *NODE1, NINTR, NODE2, NFILL, NSTRT, NINC, PKFAC*

**Generates a quadratic line of nodes from three nodes.**

[PREP7: Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NODE1***

Begin fill-in from this node location. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

***NINTR***

Intermediate or guiding node. Quadratic curve will pass through this location. *NINTR* may have any node number and any location. If the quadratic line also generates a node with number *NINTR*, the generated location overrides the previous *NINTR* location.

***NODE2***

End quadratic fill-in at this node location.

***NFILL***

Fill-in *NFILL* nodes between *NODE1* and *NODE2* (defaults to  $|NODE2-NODE1|-1$ ). *NFILL* must be positive.

***NSTRT***

Node number assigned to first filled-in node (defaults to  $NODE1 + NINC$ ).

***NINC***

Add this increment to each of the remaining filled-in node numbers (may be positive or negative). Defaults to  $(NODE2-NODE1)/(NFILL + 1)$ , i.e., linear interpolation.

### **PKFAC**

Peak location factor. If  $PKFAC=0.5$ , the peak of the quadratic shape occurs at the *NINTR* location. If  $0.0 < PKFAC < 0.5$ , the peak occurs to the *NODE2* side of the *NINTR* location. If  $0.5 < PKFAC < 1.0$ , the peak occurs to the *NODE1* side of the *NINTR* location. Defaults to 0.5.

## **Notes**

Generates a quadratic line of nodes (in the active coordinate system) from three nodes. The three nodes determine the plane of the curve and may have been defined in any coordinate system. Any number of nodes may be filled-in and any node number sequence may be assigned.

The quadratic line feature uses three nodes (*NODE1,NINTR,NODE2*) to determine the plane of the curve. The curve passes through the three points, beginning from *NODE1*, through the intermediate (or guiding) point *NINTR*, and toward *NODE2*.

Generated nodes are also quadratically spaced. If the guiding node number is within the set being generated, it will be relocated according to the quadratic spacing.

The peak location factor is used to determine how the quadratic fits through the three points. Various nodal progressions can be obtained by different combinations of *PKFAC* and the guiding node location. If the guiding node is at mid-length between *NODE1* and *NODE2*,  $0.293 \leq PKFAC < 0.707$  will ensure that all generated nodes fall within the *NODE1,NODE2* bounds. In the limit, as *PKFAC* approaches 0.0, the peak approaches the line through *NODE1* and *NINTR* at an infinite distance from *NODE1*. The **QUAD** command generates quadratic lines of nodes, which in turn may be used as a base line for generating irregular surfaces of nodes (by repeating [**\*REPEAT**], generating [**NGEN, NSCALE**], etc.). Irregular surfaces may also be generated with the meshing commands.

## **Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Nodes>Quadratic Fill**

---

## **/QUIT**

**Exits a processor.**

SESSION: Processor Entry

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## **Notes**

This is an alternative to the **FINISH** command. If any cleanup or file writing is normally done by the **FINISH** command, it is bypassed if the **/QUIT** command is used instead. A new processor may be entered after this command. See the **/EXIT** command to terminate the run.

This command is valid in any processor. This command is not valid at the Begin level.

## **Menu Paths**

**This command cannot be accessed from a menu.**

---



**QUOT**, *IR*, *IA*, *IB*, --, *Name*, --, --, *FACTA*, *FACTB*

**Divides two variables.**

POST26: Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

***IA*, *IB***

Reference numbers of the two variables to be operated on.

--

Unused field.

***Name***

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --

Unused fields.

***FACTA*, *FACTB***

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

## Notes

Divides two variables according to the operation:

$$IR = (FACTA \times IA) / (FACTB \times IB)$$

## Menu Paths

**Main Menu>TimeHist Postpro>Math Operations>Divide**



---

## R Commands

**R**, *NSET*, *R1*, *R2*, *R3*, *R4*, *R5*, *R6*

**Defines the element real constants.**

[PREP7: Real Constants](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NSET***

Set identification number (arbitrary). If same as a previous set number, set is redefined. Set number relates to that defined with the element **[REAL]**. Note that the GUI automatically sets this value.

***R1***, ***R2***, ***R3***, . . . , ***R6***

Real constant values (interpreted as area, moment of inertia, thickness, etc., as required for the particular element type using this set), or table names for tabular input of boundary conditions. Use **RMORE** command if more than six real constants per set are to be input.

## Notes

Defines the element real constants. The real constants required for an element are shown in the Input Summary of each element description in the [Element Reference](#). Constants must be input in the same order as shown in that table. If more than the required number of element real constants are specified in a set, only those required are used. If fewer than the required number are specified, zero values are assumed for the unspecified constants.

If using table inputs ([SURF151](#), [SURF152](#), [FLUID116](#), [CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#), and [CONTA175](#) only), enclose the table name in % signs (e.g., %*tabname*%).

When copying real constants to new sets, ANSYS recommends that you use the command input. If you do use the GUI, restrict the real constant copy to only the first six real constants (real constants seven and greater will be incorrect for both the master and copy set).

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>Add/Edit/Delete**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Mass**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Rotary**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Trans**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Transducer>ElecMech**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Edit Real Cnst**

**Main Menu>Preprocessor>Real Constants>Add/Edit/Delete**

**Main Menu>Solution>Load Step Opts>Other>Real Constants>Add/Edit/Delete**

**RACE**, *XC, YC, RAD, TCUR, DY, DZ, --, --, Cname*

Defines a "racetrack" current source.

PREP7: Special Purpose

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

**XC**

Location of the mid-thickness of the vertical leg along the working plane X-axis.

**YC**

Location of the mid-thickness of the horizontal leg along the working plane Y-axis.

**RAD**

Radius of curvature of the mid-thickness of the curves in the racetrack source. Defaults to .501 \* DY

**TCUR**

Total current, amp-turns (MKS), flowing in the source.

**DY**

In-plane thickness of the racetrack source.

**DZ**

Out-of-plane thickness (depth) of the racetrack source.

**--, --**

Unused fields

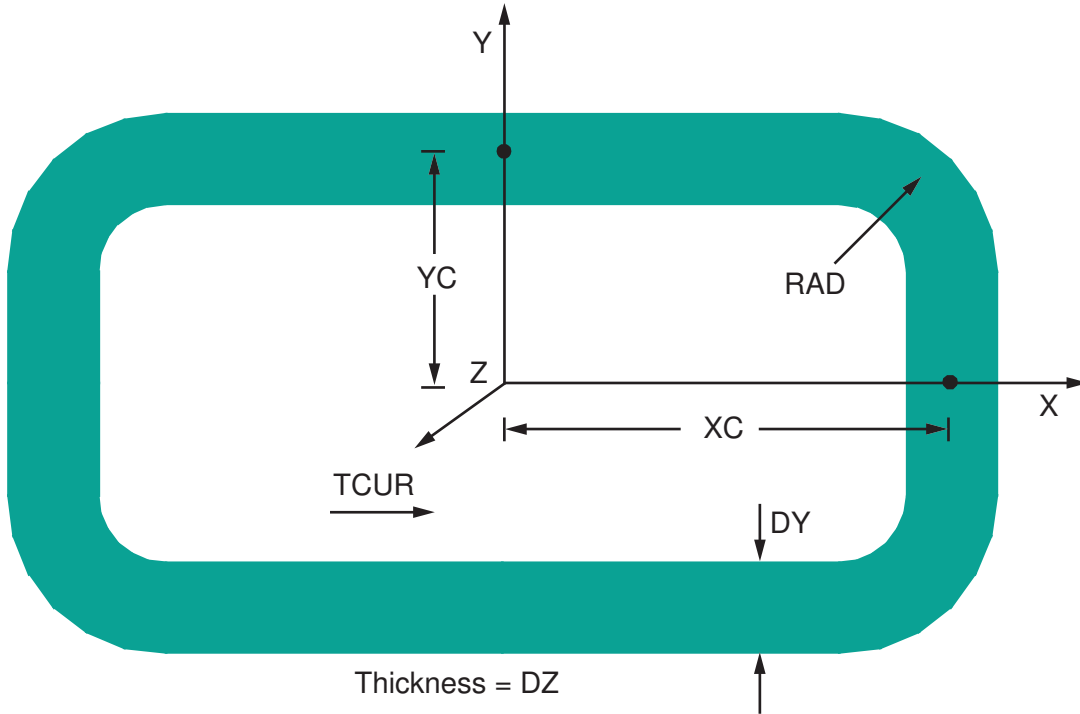
**Cname**

An alphanumeric name assigned as a component name to the group of [SOURC36](#) elements created by the command macro. Cname must be enclosed in single quotes in the **RACE** command line. Cname may be up to 32 characters, beginning with a letter and containing only letters, numbers, and underscores. Component names beginning with an underscore (e.g., `_LOOP`) are reserved for use by ANSYS and should be avoided. If blank, no component name is assigned.

## Notes

**RACE** invokes an ANSYS macro which defines a "racetrack" current source in the working plane coordinate system. The current source is generated from bar and arc source primitives using the [SOURC36](#) element (which is assigned the next available element type number). The macro is valid for use in 3-D magnetic field analysis using a scalar potential formulation. Current flows in a counterclockwise direction with respect to the working plane.

The diagram below shows you a racetrack current source.



## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>Racetrack Coil**

**Main Menu>Preprocessor>Modeling>Create>Racetrack Coil**

**Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>Racetrack Coil**

**RADOPT**, --, *FLUXTOL*, *SOLVER*, *MAXITER*, *TOLER*, *OVERRLEX*

**Specifies Gauss-Seidel Radiosity Solver options.**

[SOLUTION: Radiosity](#)  
[AUX12: Radiosity Solver](#)

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

--

Unused field.

### **FLUXTOL**

Convergence tolerance for radiation flux. Defaults to 0.0001 for ANSYS Thermal radiation analysis and FLOTTRAN surface radiation analysis.

### **SOLVER**

Choice of solver for radiosity calculation:

**0**

Iterative solver (default).

**1**

Direct solver.

### **MAXITER**

Maximum number of iterations for iterative solver (*SOLVER* = 0). Defaults to 1000.

**TOLER**

Convergence tolerance for the iterative solver (*SOLVER* = 0). Defaults to 0.1.

**OVERRLEX**

Over relaxation factor applied to the iterative solver (*SOLVER* = 0). Defaults to 0.1.

**Notes**

The radiation heat flux is linearized, resulting in a robust convergence.

**Menu Paths**

**Main Menu>Preprocessor>Radiation Opts>Solution Opt**  
**Main Menu>Radiation Opt>Radiosity Meth>Solution Opt**  
**Main Menu>Solution>Radiation Opts>Solution Opt**

---

**RAPPND**, *LSTEP*, *TIME*

**Appends results data from the database to the results file.**

[POST1: Load Case Calculations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**LSTEP**

Load step number to be assigned to the results data set. If it is the same as an existing load step number on the results file, the appended load step will be inaccessible. Defaults to 1.

**TIME**

Time value to be assigned to the results data set. Defaults to 0.0. A time value greater than the last load step should be used.

**Notes**

This command is typically used to append the results from a load case combination to the results file. See the **LCWRITE** command to create a separate load case file. Only summable and constant data are written to the results file by default; non-summable data are not written unless requested (**LCSUM** command). RAPPND should not be used to append results from a harmonic analysis.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>General Postproc>Write Results**

---

**RATE**, *Option*

**Specifies whether the effect of creep strain rate will be used in the solution of a load step.**

SOLUTION: Analysis Options

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS  
Product Restrictions

**Option**

Activates implicit creep analysis.

**0 or OFF**

No implicit creep analysis. This option is the default.

**1 or ON**

Perform implicit creep analysis.

**Command Default**

Issuing this command with no arguments deactivates implicit creep analysis.

**Notes**

Set *Option* = 1 (or ON) to perform an implicit creep analysis (**TB,CREEP** with *TBOPT*  $\geq$  1). For viscoplasticity/creep analysis, *Option* specifies whether or not to include the creep calculation in the solution of a load step. If *Option* = 1 (or ON), ANSYS performs the creep calculation. Set an appropriate time for solving the load step via a **TIME,TIME** command.

**Product Restrictions**

This command works only when modeling *implicit* creep with either von Mises or Hill potentials.

When modeling implicit creep with von Mises potential, you can use the following elements: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLID272, SOLID273, SOLID285, SOLSH190, BEAM188, BEAM189, SHELL208, SHELL209, REINF264, SHELL281, and ELBOW290.

When modeling anisotropic creep (**TB,CREEP** with **TB,HILL**), you can also use the following elements: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear**

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Strn Rate Effect**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear**

**Main Menu>Solution>Load Step Opts>Nonlinear>Strn Rate Effect**

**/RATIO**, *WN*, *RATOX*, *RATOY*

**Distorts the object geometry.**

[GRAPHICS: Scaling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***WN***

Window number (or ALL) to which command applies (defaults to 1).

***RATOX***

Distort object in the window X direction by this factor (defaults to 1.0).

***RATOY***

Distort object in the window Y direction by this factor (defaults to 1.0).

## Command Default

No distortion.

## Notes

Distorts the object geometry in a particular direction. An example of this command's use would be to allow long narrow sections to be distorted to a more square area for better display visualization.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

---

**RBE3**, *Master*, *DOF*, *Slaves*, *Wtfact*

**Distributes the force/moment applied at the master node to a set of slave nodes, taking into account the geometry of the slave nodes as well as weighting factors.**

[PREP7: Constraint Equations](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

***Master***

Node at which the force/moment to be distributed will be applied. This node must be associated with an element for the master node to be included in the DOF solution.

***DOF***

Refers to the master node degrees of freedom to be used in constraint equations. Valid labels are: UX, UY, UZ, ROTX, ROTY, ROTZ, UXYZ, RXYZ, ALL

***Slaves***

The name of an array parameter that contains a list of slave nodes. Must specify the starting index number. ALL can be used for currently selected set of nodes. The slave nodes may not be colinear, that is, not be all located on the same straight line (see **Notes** below).



***Wtfact***

The name of an array parameter that contains a list of weighting factors corresponding to each slave node above. Must have the starting index number. If not specified, the weighting factor for each slave node defaults to 1.

**Notes**

The force is distributed to the slave nodes proportional to the weighting factors. The moment is distributed as forces to the slaves; these forces are proportional to the distance from the center of gravity of the slave nodes times the weighting factors. Only the translational degrees of freedom of the slave nodes are used for constructing the constraint equations. Constraint equations are converted to distributed forces/moments on the slave nodes during solution.

**RBE3** creates constraint equations such that the motion of the master is the average of the slaves. For the rotations, a least-squares approach is used to define the "average rotation" at the master from the translations of the slaves. If the slave nodes are colinear, then one of the master rotations that is parallel to the colinear direction can not be determined in terms of the translations of the slave nodes. Therefore, the associated moment component on the master node in that direction can not be transmitted. When this case occurs, a warning message is issued and the constraint equations created by **RBE3** are ignored.

Applying this command to a large number of slave nodes may result in constraint equations with a large number of coefficients. This may significantly increase the peak memory required during the process of element assembly. If real memory or virtual memory is not available, consider reducing the number of slave nodes.

As an alternative to the **RBE3** command, you can apply a similar type of constraint using contact elements and the internal multipoint constraint (MPC) algorithm. See [Surface-based Constraints](#) for more information.

This command is also valid in SOLUTION.

**Menu Paths**

**Main Menu>Preprocessor>Coupling / Ceqn>Dist F/M at Mstr**

**RCON**

**Specifies "Real constants" as the subsequent status topic.**

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Preprocessor>Real Constants

**RDEC**, Option *REDUC*, --, *Nplace*

Defines the decimation parameters.

[SOLUTION: Radiosity](#)

MP ME ST PR <> <> <> <> <> <> <> PP <> EME MFS

### Option

Command options:

#### DEFINE

Defines the decimation parameters (default).

#### STAT

Shows the status/listing. Other command options are ignored.

### *REDUC*

Approximate reduction in the number of surface elements. Valid range is from 0.0 (no decimation, the default) to 1.0. This number is a factor applied to the initial number of element radiosity surfaces.

--

Unused field.

### *Nplace*

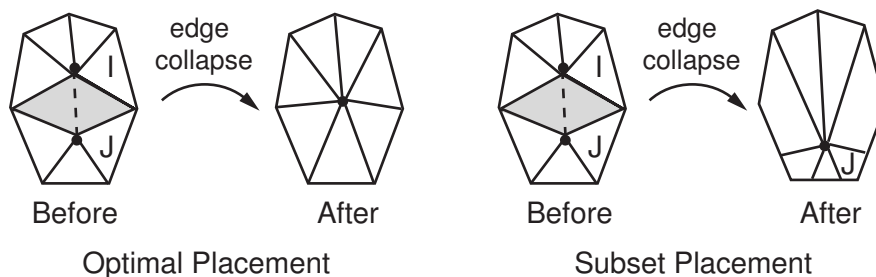
Node placement algorithm

#### OPTI

Optimal placement. An edge is collapsed by moving both nodes (I and J in the figure below) to a new location.

#### SUBS

Subset placement. An edge is collapsed by moving one node to another one. In the figure below, node I is moved to node J.



## Notes

Decimation is the process of simplifying a fine surface mesh into a coarse one. This process is accomplished by a sequence of edge collapses.

The maximum degree of decimation (1.0) is unreachable. The real degree of decimation is always less than 1.0 because the decimated mesh must always consist of at least one element.

## Menu Paths

Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Decimation Options>Define Specifications

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Decimation Options>Define Specifications

**RDELE**, *NSET1*, *NSET2*, *NINC*,--, *LCHK*

Deletes real constant sets.

[PREP7: Real Constants](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NSET1*, *NSET2*, *NINC*

Delete real constant sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL, ignore *NSET2* and *NINC* and all real constant sets are deleted.

--

Unused field.

*LCHK*

Specifies the level of element-associativity checking:

**NOCHECK**

No element-associativity check occurs. This option is the default.

**WARN**

When a section, material, or real constant is associated with an element, ANSYS issues a message warning that the necessary entity has been deleted.

**CHECK**

The command terminates, and no section, material, or real constant is deleted if it is associated with an element.

## Notes

Deletes real constant sets defined with the **R** command.

This command is also valid in SOLUTION.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>Add/Edit/Delete

Main Menu>Preprocessor>Real Constants

Main Menu>Preprocessor>Real Constants>Add/Edit/Delete

Main Menu>Solution>Load Step Opts>Other>Real Constants>Add/Edit/Delete

**REAL**, *NSET*

**Sets the element real constant set attribute pointer.**

[PREP7: Meshing](#)  
[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NSET***

Assign this real constant set number to subsequently defined elements (defaults to 1).

**Command Default**

*NSET* = 1.

**Notes**

Identifies the real constant set number to be assigned to subsequently defined elements. This number refers to the real constant set number (*NSET*) defined with the real constant sets [**R**]. Real constant set numbers may be displayed [**/PNUM**]. If the element type requires no real constants, this entry is ignored. Elements of different type should not refer to the same real constant set.

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes**

**REALVAR**, *IR, IA, --, --, Name, --, --, FACTA*

**Forms a variable using only the real part of a complex variable.**

[POST26: Operations](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

***IA***

Reference number of the variable to be operated on.

--, --

Unused fields.

***Name***

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --

Unused fields.

***FACTA***

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

## Notes

Forms a variable using only the real part of a variable. Used only with harmonic analyses (**ANTYPE**,HARMIC).

Complex variables are stored in two-column arrays with the real component stored in the first column and the imaginary component stored in the second column. This command extracts the value stored in the first column (i.e., real component). However with harmonic analyses, all variables are stored in two-column arrays as complex variables. If the variable is not complex, then the same value is stored in both columns. This command will extract the variable in the first column of the array, even if this variable is not the real component of a complex variable.

## Menu Paths

**Main Menu>TimeHist Postpro>Math Operations>Real Part**

### RECTNG, X1, X2, Y1, Y2

**Creates a rectangular area anywhere on the working plane.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**X1, X2**

Working plane X coordinates of the rectangle.

**Y1, Y2**

Working plane Y coordinates of the rectangle.

## Notes

The area will be defined with four keypoints and four lines. See the **BLC4** and **BLC5** commands for alternate ways to create rectangles.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By Dimensions**

### REMESH, Action, Filename, Ext, --, Opt1, Opt2

**Specifies the starting and ending remeshing points, and other options, for manual rezoning.**

[SOLUTION: Rezoning](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**Action**

#### START

Starts the remeshing operation.

#### FINISH

Ends the remeshing operation.

**READ**

Reads in a generic (.cdb format) new mesh file generated by a third-party application. This remeshing option applies to both 2-D and 3-D rezoning.

**SPLIT**

Splits selected elements of an existing 2-D mesh such that a quadrilateral element is split into four quadrilaterals, a degenerate quadrilateral is split into three quadrilaterals, and a quadratic triangular element is split into four quadratic triangles. This remeshing option applies to 2-D rezoning only.

**Filename**

Name of a .cdb generic mesh file. The default value is *jobname*. Valid only when *Action* = READ.

**Ext**

File name extension. The only valid (and the default) extension is CDB. Valid only when *Action* = READ.

--

Unused field.

**Opt1**

Specifies options for the new mesh when using a generic imported mesh file or the mesh-splitting remeshing method. Valid only when *Action* = READ or *Action* = SPLIT.

**REGE**

Regenerates all node and element numbers on the new mesh using an offset of the highest existing node and element numbers. This is the default behavior when *Action* = READ; otherwise, this value is ignored.

**KEEP**

Keeps the similarly numbered nodes and elements in the new and the old meshes unchanged. Valid only when *Action* = READ.

**TRAN**

Generates transition elements to ensure nodal compatibility between split and unsplit parts of the mesh. Valid only when *Action* = SPLIT.

**Opt2**

Specifies transition options for the mesh when elements are split. These options are valid only when *Action* = SPLIT.

**QUAD**

Minimizes the number of degenerate elements in the transition mesh and tries to maximize the number of quadrilateral transition elements across several layers of elements from the split regions. This is the default behavior.

**DEGE**

Creates transition zones between the split and unsplit parts of the mesh using mostly degenerate elements with a single element layer.

## Notes

The **REMESSH** command is valid only during the manual rezoning (**REZONE,MANUAL**) process.

In manual rezoning, a **REMESSH,START** command temporarily exits the **/SOLU** solution processor and enters a special mode of the **/PREP7** preprocessor, after which a limited number of preprocessing commands are available for mesh control, but no solution commands are valid.

A **REMESSH,FINISH** command exits the remeshing process and reenters the solution processor, at which point no preprocessing commands are available. If the new mesh exists, the command creates contact elements

if needed, and transfers all boundary conditions (BCs) and loads from the original mesh to the new mesh. You can issue any list or plot command to verify the created contact elements, transferred BCs, and loads. A **REMESH,FINISH** command is valid only after a previously issued **REMESH,START** command, and is the only way to safely end the remeshing operation (and exit the special mode of the **/PREP7** preprocessor).

A **REMESH,READ** command is valid only when you want to perform a rezoning operation using a generic new mesh generated by a third-party application (rather than a new mesh generated internally by the ANSYS program). The command is valid between **REMESH,START** and **REMESH,FINISH** commands. In this case, the only valid file extension is `.cdb` (*Ext* = CDB). When *Option* = KEEP, ANSYS assumes that the common node and element numbers between the old and the new mesh are topologically similar (that is, these commonly numbered areas have the same element connectivity and nodal coordinates).

A **REMESH,SPLIT** command is valid only when you wish to perform a rezoning operation by splitting the existing mesh. The command is valid between **REMESH,START** and **REMESH,FINISH** commands.

You can use **REMESH,READ** and **REMESH,SPLIT** commands for horizontal multiple rezoning provided that the meshes used in **REMESH,READ** do not intersect. (ANSYS recommends against issuing an **AREMESH** command after issuing either of these commands.)

For more detailed about the remeshing options available to you during a rezoning operation, see "[Manual Rezoning](#)" in the *Advanced Analysis Techniques Guide*.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Manual Rezoning>Cancel**  
**Main Menu>Solution>Manual Rezoning>Finish**  
**Main Menu>Solution>Manual Rezoning>Read Mesh**  
**Main Menu>Solution>Manual Rezoning>Select and Split Elems**  
**Main Menu>Solution>Manual Rezoning>Start**

---

**/RENAME**, *Fname1*, *Ext1*, --, *Fname2*, *Ext2*, --

**Renames a file.**

SESSION: Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Fname1*

Name of file to be renamed and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

File name defaults to the current Jobname.

### *Ext1*

Filename extension (8 character maximum).

--

Unused field.

### ***Fname2***

Name of file to be renamed and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

*Fname2* defaults to *Fname1*.

### ***Ext2***

Filename extension (8 character maximum).

*Ext2* defaults to *Ext1*.

--

Unused field.

## Notes

Renames a file. Ex: **/RENAME,A,,,B** renames file A to B in the same directory. **/RENAME,A,DAT,,,INP** renames file A.DAT to A.INP. On all systems, this command will overwrite any existing file named B. See the [Operations Guide](#) for details. Only ANSYS binary files should be renamed. Use **/SYS** and system renaming commands for other files.

Renaming across system partitions may be internally done by a copy and delete operation on some systems.

This command is valid only at the Begin Level.

## Menu Paths

**Utility Menu>File>File Operations>Rename**

---

## REORDER

Specifies "Model reordering" as the subsequent status topic.

PREP7: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Main Menu>Preprocessor>Numbering Ctrl>Element Reorder>List Wave Lists**  
**Utility Menu>List>Status>Preprocessor>Reorder Module**

---



**\*REPEAT**, *NTOT*, *VINC1*, *VINC2*, *VINC3*, *VINC4*, *VINC5*, *VINC6*, *VINC7*, *VINC8*, *VINC9*, *VINC10*, *VINC11*

Repeats the previous command.

APDL: Process Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *NTOT*

Number of times the preceding command is executed (including the initial execution). Must be 2 or greater. *NTOT* of 2 causes one repeat (for a total of 2 executions).

### *VINC1*, *VINC2*, *VINC3*, . . . , *VINC11*

Value increments applied to first through eleventh data fields of the preceding command.

## Notes

**\*REPEAT** must immediately follow the command that is to be repeated. The numeric arguments of the initial command may be incremented in the generated commands. The numeric increment values may be integer or real, positive or negative, zero or blank. Alphanumeric arguments cannot be incremented. For large values of *NTOT*, consider printout suppression (**/NOPR** command) first.

Most commands beginning with slash (/), star (\*), as well as "unknown command" macros, cannot be repeated. For these commands, or if more than one command is to be repeated, include them within a do-loop. File switching commands (those reading additional commands) cannot be repeated. If a **\*REPEAT** command immediately follows another **\*REPEAT** command, the repeat action only applies to the last non-**\*REPEAT** command. Also, **\*REPEAT** should not be used in interactive mode immediately after a) a command (or its log file equivalent) that uses picking, or b) a command that requires a response from the user.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## **/REPLOTT**, *Label*

Automatically reissues the last display command for convenience.

GRAPHICS: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Label*

Controls the type of replot.

#### **RESIZE**

Issued internally when a graphics window resize occurs (Default).

#### **FAST**

Only applicable for 3-D devices that allow a fast redisplay for changes in the view characteristics only.

## Notes

Reissues the last display command (**NPLOT**, **EPLLOT**, **KPLOT**, **PLNSOL**, **PLVAR**, etc.), along with its parameters, for convenience. The current display specifications are used.

When the last display command is invalid in a particular processor, the use of the **/REPLOT** command is also invalid in that processor. However, if you attempt a **/REPLOT** and the last display command is invalid in the current processor, ANSYS produces an element display [**EPLLOT**] instead, *as long as the last display command was **PLNSOL**, **PLESOL**, or **PLDISP***. ANSYS performs this substitution of **/REPLOT** with **EPLLOT** for your convenience.

For example, the **PLNSOL** command, which is used to display solution results as continuous contours, is a valid command in the general postprocessor [**/POST1**]. If you issue **PLNSOL** followed by **/REPLOT** while in the general postprocessor, the **/REPLOT** command effectively reissues your earlier **PLNSOL** command, along with its parameters. But if you then exit the general postprocessor, enter the preprocessor [**/PREP7**], and issue the **/REPLOT** command again, ANSYS internally issues **EPLLOT** instead. This occurs because **PLNSOL** is not a valid command in the preprocessor.

When you click on one of the buttons on the **Pan, Zoom, Rotate** dialog box to manipulate the view of a model, the **/REPLOT** command is issued internally. Thus, the substitution of **/REPLOT** with **EPLLOT** as described above may occur not only for the **PLNSOL**, **PLESOL**, and **PLDISP** results display commands, but also for operations that you perform with the **Pan, Zoom, Rotate** dialog box.

**/REPLOT** will not show boundary conditions if they are only applied to a solid model and the last display command (for example, **EPLLOT**) displays the finite element model. To show boundary conditions, the following options are available:

- Issue **/REPLOT** after you issue the **SBCTRAN** command to transfer solid model boundary conditions to the finite element model.
- Issue **/REPLOT** after you issue a solid model display command (for example, **VPLLOT**).

This command is valid in any processor (except as noted above).

## Menu Paths

**Main Menu>General Postproc>Path Operations>Define Path>On Working Plane**

**Main Menu>General Postproc>Path Operations>Delete Path>All Paths**

**Main Menu>General Postproc>Path Operations>Delete Path>By Name**

**Main Menu>General Postproc>Path Operations>Plot Paths**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon**

**Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane**

**Main Menu>Preprocessor>Path Operations>Delete Path>All Paths**

**Main Menu>Preprocessor>Path Operations>Delete Path>By Name**

**Main Menu>Preprocessor>Path Operations>Plot Paths**

**Utility Menu>Plot>Replot**

**Utility Menu>PlotCtrls>Style>Symmetry Expansion>Expansion by values**

**RESCONTROL**, *Action*, *Ldstep*, *Frequency*, *MAXFILES***Controls file writing for multiframe restarts.**

SOLUTION: Misc Loads

MP ME ST PR PRN DS DSS FL EM EH &lt;&gt; PP &lt;&gt; EME MFS

**Action**

Specify the command action.

**DEFINE**

Issuing the command will specify how frequently the *.Xnnn* restart files are written for a load step (default).

**FILE\_SUMMARY**

Issuing the command will print the substep and load step information for all *.Xnnn* files for the current job name in the current directory. If this option is specified, all other arguments are ignored.

**STATUS**

Issuing the command will list the current status in the tables of restart control specified earlier by **RESCONTROL**.

**NORESTART**

Issuing the command will not allow restart files to be created for Distributed ANSYS runs only. Therefore, the remote processors will not have files such as *.ESAV*, *.OSAV*, *.RST*, *.x000*, etc. in the directory at the end of the run. The host processor will not have files such as *.ESAV*, *.OSAV*, *.x000*, *.RDB*, *.LDHI*, etc. at the end of the run. Distributed ANSYS will remove all the above scratch files at the end of the solution phase (**FINISH** or **/EXIT**). This option is useful for file clean-up on remote or host processors. If this field is specified, the remaining fields (*Ldstep*, *Frequency*, and *MAXFILES*) are not used.

If this option is used in shared-memory parallel ANSYS, most of the restart files in the working directory are removed. It has the same effect as issuing **RESCONTROL,,NONE**.

**LINEAR**

Issuing the command will specify the same actions as *Action* = DEFINE. However, this option is intended for linear static applications. For linear static analysis, the restart capability is normally not needed. It is only needed when a subsequent linear perturbation analysis is desired. By default, none of the restart files are written for a linear static analysis.

**Ldstep**

Specify how the *.Xnnn* files are written.

**ALL**

The *.Xnnn* files are written at the same frequency for all load steps.

**LAST**

Write the *.Xnnn* files for the last load step only (default for nonlinear static and full transient analyses). If this option is specified, results are written for the last substep and the remaining arguments are ignored.

**N**

Write the *.Xnnn* files at the frequency indicated only for load step *N*. Other load steps will be written at the default frequency or at a frequency defined by a previous **RESCONTROL,DEFINE,ALL,Frequency** command.

**NONE**

No multiframe restart files (.RDB [restart database file], .LDHI [load history file], .Xnnn) will be created. If you specify this option, all other arguments will be ignored.

For nonlinear static and static and full transient analyses, this option allows a restart to be done at the last or abort point using the same procedure as in ANSYS 5.5 or earlier (using the .EMAT, .ESAV or .OSAV, and .DB files).

For mode-superposition transient analyses, this option allows a restart from the last point using the .RDSP file and the .DB file (default for mode-superposition analysis).

**Frequency**

Frequency at which the .Xnnn files are written.

**NONE**

Do not write any .Xnnn files for this load step (default for mode superposition analysis).

**LAST**

Write the .Xnnn files for the last substep of the load step only (default for nonlinear static and full transient analyses).

**N**

If  $N$  is positive, write the .Xnnn file every  $N$ th substep of a load step. If  $N$  is negative, write  $N$  equally spaced .Xnnn files within a load step.

In nonlinear static and full transient analyses, negative  $N$  is valid only when **AUTOTS,ON**.

In mode-superposition analyses, negative  $N$  is always valid .

**MAXFILES**

Maximum number of .Xnnn files to save for *Ldstep*.

**0**

Do not overwrite any existing .Xnnn files (default). The total maximum number of .Xnnn files for one run is 999. If this number is reached before the analysis is complete, the analysis will continue but will no longer write any .Xnnn files.

**N**

The maximum number of .Xnnn files to keep for each load step. When  $N$  .Rnnn files have been written for a load step, ANSYS will overwrite the first .Xnnn file of that load step for subsequent substeps.

## Command Default

If the **RESCONTROL** command is not issued during a structural analysis, the .RDB and .LDHI files will be written as described in [Restarting an Analysis](#) in the *Basic Analysis Guide*.

In nonlinear static and full transient analyses, the default behavior is multiframe restart (command default **RESCONTROL,DEFINE,LAST,LAST**). The .Rnnn file will be written at the last substep of the last load step by default. A .Rnnn file will also be written at the iteration prior to the abort point of the run if a Jobname .ABT file was used in batch mode or by pressing the **Stop** button in the GUI, or if the job terminates because of a failure to reach convergence or some other solution error. No information at the aborted substep will be saved to the .Rnnn file.

## Notes

Multiframe restart files are indicated here as `.Xnnn` files. They correspond to `.Rnnn` files for nonlinear static and full transient analyses, and `.Mnnn` files for mode-superposition analyses.

This command sets up the restart parameters for a multiframe restart, which allows you to restart an analysis from any load step and substep for which there is a `.Xnnn` file. You can perform a multiframe restart only for structural static, structural transient (full or mode-superposition methods), and thermal analyses. For information on how to do a multiframe restart and descriptions of the contents of the files used, see [Restarting an Analysis](#) in the *Basic Analysis Guide*. For information on thermal analysis restarts, please see the *Thermal Analysis Guide*.

If you have many substeps for each load step, and are writing `.Xnnn` files frequently, you may want to use `MAXFILES` to limit the number of `.Xnnn` saved, since these files can fill up your disk quickly. You may specify `MAXFILES` and `Frequency` for individual load steps. These arguments will take on the default value or the value defined by `RESCONTROL,,ALL,Frequency,MAXFILES` if they are not explicitly defined for a specific load step.

You can specify a maximum of ten load steps; that is, you can issue the `RESCONTROL,,N` command a maximum of ten times. Specified load steps cannot be changed in a restart.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Restart Control**  
**Main Menu>Solution>Load Step Opts>Nonlinear>Restart Control**

---

## RESET

**Resets all POST1 or POST26 specifications to initial defaults.**

[POST1:Set Up](#)  
[POST26:Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Has the same effect as entering the processor the first time within the run. In POST1, resets all specifications to initial defaults, erases all element table items, path table data, fatigue table data, and load case pointers. In POST26, resets all specifications to initial defaults, erases all variables defined, and zeroes the data storage space.

## Menu Paths

**Main Menu>General Postproc>Reset**  
**Main Menu>TimeHist Postpro>Reset Postproc**

---

## /RESET

Resets display specifications to their initial defaults.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Resets slash display specifications ([/WINDOW](#), [/TYPE](#), [/VIEW](#), etc.) back to their initial default settings (for convenience). Also resets the focus location to the geometric center of the object.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Reset Plot Ctrls

**RESP**, *IR*, *LFTAB*, *LDTAB*, *ITYPE*, *RATIO*, *DTIME*, *TMIN*, *TMAX*

Generates a response spectrum.

[POST26: Special Purpose](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *IR*

Arbitrary reference number assigned to the response spectrum results (2 to NV [[NUMVAR](#)]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with these results.

### *LFTAB*

Reference number of variable containing frequency table (created with [FILLDATA](#) or [DATA](#) command). The frequency table defines the number and frequency of oscillating systems used to determine the response spectrum. The frequency interval need not be constant over the entire range. Frequencies must be input in ascending order.

### *LDTAB*

Reference number of variable containing the displacement time-history.

### *ITYPE*

Defines the type of response spectrum to be calculated:

#### **0 or 1**

Displacement (relative to base excitation).

#### **2**

Velocity.

#### **3**

Acceleration response spectrum (absolute).

### *RATIO*

Ratio of viscous damping to critical damping (input as a decimal number).

**DTIME**

Integration time step (ITS) size used in the numerical integration scheme. This value should be equal to or greater than that actually used in the initial transient analysis. With **ANTYPE,TRANS** data, *DTIME* defaults to a value of  $1/((20)(FMAX))$ , where FMAX is the highest frequency in *LFTAB*. For reduced linear transient dynamic (**ANTYPE,TRANS**) displacement pass data, the ITS read from the file (previously input for *DTIME* in the first load step of the reduced linear transient dynamic (**ANTYPE,TRANS**) analysis) is used for the default.

**TMIN, TMAX**

Specifies a subset of the displacement-time history to be used in the response spectrum calculation. Defaults to the full time range.

**Notes**

Generates a response spectrum from displacement time-history and frequency data.

The ANSYS modal analysis (**ANTYPE,MODAL**) may be followed by a spectrum analysis (**ANTYPE,SPECTR**). This analysis requires a response spectrum input of up to 100 points. This input may be determined from the response spectrum printout or display of this command and input to the modal analysis (by hand). The response spectrum generator uses the displacements from either a full or reduced transient dynamic (**ANTYPE,TRANS**) analysis. If a response spectrum is to be calculated from a given displacement time-history, the displacement time-history may be input to a single one-element reduced linear transient dynamic (**ANTYPE,TRANS**) analysis, so that the calculated output (which should be the same as the input) will be properly located on the file.

The response spectrum is defined as the maximum response of single degree of freedom systems of varying frequency (or period) to a given input support excitation. The equation describing the response of the system in terms of the relative displacement ( $X$ ) is:

$$\ddot{X} + 2\xi_n\omega_n\dot{X} + \omega_n^2 X = -\ddot{X}_o$$

where:

$$\begin{aligned}\Omega_n &= \text{natural frequency of the system, } \sqrt{k/m} \\ \xi_n &= \text{ratio of viscous damping to critical damping, } c/c_{cr} \\ X_o &= \text{ground displacement}\end{aligned}$$

The solution of this equation for the maximum response,  $\bar{X}_{max}$ , at various frequencies results in the spectral response curve. See the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for calculation details.

Calculations are based on a numerical integration scheme with the displacement time-history data from the file as the input ground-forcing function. The integration time step (argument *DTIME* on the **RESP** command) and the damping coefficient (argument *RATIO*) are constant over the frequency range. The number of calculations done per displacement spectral response curve is the product of the number of input solution points  $(TMAX-TMIN)/DTIME$  and the number of oscillating systems (frequencies located in variable *LFTAB*). Input solution points requested (by *DTIME* and the frequency range) at a time not corresponding to an actual displacement solution time on the file are read from the next available time. The user has the option of calculating either a displacement, velocity, or acceleration spectral response.

## Menu Paths

Main Menu>TimeHist Postpro>Generate Spectrm

**RESUME**, *Fname*, *Ext*, --, *NOPAR*, *KNOPTOT*

Resumes the database from the database file.

[DATABASE: Set Up](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

### *Ext*

Filename extension (8 character maximum).

The extension defaults to `DB` if *Fname* is blank.

--

Unused field.

### *NOPAR*

Parameter resume key:

**0**

All data in the database, *including* the scalar parameters, are replaced with the data saved on `File.DB` (default).

**1**

All data in the database, *except* the scalar parameters, are replaced with the data saved on `File.DB`.

---

### Note

This option should *not* be used if array parameters are defined, since existing array parameters might be redefined with arbitrary values. See [PARSAV](#) and [PARRES](#) for a more general method of preventing the replacement of both scalar and array parameters.)

### *KNOPTOT*

If equal to 1, will suppress automatic plot. Otherwise, if the GUI is on and this **RESUME** command was not read from a file, the selected elements from *Fname* are plotted. (If there are no selected elements, selected nodes are plotted. If no nodes, volumes; if no volumes, areas; if no areas, lines; if no lines, keypoints. If there are no selected keypoints, the screen is erased.)

## Notes

The **RESUME** command resumes a database file into the ANSYS program. The command causes the database file (`File.DB`) to be read, thereby resetting the database (including any geometry settings) either a) as it was at the last **SAVE** command, or b) as it was saved with the last **/EXIT** command, whichever was last. For



multiple load step analyses (because only the data for one load step at a time may reside in the database), the load step data restored to the database will correspond to the load step data written when the save occurred.

If the database file was saved in another ANSYS product, it may contain element type and **KEYOPT** specifications which are invalid in the "resuming" product. Immediately after the database resume is completed, you should redefine these invalid element types and **KEYOPT** settings to valid ones (**ET, KEYOPT**).

This command is valid in any processor. If used in SOLUTION, this command is valid only within the first load step.

## Menu Paths

**Utility Menu>File>Resume from  
Utility Menu>File>Resume Jobname.db**

## RESVEC, Key

**Calculates or includes residual vectors.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### Key

Residual vector key:

#### OFF

Do not calculate or include residual vectors. This option is the default.

#### ON

Calculate or include residual vectors.

## Command Default

No residual vectors are calculated or included in the analysis.

## Notes

In a modal analysis, the **RESVEC** command calculates residual vectors. In a [mode-superposition transient dynamic](#), [mode-superposition harmonic](#), [PSD](#) or [spectrum](#) analysis, the command includes residual vectors.

If rigid-body modes exist, pseudo-constraints are required for the calculation. Issue the **D,,,SUPPORT** command to specify only the minimum number of pseudo-constraints necessary to prevent rigid-body motion.

## Menu Paths

**This command cannot be accessed from a menu.**

**RESWRITE**, *Fname*

**Appends results data from the database to a results file.**

[POST1: Special Purpose](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name extension varies as follows:

- .RST for structural, fluid, or coupled-field analyses
- .RTH for thermal or electrical analyses
- .RMG for magnetic analyses
- .RFL for FLOTRAN analyses

**Notes**

The **RESWRITE** command appends to the specified file a data set by writing the results data currently in the database. If the file does not exist, it will be created including the geometry records. The current load step, substep, and time value are maintained. All data (summable and nonsummable) are written.

The command is primarily intended for use in a [top-down substructuring](#) analysis, where the full model is resumed and the results data read from the [use pass](#) results file (**SET**), and subsequently from all substructure [expansion pass](#) results files (**APPEND**). The full set of data in memory can then be written out via the **RESWRITE** command to create a complete results file (as though you had run a nonsubstructured analysis).

**Menu Paths**

**This command cannot be accessed from a menu.**

**\*RETURN**, *Level*

**Returns input stream to a higher level.**

[APDL: Process Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Argument Descriptions*****Level***

Number of levels to move up from the current level.

**Negative --**

Move relative to current level. For example: \*Return,-2 will go up two levels from the current level.

**Positive --**

Move to absolute level. For example: \*Return,2 will go to level 2.

Level 0 is the primary input file.

## Notes

This command is used to jump to the macro call sequence, ending the current macro file, and returning to the line after the calling line in the previous file. Unlike the **\*GO** command, this command may be used inside **\*IF** or **\*DO** constructs.

## Menu Paths

**This command cannot be accessed from a menu.**

**REXPORT**, *Target*, *--*, *--*, *LSTEP*, *SBSTEP*, *Fname*, *Ext*, *--*

**Exports displacements from an implicit run to ANSYS LS-DYNA.**

**SOLUTION: Explicit Dynamics**

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### **Target**

The type of analysis run to which displacements are exported.

#### **OFF**

Ignore initial displacements.

#### **DYNA**

Get initial displacements from an earlier implicit (ANSYS) run and export to an explicit ANSYS LS-DYNA run (Default).

*--*, *--*

Unused fields.

### **LSTEP**

Load step number of data to be exported. Defaults to the last load step.

### **SBSTEP**

Substep number of data to be exported. Defaults to the last substep.

### **Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name does not have a default; you must specify a name. It CANNOT be the current Jobname.

### **Ext**

Filename extension (8 character maximum).

The extension must be an RST extension (default). Currently, only structural results are allowed.

*--*

Unused field.

## Notes

This command exports the displacements, rotations, and temperatures calculated in an ANSYS implicit analysis into the `drelax` file, which is subsequently read in by ANSYS LS-DYNA when a dynamic relaxation or stress initialization is conducted [[EDDRELAX](#)].

This command is not written to the `Jobname.CDB` file when the [CDWRITE](#) command is issued.

## Menu Paths

**Main Menu>Preprocessor>LS-DYNA Options>Constraints>Read Disp**

**Main Menu>Solution>Constraints>Read Disp**

## REZONE, *Option*, *LDSTEP*, *SBSTEP*

Initiates the rezoning process, sets rezoning options, and rebuilds the database.

[SOLUTION: Rezoning](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *Option*

The [rezoning](#) method to employ:

#### **MANUAL**

Manual rezoning. This method is the default.

### *LDSTEP*

The load step number at which rezoning should occur. The default value is the highest load step number found in the `Jobname.Rnnn` files (for the current jobname and in the current directory).

### *SBSTEP*

The substep number of the specified load step (*LDSTEP*) at which rezoning should occur. The default value is the highest substep number found in the specified load step in the `Jobname.Rnnn` files (for the current jobname and in the current directory).

## Notes

The **REZONE** command rebuilds the database (`.db` file) based on the specified load step and substep information, and updates nodes to their deformed position for remeshing.

Before issuing this command, clear the database via the [/CLEAR](#) command.

Currently, support is available for manual rezoning (*Option* = MANUAL) only.

For more information, see "Manual Rezoning" in the [Advanced Analysis Techniques Guide](#).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Solution>Manual Rezoning>Start**

**RFORCE**, *NVAR*, *NODE*, *Item*, *Comp*, *Name*

**Specifies the total reaction force data to be stored.**

[POST26: Set Up](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***NVAR***

Arbitrary reference number assigned to this variable (2 to NV [**NUMVAR**]). Overwrites any existing results for this variable.

***NODE***

Node for which data are to be stored. If *NODE* = P, graphical picking is enabled (valid only in the GUI).

***Item***

Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label.

***Comp***

Component of the item (if required). Valid component labels are shown in the table below.

***Name***

Thirty-two character name identifying the item on printouts and displays. Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

## Notes

Defines the total reaction force data (static, damping, and inertial components) to be stored from single pass (**ANTYPE**,STATIC or TRANS) solutions or from an expansion pass of reduced two-pass (**ANTYPE**,HARMIC or TRANS) solutions.

**Table 251 RFORCE - Valid Item and Component Labels**

Valid item and component labels for node results are:		
Item	Comp	Description
F	X,Y,Z	X, Y, or Z structural force.
M	X,Y,Z	X, Y, or Z structural moment.
HEAT[1]		Heat flow.
FLOW		Fluid flow.
AMPS		Current flow.
FLUX		Magnetic flux.
CSG	X,Y,Z	X, Y, or Z magnetic current segment component.
VLTG		Voltage drop
CURT		Current
CHRG		Charge

- For [SHELL131](#) and [SHELL132](#) elements with KEYOPT(3) = 0 or 1, use the labels HBOT, HE2, HE3, . . . , HTOP instead of HEAT.

## Menu Paths

**Main Menu>TimeHist Postpro>Define Variables**

**Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables**

---

**/RGB**, *Kywrđ*, *PRED*, *PGRN*, *PBLU*, *N1*, *N2*, *NINC*, *NCNTR*

**Specifies the RGB color values for indices and contours.**

[POST26: Set Up](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### *Kywrđ*

Determines how RGB modifications will be applied.

### **INDEX**

Specifies that subsequent color values apply to ANSYS color indices (0-15).

### **CNTR**

Specifies that subsequent color values apply to contours (1-128). Applies to C-option devices only (i.e. X11C or Win32C).

### *PRED*

Intensity of the color red, expressed as a percentage.

### *PGRN*

Intensity of the color green, expressed as a percentage.

### *PBLU*

Intensity of the color blue, expressed as a percentage.

### *N1*

First index (0-15), or contour (1-128) to which the designated RGB values apply.

### *N2*

Final index (0-15), or contour (1-128) to which the designated RGB values apply.

### *NINC*

The step increment between the values *N1* and *N2* determining which contours or indices will be controlled by the specified RGB values.

### *NCNTR*

The new maximum number of contours (1-128).

## Notes

Issuing the **/CMAP** command (with no filename) will restore the default color settings.

## Menu Paths

**Utility Menu>PlotCtrls>Redirect Plots>To GRPH File**

**Utility Menu>PlotCtrls>Redirect Plots>To HPGL File**

**Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File**

**Utility Menu>PlotCtrls>Redirect Plots>To PSCR File**

---

**RIGID**, *Dof1, Dof2, Dof3, Dof4, Dof5, Dof6*

**Specifies known rigid body modes (if any) of the model.**

**SOLUTION:** Dynamic Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

*Dof1, Dof2, Dof3, . . . , Dof6*

Up to six global Cartesian directions of the rigid modes. For a completely free 2-D model, use ALL or UX, UY, ROTZ. For a completely free 3-D model, use ALL or UX, UY, UZ, ROTX, ROTY, ROTZ. For a constrained model, use UX, UY, UZ, ROTX, ROTY, or ROTZ, as appropriate, to specify each and every unconstrained direction which exists in the model (not specifying every direction may cause difficulties in extracting the modes).

## Notes

Specifies known rigid body modes (if any) of the model. This command applies only to a component mode synthesis (CMS) analysis (see the **CMSOPT** command). Any rigid body modes specified must be permitted by the applied displacement constraints (i.e., do not specify a rigid body mode in a constrained direction). Reissue the command to redefine the specification. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>Analysis Options**

**RIGRESP**, *Option, Method, Val1, Val2*

**Specifies the rigid response calculation.**

**SOLUTION:** Spectrum Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**Product Restrictions**

*Option*

Flag to activate or deactivate the rigid response calculation:

**1 (ON or YES)**

Activate.

**2 (OFF or NO)**

Deactivate. This value is the default.

*Method*

Method used to calculate the rigid response:

**GUPTA**

Gupta method.

**LINDLEY**

Lindley-Yow method.

**Val1**

If *Method* = GUPTA, *Val1* represents the frequency F1 in Hertz.

If *Method* = LINDLEY, *Val1* is the Zero Period Acceleration (ZPA). If a scale factor is defined (FACT in the **SVTYP** command), it is used to scale this value

**Val2**

If *Method* = GUPTA, *Val2* represents the frequency F2 in Hertz.

**Notes**

For more information on the rigid response calculation, see [Rigid Responses](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*

This rigid response calculation is only valid for single point response spectrum analysis (**SPOPT**, SPRS) and multiple point response spectrum analysis (**SPOPT**, MPRS) with combination methods (**SRSS**), complete quadratic (**CQC**) or Rosenblueth (**ROSE**)

This command is also valid in PREP7.

**Product Restrictions**

Only *Sptype* = SPRS is allowed in ANSYS Professional.

**Menu Paths**

To be supplied.

---

**RIMPORT**, *Source*, *Type*, *Loc*, *LSTEP*, *SBSTEP*, *Fname*, *Ext*, --, *SPSCALE*, *MSCALE*

**Imports initial stresses from an explicit dynamics run into ANSYS.**

[SOLUTION: FE Body Loads](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**Source**

The type of analysis run from which stresses are imported.

**OFF**

Ignore initial stresses.

**DYNA**

Get initial stresses from an earlier explicit (ANSYS LS-DYNA) run (default).

**Type**

Type of data imported. Note that this is an ANSYS-defined field; the only valid value is STRESS.

**Loc**

Location where the data is imported. Note that this is an ANSYS-defined field; the only valid value is ELEM (data imported at the element integration points).



**LSTEP**

Load step number of data to be imported. Defaults to the last load step.

**SBSTEP**

Substep number of data to be imported. Defaults to the last substep.

**Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name does not have a default; you must specify a name. It CANNOT be the current Jobname.

**Ext**

Filename extension (8 character maximum).

The extension must be an RST extension (default).

--

Unused field.

**SPSCALE**

Stabilization factor. This factor is used in a springback analysis to scale (up or down) the initial stiffness of the applied spring. No default; input a value only if you want to activate stabilization. If *SPSCALE* is blank, stabilization is not activated.

**MSCALE**

Acceptable stabilization stiffness (defaults to  $1.0 \times 10^{-4}$ ). In a springback analysis, iterations will stop when the applied spring stiffness comes down to this value. *MSCALE* is not used if *SPSCALE* is blank.

## Notes

This command imports initial stress information into ANSYS from an earlier explicit (ANSYS LS-DYNA) run. The stress state from [SHELL163](#) and [SOLID164](#) elements in the explicit analysis is imported to the corresponding [SHELL181](#) and [SOLID185](#) implicit elements. For the shell elements, the current shell element thickness is also imported. This command is valid only before the first **SOLVE** command of the implicit analysis (which comes after the explicit analysis) and is ignored if issued after subsequent **SOLVE** commands (that is, stresses will not be re-imported).

**RIMPORT** is typically used to perform springback analysis of sheet metal forming. We recommend that you use [SHELL163](#) elements in the explicit analysis with 3 to 5 integration points through the thickness. This ensures that the through-thickness stress distribution is transferred accurately to the [SHELL181](#) elements. If more than 5 integration points are used, ANSYS imports resultants (forces and moments) to the [SHELL181](#) elements. This implies that linearization of the through-thickness stress distribution is assumed in [SHELL181](#) elements. If [SHELL163](#) uses full integration in the shell plane, stress and thickness data are averaged and then transferred. For the solid elements, the stress at the [SOLID164](#) element centroid is transferred to the [SOLID185](#) element centroid. If [SOLID164](#) has full integration, the stress is averaged and then transferred.

When the *SPSCALE* argument is specified, artificial springs with exponentially decaying stiffness (as a function of iterations) are applied. This technique is recommended only for those cases in which there are severe convergence difficulties. In general, you should first attempt a springback analysis without using the stabilization factors *SPSCALE* and *MSCALE*. (For more information on springback stabilization, see the [ANSYS LS-DYNA User's Guide](#).)

This command is not written to the `Jobname.CDB` file when the **CDWRITE** command is issued. Further, the **RIMPORT** information is not saved to the database; therefore, the **RIMPORT** command must be reissued if the database is resumed.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Import Stress**  
**Main Menu>Solution>Define Loads>Apply>Structural>Other>Import Stress**

---

## RLIST, NSET1, NSET2, NINC

**Lists the real constant sets.**

[PREP7: Real Constants](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NSET1, NSET2, NINC*

List real constant sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL (default), ignore *NSET2* and *NINC* and list all real constant sets [R].

## Notes

The real constant sets listed contain only those values specifically set by the user. Default values for real constants set automatically within the various elements are not listed.

This command is valid in any processor.

## Menu Paths

**Utility Menu>List>Properties>All Real Constants**  
**Utility Menu>List>Properties>Specified Real Constants**

---

## RMALIST

**Lists all defined master nodes for a ROM method.**

[REDUCED ORDER MODELING: Generation Pass](#)

MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>ROM Tool>Setup>Master Nodes>List**

---

**RMANL**, *Fname*, *Ext*, *--*, *Dimn*, *Oper*

**Assigns model database, dimensionality, and operating direction for the ROM method.**

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

***Fname***

Database file name and directory path (248 characters maximum, including directory). The file name defaults to Jobname.

***Ext***

File extension (8 character maximum). The extension defaults to db.

***--***

Unused field.

***Dimn***

Model dimensionality:

- 2**  
2-D models
- 3**  
3-D Models

***Oper***

Primary operating direction:

- X**  
direction
- Y**  
direction
- Z**  
direction

**Notes**

**Required Inputs:**

- Model database containing a structural physics file, title "STRU".
- Model database containing an electrostatic physics file, title "ELEC".
- Model database containing an area or volume component of the electrostatic domain to be morphed, title "AIR".
- Model database containing a node component of the neutral plane nodes, named "NEUN".
- Model database containing node components of conductors, named "CONDi", where "i" is the conductor number.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>ROM Tool>Setup>Model Features**

**RMASTER**, *Node*, *Lab*

**Defines master nodes for the ROM method.**

REDUCED ORDER MODELING: Generation Pass  
MP <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

***Node***

Node number at which master degree of freedom is defined. If *Node* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

***Lab***

Valid labels are "ADD" (default) and "DEL".

**Notes**

Defines master nodes for the ROM. Master nodes are used to track the total displacement of a structure in the operating direction [RMANL]. They may be used as attachment points for 1-D structural elements during a ROM use pass via the UX degree of freedom.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>ROM Tool>Setup>Master Nodes>Define  
Main Menu>ROM Tool>Setup>Master Nodes>Delete

---

**RMCAP**, *RefName*, *C1*, *C2*

**Defines lumped capacitance pairs between conductors C1 and C2 for a ROM method.**

REDUCED ORDER MODELING: Generation Pass  
MP <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

***RefName***

Reference name for capacitance pair definition.

***C1***

First conductor (between 1 and 5).

***C2***

Second conductor (between 1 and 5).

**Notes**

For a capacitance definition between conductor *C1* and *C2*, node components COND%*C1*% and COND%*C2*% (see **CM** command) must be present containing the conductor nodes. If *C1* and *C2* are blank, the capacitance definition with *RefName* will be deleted. (For example, if *C1* = 1, and *C2* = 2, then node components COND1 and COND2 must be defined).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>ROM Tool>Setup>Capacitances>Define>All Capacitances  
 Main Menu>ROM Tool>Setup>Capacitances>Define>Single Capacitance  
 Main Menu>ROM Tool>Setup>Capacitances>Delete

---

## RMCLIST

Lists all lumped capacitance pairs defined.

REDUCED ORDER MODELING: Generation Pass  
 MP <> <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>ROM Tool>Setup>Capacitances>Define>List  
 Main Menu>ROM Tool>Setup>Capacitances>List

---

## /RMDIR, *Dir*

Removes (deletes) a directory.

APDL: Macro Files  
 MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Dir*

The directory to remove. If no path is provided, it will be assumed to be in the current working directory. All files in the directory are also removed.

## Notes

Removes a directory on the computer ANSYS is currently running on. No warning or prompt is given, so use with extreme caution.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## RMFLVEC

**Writes eigenvectors of fluid nodes to a file for use in damping parameter extraction.**

POST1: Special Purpose

MP ME <> <> <> <> <> <> <> <> <> PP <> EME MFS

### Notes

**RMFLVEC** extracts the modal information from the modal results file for all nodes specified in a node component called 'FLUN'. This component should include all nodes which are located at the fluid-structural interface. Mode shapes, element normal orientation, and a scaling factor are computed and stored in a file `Jobname.EFL`. For damping parameter extraction, use the **DMPEXT** command macro. See "Thin Film Analysis" for more information on thin film analyses.

**FLUID136** and **FLUID138** are used to model the fluid interface. Both the structural and fluid element types must be active. The fluid interface nodes must be grouped into a component 'FLUN'. A results file of the last modal analysis must be available.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Menu Paths

**Main Menu>General Postproc>ThinFilm>Extract Eigv**  
**Main Menu>Solution>ThinFilm>DampExtract>Eigenfrequency**  
**Main Menu>Solution>ThinFilm>DampExtract>Frequency Range**  
**Main Menu>Solution>ThinFilm>RayleighDamp**

---

## RMLVSCALE, *Nload, Fact1, Fact2, Fact3, Fact4, Fact5*

**Defines element load vector scaling for a ROM use pass.**

REDUCED ORDER MODELING: Use Pass

MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

### *Nload*

Total number of load cases to be considered within a ROM use pass. If *Nload* = "DELETE", all defined load vectors are deleted.

### *Fact1, Fact2, Fact3, ..., Fact5*

Scale factors applied to load vectors (maximum 5). Defaults to 0.

### Notes

Specifies the element load scale factor applied to a ROM analysis use pass. Element load vectors are extracted from a Static Analysis using the **RMNDISP** command. Up to 5 element load vectors may be scaled and applied to a ROM use pass.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For ROM**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For ROM**  
**Main Menu>Solution>Define Loads>Apply>Load Vector>For ROM**  
**Main Menu>Solution>Define Loads>Delete>Load Vector>For ROM**

## RMMLIST

Lists all mode specifications for the ROM method.

[REDUCED ORDER MODELING: Generation Pass](#)  
 MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>ROM Tool>Mode Selection>List**

## RMMRANGE, *Mode, Key, Min, Max, Nstep, Damp, Scale*

Defines and edits various modal parameters for the ROM method.

[REDUCED ORDER MODELING: Generation Pass](#)  
 MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

### *Mode*

Mode number. Must be lower or equal to the number of modes extracted via the **RMNEVEC** command.

### *Key*

Mode classification key. Valid keys are:

#### **DOMINANT**

Dominant mode

#### **RELEVANT**

Relevant mode

#### **UNUSED**

Unused mode. Do not consider mode in ROM.

### *Min*

Lower bound for fit range of mode.

### *Max*

Upper bound for fit range of mode.

### *Nstep*

Number of equidistant steps in fit range of mode.

**Damp**

Modal damping factor. Defaults to 0.0.

**Scale**

Modal scaling factor.

## Notes

When selected manually (**RMMSELECT**), modes must be classified as dominant, relevant, or unused. Dominant modes (*key* = DOMINANT) are basis functions with large amplitudes. Relevant modes (*key* = RELEVANT) are influenced by the dominant modes but do not cause interactions among themselves due to the small amplitude. This assumption leads to essential speed up of the sample point generator (see **RMSMPLE**).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>ROM Tool>Mode Selection>Edit

---

## RMMSELECT, *Nmode*, *Method*, *Dmin*, *Dmax*

Selects modes for the ROM method.

REDUCED ORDER MODELING: Generation Pass  
MP <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

***Nmode***

Total number of modes to be selected

***Method***

Method for mode selection. Valid methods are:

**TMOD**

Automated selection using a test load. TMOD must be enclosed in single quotes.

**NMOD**

First *Nmode* eigenmodes. NMOD must be enclosed in single quotes.

***Dmin***

Lower bound for total deflection range.

***Dmax***

Upper bound for total deflection range.

## Notes

Select pertinent modes for use in a ROM. Pertinent mode selection may be enhanced by using the deflection state of the structure representative of the operating nature of the device (*Method* = TMOD). A static analysis with an applied Test Load may be used. The test load displacements must be extracted at the neutral plane of the device (if the device is stress-stiffened), or at any plane of the device (non-stress-stiffened). A node component "NEUN" must be defined for the plane of nodes, and the displacements extracted using the **RMNDISP** command prior to issuing this command. If *Method* = NMOD, use the first *Nmode* eigenmodes to select the pertinent modes for the ROM tool. Only those modes are selected that act in the operating direction of the structure [**RMANL**].



**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Required Input Files

jobname.evx, jobname.evy, jobname.evz, jobname.evn, jobname.evl

### Optional Input File

Test load and element load neutral plane displacement files: jobname.tld, jobname.eld

## Menu Paths

Main Menu>ROM Tool>Mode Selection>Select

---

## RMNDISP, LoadT, Loc

Extracts neutral plane displacements from a test load or element load solution for the ROM method.

REDUCED ORDER MODELING: Preparation

MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

### LoadT

Load type. Load type must be an alphanumeric string enclosed in single quotes. Valid load types are 'TLOAD' for the test load and 'ELOAD' for the element load.

### Loc

Determines whether file will be overwritten or appended. Valid labels are 'WRITE' or 'APPEND'. Defaults to 'WRITE' for test load.

## Notes

This command extracts the displacements at a neutral plane of a model. If *LoadT* = 'TLOAD', extract displacements for a test load on a structure that represents the expected deflection state. A test load is used to assist in the automatic mode selection for the ROM mode characterization. If *LoadT* = 'ELOAD', extract the neutral plane displacements for an element load that will be used in the use pass of a ROM analysis. Typical element loads are gravity, and pressure loading. The element loads may be scaled [RMLVSCALE] during the use pass.

The command requires a node component named "NEUN" to be defined. These nodes represent the nodes at the neutral plane of a structure (in the case of a stress-stiffened structure), or at any plane in the structure (non stress-stiffened case).

For *LoadT* = 'TLOAD', node displacements are written to the file jobname.tld. For *LoadT* = 'ELOAD', node displacements are written to the file jobname.eld. Up to 5 element load cases may be written to the file jobname.eld.

This command is only valid in POST1.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Output Files

For a test load jobname.tld

For an element load `jobname.eld`

## Menu Paths

Main Menu>General Postproc>ROM Operations>Extract NP DISP

## RMNEVEC

Extracts neutral plane eigenvectors from a modal analysis for the ROM method.

[REDUCED ORDER MODELING: Preparation](#)

MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

## Notes

This command extracts the eigenvectors at a neutral plane of a model from a modal analysis. The modal analysis must have expanded modes [[MXPAND](#)] in order to process the data. Only the first 9 modes are considered. The command requires a node component named "NEUN" to be defined. These nodes represent the nodes at the neutral plane of a structure (in the case of a stress-stiffened structure), or at any plane in the structure (non stress-stiffened case).

This command is only valid in POST1.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Output Files

`jobname.evx`, `jobname.evy`, `jobname.evz`, `jobname.evn`, `jobname.ev1`

## Menu Paths

Main Menu>General Postproc>ROM Operations>Extract NP Eigv

## RMODIF, NSET, STLOC, V1, V2, V3, V4, V5, V6

Modifies real constant sets.

[PREP7: Real Constants](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **NSET**

Existing set to be modified.

### **STLOC**

Starting location in table for modifying data. For example, if `STLOC = 1`, data input in the `V1` field is the first constant in the set. If `STLOC = 7`, data input in the `V1` field is the seventh constant in the set, etc. Must be greater than zero.

### **V1**

New value assigned to constant in location `STLOC`. If zero (or blank), a zero value will be assigned.

*V2, V3, V4, . . . , V6*

New values assigned to constants in the next five locations. If blank, the value remains unchanged.

## Notes

Allows modifying (or adding) real constants to an existing set **[R]** at any location.

This command is also valid in SOLUTION.

## Menu Paths

**This command cannot be accessed from a menu.**

**RMORE**, *R7, R8, R9, R10, R11, R12*

**Adds real constants to a set.**

[PREP7: Real Constants](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*R7, R8, R9, . . . , R12*

Add real constants 7 to 12 (numerical values or table names) to the most recently defined set.

## Notes

Adds six more real constants to the most recently defined set. Repeat the **RMORE** command for constants 13 to 18, again for 19-24, etc.

If using table inputs ([SURF151](#), [SURF152](#), [FLUID116](#), [CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#), and [CONTA175](#) only), enclose the table name in % signs (e.g., *%tablename%*).

When copying real constants to new sets, ANSYS recommends that you use the command input. If you do use the GUI, restrict the real constant copy to only the first six real constants (real constants seven and greater will be incorrect for both the master and copy set).

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Rotary**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Trans**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Transducer>ElecMech**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Edit Real Cnst**

**RMPORDER**, *Ord1, Ord2, Ord3, Ord4, Ord5, Ord6, Ord7, Ord8, Ord9*

**Defines polynomial orders for ROM functions.**

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

*Ord1, Ord2, Ord3, . . . , Ord9*

Polynomial orders for modes. *Ord<sub>i</sub>* specifies the polynomial order for mode *i*. Modes are ordered as extracted from a modal analysis using the **RMNEVEC** command. Defaults to 0 if mode *i* is unused; default to *nstep(i) - 1* for dominant or relevant modes, where *nstep(i)* is the number of equidistant steps in fit range of mode *i*. *nstep(i)* is automatically set by **RMMSELECT** or modified by the **RMMRANGE** command.

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>ROM Tool>Resp Surface>Poly Order

**RMRESUME**, *Fname, Ext, --*

**Resumes ROM data from a file.**

REDUCED ORDER MODELING: Set Up

MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

*Fname*

Name and directory path of the ROM database file (248 character maximum). Default to Jobname.

*Ext*

Extension of the ROM database file. Default to .rom.

--

Unused field.

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Preprocessor>Loads>Analysis Type>ExpansionPass

Main Menu>Preprocessor>Loads>Analysis Type>New Analysis

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>ROM>Database

Main Menu>ROM Tool>Rom Database>Resume

Main Menu>Solution>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>ExpansionPass

Main Menu>Solution>Analysis Type>New Analysis

## RMREGENERATE

Performs fitting procedure for all ROM functions to generate response surfaces.

REDUCED ORDER MODELING: Generation Pass  
MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

### Notes

The fitting procedure uses modal analysis data and function data generated using the **RMSMPLE** command and specifications set forth in the **RMROPTIONS** command. The files `jobname_ijk.pcs` (modes *i, j, k*) will be generated containing the coefficients of the response surfaces. These files are needed for the ROM Use Pass along with a ROM data base file [**RMSAVE**].

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Input Files

Strain energy and capacitance data file `jobname_ijk.dec`

### Output Files

Response surface coefficients `jobname_ijk.pcs` (modes *i, j, k*)

### Menu Paths

Main Menu>ROM Tool>Resp Surface>Fit Functions

---

## RMROPTIONS, *RefName, Type, Invert*

Defines options for ROM response surface fitting.

REDUCED ORDER MODELING: Generation Pass  
MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

#### *RefName*

Reference name of ROM function to be fitted. Valid reference names are "SENE" for the strain energy of the structural domain and any capacitance reference name previously defined by means of **RMCAP** command for the electrostatic domain.

#### *Type*

Type of fitting function to be applied for regression analysis. Valid types are:

#### **LAGRANGE**

Lagrange type (default)

#### **REDLAGRANGE**

Reduced Lagrange type

#### **PASCAL**

Pascal type

#### **REDPASCAL**

Reduced Pascal type

***Invert***

Flag to specify whether data should be inverted prior to fitting.

**0**

Do not invert data (default for SENE)

**1**

Invert data input on *RefName* field (recommended for capacitance functions).

**Notes**

The objective of response surface fit is to compute an analytical expression for the strain energy and the capacitance as functions of modal amplitudes.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>ROM Tool>Resp Surface>Options**

---

**RMRPLOT**, *RefName*, *Type*, *Mode1*, *Mode2*

**Plots response surface of ROM function or its derivatives with respect to the dominant mode(s).**

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

***RefName***

Reference name of ROM function. Valid reference names are "SENE" for the strain energy of the mechanical domain and any capacitance definition, previously defined by means of the **RMCAP** command, for the electrostatic domain.

***Type***

Type of data to be plotted. Valid types are:

**FUNC**

Response surface (default)

**FIRST**

First derivative of response surface with respect to *Mode1*.

**SECOND**

Second derivative of response surface with respect to *Mode1* and *Mode2*.

***Mode1***

First mode number (used for *Type* = "FIRST" and *Type* = "SECOND" only).

***Mode2***

Second mode number (used for *Type* = "SECOND" only).

**Notes**

The objective of response surface fit is to compute an analytical expression for the strain energy and the capacitance as functions of modal amplitudes. This command assumes that the coefficient files job-

`nam_ijk.pcs` are available [**RMREGENERATE**]. Visualization of the response surface will help to evaluate the validity of the function fit.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>ROM Tool>Resp Surface>Plot

---

## RMRSTATUS, *RefName*

Prints status of response surface for ROM function.

REDUCED ORDER MODELING: Generation Pass  
MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

### *RefName*

Reference name of ROM function. Valid reference names are "SENE" for the strain energy of the mechanical domain and any capacitance reference names [**RMCAP**], for the electrostatic domain.

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>ROM Tool>Resp Surface>Status

---

## RMSAVE, *Fname*, *Ext*, --

Saves ROM data to file.

REDUCED ORDER MODELING: Set Up  
MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

### *Fname*

Name and directory path of the ROM database file. Default to Jobname.

### *Ext*

Extension of the ROM database file. Default to .rom.

--

Unused field.

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>ROM Tool>Rom Database>Save

---

**RMSMPLE**, *Nlgeom*, *Cap*, *Seqslv*, *Eeqslv*

**Runs finite element solutions and obtains sample points for the ROM method.**

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

***Nlgeom***

Specify whether a large or small deflection analysis is to be performed for the mechanical domain:

**OFF (or 0)**

Perform small deflection analysis (default).

**ON (or 1)**

Perform large deflection analysis.

***Cap***

Capacitance calculation method.

**CHARGE**

Compute capacitance based on the charge voltage relationship (default).

**CMATRIX**

Employ CMATRIX macro to calculate capacitance.

***Seqslv***

Solver for structural analysis:

**SPARSE**

Sparse direct equation solver (default).

**PCG**

Pre-conditioned Conjugate Gradient iterative equation solver.

***Eeqslv***

Solver for electrostatic analysis:

**SPARSE**

Sparse direct equation solver (default).

**JCG**

Jacobi Conjugate Gradient iterative equation solver.

**ICCG**

Incomplete Cholesky Conjugate Gradient iterative equation solver.

## Notes

This command prepares and runs multiple finite element solutions on the Structural domain and the Electrostatic domain of a model to collect sample points of data for ROM response curve fitting. The command requires a model database [**RMANL**] and two Physics Files (Structural domain, titled "STRU" and an Electrostatic domain, titled "ELEC"; see **PHYSICS** command). Also required is a complete ROM database generated from the ROM Tools. The *Cap* = CHARGE method is preferred when capacitance to "infinity" is not required. Capacitance conductor pairs are defined by the **RMCAP** command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.



## Required Input

Model Database filename.db  
 ROM Database jobname.rom, jobname.evx, jobname.evy, jobname.evz

## Output Files

Strain energy and capacitance data files jobname\_ijk.dec (mode i, j, k).

## Menu Paths

**Main Menu>ROM Tool>Sample Pt Gen>Compute Points**

## RMUSE, Option, Usefil

**Activates ROM use pass for ROM elements.**

REDUCED ORDER MODELING: Use Pass

MP <> <> <> <> <> <> <> <> <> <> <> PP <> EME <>

### Option

Type of data to be plotted. Valid types are:

#### 1 or "ON"

Activates ROM use pass.

#### 0 or "OFF"

Deactivates ROM use pass.

### Usefil

Name of the reduced displacement file (.rdsp) created by the ROM Use Pass (required field only for the Expansion Pass).

## Notes

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Preprocessor>Loads>Analysis Type>ExpansionPass**

**Main Menu>Preprocessor>Loads>Analysis Type>New Analysis**

**Main Menu>Solution>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>ExpansionPass**

**Main Menu>Solution>Analysis Type>New Analysis**

## RMXPORT

Exports ROM model to external VHDL-AMS simulator.

[REDUCED ORDER MODELING: Generation Pass](#)  
MP <> <> <> <> <> <> <> <> <> <> PP <> EME <>

### Notes

Use this command to generate all files necessary to run the ROM analysis in an external VHDL-AMS Simulator.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Output Files

VHDL files: Initial.vhd, S\_ams\_ijk.vhd, Cxxx\_ams\_ijk.vhd, transducer.vhd.

### Menu Paths

Main Menu>ROM Tool>Export>VHDL-AMS

---

**ROCK**, *CGX*, *CGY*, *CGZ*, *OMX*, *OMY*, *OMZ*

Specifies a rocking response spectrum.

[SOLUTION: Spectrum Options](#)  
MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

*CGX*, *CGY*, *CGZ*

Global Cartesian X, Y, and Z location of center of rotation about which rocking occurs.

*OMX*, *OMY*, *OMZ*

Global Cartesian angular velocity components associated with the rocking.

### Notes

Specifies a rocking response spectrum effect in the spectrum (**ANTYPE**,SPECTR) analysis.

The excitation direction with rocking included is not normalized to one; rather, it scales the spectrum. For more information, see [Participation Factors and Mode Coefficients](#).

This command is also valid in PREP7.

### Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Settings  
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Settings

---

**ROSE**, *SIGNIF*, *Label*, *TD*

Specifies the Rosenblueth mode combination method.

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

**SIGNIF**

Combine only those modes whose significance level exceeds the SIGNIF threshold. For single point, multipoint, or DDAM response (**SPOPT**, SPRS, MPRS, or DDAM), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and does not contribute to the mode combinations. The higher the SIGNIF threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If SIGNIF is specified as 0.0, it is taken as 0.0.

**Label**

Label identifying the combined mode solution output.

**DISP**

Displacement solution (default). Displacements, stresses, forces, etc., are available.

**VELO**

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

**ACEL**

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc. are available.

**TD**

Time duration for earthquake or shock spectrum. *TD* defaults to 10.

**Notes**

For more information on spectrum analysis combination methods, see [Combination of Modes](#)

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Menu paths generated with the final build.

**RPOLY**, *NSIDES*, *LSIDE*, *MAJRAD*, *MINRAD*

Creates a regular polygonal area centered about the working plane origin.

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NSIDES**

Number of sides in the regular polygon. Must be greater than 2.

**LSIDE**

Length of each side of the regular polygon.

**MAJRAD**

Radius of the major (or circumscribed) circle of the polygon. Not used if *LSIDE* is input.

**MINRAD**

Radius of the minor (or inscribed) circle of the polygon. Not used if *LSIDE* or *MAJRAD* is input.

**Notes**

Defines a regular polygonal area on the working plane. The polygon will be centered about the working plane origin, with the first keypoint defined at  $\theta = 0^\circ$ . The area will be defined with *NSIDES* keypoints and *NSIDES* lines. See the **RPR4** and **POLY** commands for other ways to create polygons.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Circumscr Rad**

**Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Inscribed Rad**

**Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Side Length**

**RPR4**, *NSIDES*, *XCENTER*, *YCENTER*, *RADIUS*, *THETA*, *DEPTH*

**Creates a regular polygonal area or prism volume anywhere on the working plane.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NSIDES**

The number of sides in the polygon or prism face. Must be greater than 2.

**XCENTER, YCENTER**

Working plane X and Y coordinates of the center of the polygon or prism face.

**RADIUS**

Distance (major radius) from the center to a vertex of the polygon or prism face (where the first keypoint is defined).

**THETA**

Angle (in degrees) from the working plane X-axis to the vertex of the polygon or prism face where the first keypoint is defined. Used to orient the polygon or prism face. Defaults to zero.

**DEPTH**

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the prism. If *DEPTH* = 0 (default), a polygonal area is created on the working plane.

**Notes**

Defines a regular polygonal area anywhere on the working plane or prism volume with one face anywhere on the working plane. The top and bottom faces of the prism are polygonal areas. See the **RPOLY**, **POLY**, **RPRISM**, and **PRISM** commands for other ways to create polygons and prisms.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Hexagon**

**Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Octagon**  
**Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Pentagon**  
**Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Septagon**  
**Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Square**  
**Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Triangle**  
**Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Hexagonal**  
**Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Octagonal**  
**Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Pentagonal**  
**Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Septagonal**  
**Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Square**  
**Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Triangular**

**RPRISM**, *Z1*, *Z2*, *NSIDES*, *LSIDE*, *MAJRAD*, *MINRAD*

Creates a regular prism volume centered about the working plane origin.

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Z1*, *Z2***

Working plane Z coordinates of the prism.

***NSIDES***

Number of sides in the polygon defining the top and bottom faces of the prism. Must be greater than 2.

***LSIDE***

Length of each side of the polygon defining the top and bottom faces of the prism.

***MAJRAD***

Radius of the major (or circumscribed) circle of the polygon defining the top and bottom faces of the prism. Not used if *LSIDE* is input.

***MINRAD***

Radius of the minor (or inscribed circle) of the polygon defining the top and bottom faces of the prism. Not used if *LSIDE* or *MAJRAD* is input.

## Notes

Defines a regular prism volume centered about the working plane origin. The prism must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) The top and bottom faces are polygonal areas that are parallel to the working plane but neither face need be coplanar with (i.e., "on") the working plane. The first keypoint defined for each face is at  $\theta = 0^\circ$ . See the [RPR4](#) and [PRISM](#) commands for other ways to create prisms.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Circumscr Rad**  
**Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Inscribed Rad**  
**Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Side Length**

**RPSD**, *IR*, *IA*, *IB*, *ITYPE*, *DATUM*, *Name*

**Computes response power spectral density (PSD).**

[POST26: Special Purpose](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previous variable, the previous variable will be overwritten with this result.

***IA*, *IB***

Reference numbers of the two variables to be operated on. If only one, leave *IB* blank.

***ITYPE***

Defines the type of response PSD to be calculated:

**0,1**

Displacement (default).

**2**

Velocity.

**3**

Acceleration.

***DATUM***

Defines the reference with respect to which response PSD is to be calculated:

**1**

Absolute value.

**2**

Relative to base (default).

***Name***

Thirty-two character name identifying variable on listings and displays. Embedded blanks are compressed for output.

## Notes

This command computes response power spectral density (PSD) for the variables referenced by the reference numbers *IA* and *IB*. The variable referred by *IR* will contain the response PSD. You must issue the **STORE,PSD** command first; *File.PSD* must be available for the calculations to occur.

The response power spectral density you calculate will include all modes, including those corresponding to a significance level smaller than the threshold (**PSDCOM**, *SIGNIF*). See [POST26 - Response Power Spectral Density](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more information on these equations.

## Menu Paths

**Main Menu>TimeHist Postpro>Calc Resp PSD**

---

**RSFIT**, *RSlab*, *Slab*, *Name*, *Rmod*, *Ytrans*, *Yval*, *Xfilt*, *CONF*

**Fit a response surface for an output parameter in a solution set.**

PROBABILISTIC: Response Surfaces

MP ME ST PRN <> <> FL EM EH DY PP <> EME MFS

### ***RSlab***

Response surface set label. Identifies the response surface results for later postprocessing. This label can be used as *Rlab* for postprocessing response surface results, and for evaluating the fit results with **RSPLLOT** and **RSPRNT**. You must have separate identification labels for the solution sets and for the response surface sets, because you can perform a response surface fit based on Monte Carlo results. For this case, the original Monte Carlo results are identified by the solution set label provided in the **PDEXE** command and the results generated with the fitted response surfaces are identified by the response surface set label (*RSlab*) provided here. *RSlab* must not contain blanks. Maximum length of this field is 16 characters; if this field contains more than 16 characters, it will be truncated to 16 characters.

### ***Slab***

Solution set label that identifies the probabilistic analysis containing the results to be fitted. This solution set label is defined with the **PDEXE** command.

### ***Name***

Parameter name. The parameter must have been previously defined as a random output parameter with the **PDVAR** command. The result values of this parameter (stored in the solution set identified by *Slab*) are fitted as a function of all random input variables.

### ***Rmod***

Regression model to use for the evaluation of the response surface.

#### **LIN**

Regression model with linear terms only.

#### **QUAD**

Regression model with linear and pure quadratic terms (no cross-terms).

#### **QUAX**

Regression model with linear and all quadratic terms (including cross-terms). (Default.)

### ***Ytrans***

Option for the transformation type applied to the output parameter identified with *Name*.

#### **NONE**

The values of the output parameter *Name* are not transformed (default).

#### **EXP**

The values of the output parameter *Name* are transformed according to  $Y^* = \exp(Y)$ . The transformed values  $Y^*$  are used for the fitting process.

#### **LOGA**

The values of the output parameter *Name* are transformed according to  $Y^* = \log_a(Y)$ , where the basis  $a$  is specified by *YVAL*. The transformed values  $Y^*$  are used for the fitting process.

#### **LOGN**

The values of the output parameter *Name* are transformed according to  $Y^* = \ln(Y)$ . The transformed values  $Y^*$  are used for the fitting process.

#### **LOG10**

The values of the output parameter *Name* are transformed according to  $Y^* = \log_{10}(Y)$ . The transformed values  $Y^*$  are used for the fitting process.

**SQRT**

The values of the output parameter *Name* are transformed according to  $Y^* = \sqrt{y}$ . The transformed values  $Y^*$  are used for the fitting process.

**POW**

The values of the output parameter *Name* are transformed according to  $Y^* = Y^a$ , where the exponent *a* is specified by *Yval*. The transformed values  $Y^*$  are used for the fitting process.

**BOX**

The values of the output parameter *Name* are transformed according to the Box-Cox transformation.

$$Y^* = \frac{Y^{\lambda_i - 1}}{\lambda_i} \quad \text{for } \lambda_i \neq 0$$

$$Y^* = \ln(Y) \quad \text{for } \lambda_i = 0$$

The parameter  $\lambda_i$  is automatically determined within the interval [-2, 2]. The step length for this interval search is given in *Yval*. The transformed values  $Y^*$  are used for the fitting process.

**Yval**

Value needed for *Ytrans* = LOGA, *Ytrans* = POW, and *Ytrans* = BOX. Ignored for all other options of *Ytrans*. For *Ytrans* = LOGA and *Ytrans* = POW, the default value is *Yval* = 1.0. For *Ytrans* = BOX, the default value is *Yval* = 0.01.

**Xfilt**

Option that specifies if irrelevant terms of the regression model should be filtered out.

**NONE**

Regression term filtering is not done. A full regression model as specified with the *Rmod* option is evaluated.

**FSR**

Use the "forward-stepwise-regression" to automatically filter out individual and irrelevant terms of the regression model. If irrelevant terms are sorted out this leads to a smaller number of coefficients to be determined in the regression analysis, which then leads to increased accuracy of the coefficients of the remaining terms in the regression model (default).

**CONF**

Confidence level that is used to filter the terms of the regression model if the *Xfilt* = FSR option has been selected. The value of *CONF* must be between 0.0 and 1.0 (default is 0.95). The higher this value, the more terms will be filtered out. Consequently, higher *CONF* values lead to a fewer terms in the regression model. Likewise, lower *CONF* values lead to more terms being included in the regression model.

**Command Default**

*Rmod* = QUAX, *Ytrans* = NONE, *Xfilt* = FSR, *CONF* = 0.95, *Yval* as explained above.

**Notes**

Evaluates a response surface for a random output parameter *Name*. The result values for the parameter *Name* as stored in the solution set *Slab* are fitted as a function of all random input parameters. There can



be only one response surface set for each solution set. A response surface set can include the fitted response surfaces of one or more output parameters.

---

### Note

If the **RSFIT** command is used for an existing response surface set for which Monte Carlo simulations have already been generated using the **RSSIMS** command then these Monte Carlo samples are deleted by the **RSFIT** command. In this case they need to be generated again using the **RSSIMS** command. It is necessary to delete the Monte Carlo simulations in order to make sure that the samples, their statistics and correlations are consistent with the response surfaces.

## Menu Paths

**Main Menu>Prob Design>Response Surf>Fit Resp Surf**

---

**RSOPT**, *Opt*, *Filename*, *Ext*, *Dir*

Creates or loads the radiosity mapping data file for **SURF251** or **SURF252** element types.

SOLUTION: Radiosity

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Opt*

File option:

#### **SAVE**

Write the radiosity mapping data to a file. (Default)

#### **LOAD**

Read in the specified mapping data file.

### *Fname*

File name for radiosity mapping data file. Defaults to Jobname.

### *Ext*

Filename extension for radiosity mapping data file (default = .rsm).

### *Dir*

Directory path for radiosity mapping data file. If you do not specify a directory path, it will default to your working directory.

## Notes

Use this command to manually create or load a radiosity mapping data file. This command is useful if you want to create the mapping data file without issuing **SAVE** or **CDWRITE**, or if you want to specify that the file be located in a directory other than your working directory. Also use this command to manually load an existing mapping data file during a restart.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**RSPLIT**, *Option, Label, Name1, Name2, Name3, Name4, Name5, Name6, Name7, Name8, Name9, Name10, Name11, Name12, Name13, Name14, Name15, Name16*

**Creates one or more results file(s) from the current results file based on subsets of elements.**

POST1: Special Purpose

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Option*

Specify which results to include for the subset of elements.

#### **ALL**

Write all nodal and element results based on the subset of elements (default).

#### **EXT**

Write only the nodal and element results that are on the exterior surface of the element subset. The results data will be averaged as in PowerGraphics (see **AVRES**) when this results file is brought into POST1. Only valid for solid elements.

### *Label*

Define where the element subset is coming from.

#### **ALL**

Use all selected element components (**CMSEL**) (default).

#### **ESEL**

Use the currently selected (**ESEL**) set of elements. *Name1* defines the results file name.

#### **LIST**

Use *Name1* to *Name16* to list the element component and/or assembly names (that contain element components).

### *Name1, Name2, Name3, . . . , Name16*

Up to 16 element component and/or assembly names (that contain element components).

## **Command Default**

Write all data available for the element subset.

## **Notes**

Results files will be named based on the element component or assembly name, e.g., *Cname.rst*, except for the ESEL option, for which you must specify the results file name (no extension) using the *Name1* field. Note that the *.rst* filename will be written in all uppercase letters (*CNAME.rst*) (unless using the ESEL option); when you read the file, you must specify the filename using all uppercase letters (i.e., *file,CNAME*). You may repeat the **RSPLIT** command as often as needed. All results sets on the results file are processed. Use **/AUX3** to produce a results file with just a subset of the results sets.

Use **INRES** to limit the results data written to the results files.

The subset geometry is also written so that no database file is required to postprocess the subset results files. You must not resume any database when postprocessing one of these results files. The input results file must have geometry written to it (i.e., do not use **/CONFIG,NORSTGM,1**).

Applied forces and reaction forces are not apportioned if their nodes are shared by multiple element subsets. Their full values are written to each results file.

Each results file rennumbers its nodes and elements starting with 1.

This feature is useful when working with large models. For more information on the advantages and uses of the **RSPLIT** command, see [Splitting Large Results Files](#) in the *Basic Analysis Guide*.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**RSPLIT**, *RSlab*, *YName*, *X1Name*, *X2Name*, *Type*, *NPTS*, *PLOW*, *PUP*

**Plot a response surface.**

PROBABILISTIC: Response Surfaces

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *RSlab*

Response Surface set label. Identifies the response surfaces generated by the **RSFIT** command.

### *YName*

Parameter name. The parameter must have been previously defined as a random output parameter with the **PDVAR** command.

### *X1Name*

Parameter name. The parameter must have been previously defined as a random input variable with the **PDVAR** command.

### *X2Name*

Parameter name. The parameter must have been previously defined as a random input variable with the **PDVAR** command. *X2Name* must be different than *X1Name*.

### *Type*

Type of the response surface visualization.

#### **2D**

2-D contour plot.

#### **3D**

3-D surface plot.

### *NPTS*

Number of grid points for both the X1-axis and the X2-axis. The grid points are used for the evaluation of the response surface. The number must be between 1 and 500. Defaults to 20. If *NPTS* = 0 or greater than 500, then a value of 20 is used.

### *PLOW*

Lower probability level used to determine the lower boundary (plotting range) of the curve in case the random input variable does not have a minimum value (such as Gauss). This probability must be between 0.0 and 1.0. Defaults to 0.0025.

### *PUP*

Upper probability level used to determine the upper boundary of the curve. This probability must be between 0.0 and 1.0. Defaults to 0.9975.

## Command Default

$NPTS = 20$ ,  $FLOW = 0.0025$  (applied if distribution type has no minimum),  $PUP = 0.9975$  (applied if distribution type has no maximum)

## Notes

Plots the response surface of an output parameter  $YName$  as a function of two input parameters  $X1Name$  and  $X2Name$ .

If  $FLOW$  is left blank, then a minimum value of the distribution is used for plotting, provided it exists (for example, uniform distribution). If the distribution type has no minimum value (for example, Gaussian distribution), then the default value is used to determine the lower plotting range value. The same is true for the maximum value if  $PUP$  is left blank.

In addition to the response surface, the sampling points that are fitted by the response surface are also plotted by this command. However, sampling points falling outside of the plotting range defined by the  $FLOW$  and  $PUP$  fields will not be shown in the plot.

## Menu Paths

Main Menu>Prob Design>Response Surf>Plt Resp Surf

**RSPRNT**,  $RSlab$ ,  $YName$ ,  $Xout$

**Print a response surface.**

PROBABILISTIC: Response Surfaces

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**$RSlab$**

Response Surface set label. Identifies the response surfaces generated by the **RSFIT** command.

**$YName$**

Parameter name. The parameter must have been previously defined as a random output parameter with the **PDVAR** command. Identifies the parameter for which a response surface has been generated by the **RSFIT** command.

**$Xout$**

An option if an extended print-out of more feedback about goodness-of-fit and the details of the regression analysis of the response surface is requested.

**No**

Use the standard print-out (default).

**Yes**

Use the extended print-out.

## Notes

Prints the results and details of a response surface analysis generated by the **RSFIT** command. For the specified output parameter  $Yname$ , the fitting details such as the individual terms of the response surface model and their corresponding coefficients are listed. The command also produces a comparison of the original values

of *Yname* used for the fitting process and the approximate values derived from the fitting, and some goodness of fit measures.

If *Xout* = Yes, then more information about the regression analysis of the response surface will be printed. For example, the confidence intervals on the regression coefficients and the correlation between the regression coefficients among others.

## Menu Paths

Main Menu>Prob Design>Response Surf>Prn Resp Surf

**RSSIMS**, *RSlab*, *NSIM*, *Seed*

Performs Monte Carlo simulations on response surface(s).

PROBABILISTIC: Response Surfaces

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *RSlab*

Response Surface set label. Identifies the response surfaces generated by the **RSFIT** command.

### *NSIM*

Number of simulation loops on the response surfaces that will be generated for all random output parameters. If the **RSSIMS** command is issued multiple times using the same response surface set label the *NSIM* Monte Carlo simulations is appended to previous ones. The default value for *NSIM* is 10,000.

### *Seed*

Seed value label. Random number generators require a seed value that is used to calculate the next random number. After each random number generation finishes, the seed value is updated and is used again to calculate the next random number. By default ANSYS initializes the seed value with the system time (one time only) when the ANSYS session started.

### **CONT**

Continues updating using the derived seed value (default).

### **TIME**

Initializes the seed value with the system time. You can use this if you want the seed value set to a specific value for one analysis and then you want to continue with a "random" seed in the next analysis. It is not recommended to "randomize" the seed value with the *Seed* = TIME option for multiple analyses. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

### **INIT**

Initializes the seed value using 123457. This value is a typical recommendation used very often in literature. This option leads to identical random numbers for all random input variables when the exact analysis will be repeated, making it useful for benchmarking and validation purposes (where identical random numbers are desired). If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

### **Value**

Uses the specified (positive) value for the initialization of the seed value. This option has the same effect as *Seed* = INIT, except you can chose an arbitrary (positive) number for the initialization. If

the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

## Command Default

*NSIM* = 10,000, *Seed* = CONT

## Notes

Generate the Monte Carlo simulations on the response surfaces that are included in a response surface set. Simulations are evaluated only for the output parameters that have been fitted in a response surface set using the **RSFIT** command.

If the **RSSIMS** command is issued multiple times using the same response surface label the probabilistic design system appends the samples generated here to the previous ones. This way you can start with a moderate *NSIM* number and add more samples if the probabilistic results are not accurate enough.

## Menu Paths

Main Menu>Prob Design>Response Surf>RS Simulation

---

**RSTMAC**, *File1*, *Lstep1*, *Sbstep1*, *File2*, *Lstep2*, *Sbstep2*, *TolerN*, *MacLim*, *Cname*, *KeyPrint*

**Calculates modal assurance criterion (MAC) and matches nodal solutions from two results files.**

POST1: Special Purpose

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *File1*

Name (32 characters maximum) corresponding to first Jobname (.db and .rst files).

### *Lstep1*

Load step number of the results to be read in *File1*.

### *N*

Reads load step *N*. Defaults to 1.

### *Sbstep1*

Substep number of the results to be read in *File1*.

### *N*

Reads substep *N*.

### All

Reads all substeps. This value is the default.

### *File2*

Name (32 characters maximum) corresponding to second Jobname (.rst files).

### *Lstep2*

Load step number of the results to be read in *File2*.

### *N*

Reads load step *N*. Defaults to 1.

**Sbstep2**

Substep number of the results to be read in *File2*.

**N**

Reads substep *N*.

**All**

Reads all substeps. This value is the default.

**TolerN**

Tolerance for nodes matching. If *TolerN* = -1, the nodes on *File2* are mapped into the elements of *File1*. If *TolerN* is positive, the nodes of *File1* and *File2* are matched. The default value is 0.01 (1%).

**MacLim**

Smallest acceptable MAC value. Must be  $\geq 0$  and  $\leq 1$ . The default value is 0.90.

**Cname**

Name of the component from the first Jobname (*File1*). The component must be based on nodes. If unspecified, all nodes are matched and used for MAC calculations. If a component name is specified, only nodes included in the specified component are used. Not applicable to node mapping (*TolerN*=-1).

**KeyPrint**

Printout options:

**0**

Printout matched solutions table. This value is the default.

**1**

Printout matched solutions table and full MAC table.

**2**

Printout matched solutions table, full MAC table and matched nodes table.

**Notes**

The **RSTMAC** command allows the comparison of the solutions from two different results files using the modal assurance criterion (MAC).

The meshes read on *File1* and *File2* may be different. If *TolerN*>0, they are matched. This is the default. If *TolerN* = -1, the solutions are interpolated from *File1*.

Units and coordinate systems must be the same for both models.

The corresponding database file (.db) for *File1* must be resumed before running the command only if a component (*Cname*) is used or if the nodes are mapped (*TolerN* = -1).

Results may be real or complex; however, if results from *File1* have a different type from results in *File2*, only the real parts of the solutions are taken into account in MAC calculations. The analysis type can be arbitrary.

Only structural degrees of freedom are considered. Degrees of freedom can vary between *File1* and *File2*, but at least one common degree of freedom must exist. When node mapping and solution interpolation is performed (*TolerN*=-1), *File1* must correspond to a model meshed in solid and/or shell elements. Other types of elements can be present but the node mapping is not performed for those elements. Interpolation is performed on UX, UY, and UZ.

Solutions read in results files are not all written to the database, therefore, subsequent plotting or printing of solutions is not possible. A **SET** command must be issued after the **RSTMAC** command to post-process each solution.

Comparing cyclic symmetry solutions written on selected set of node (**OUTRES**) is not supported.

**RSTMAC** comparison on cyclic symmetry analysis works only if the number of sectors on *File1* and *File2* are the same. Also comparison cannot be made between cyclic symmetry results and full 360 degree model results (*File1* – cyclic solution, *File2* – full 360 degree model solution).

For more information and an example, see [Comparing Nodal Solutions From Two Models \(RSTMAC\)](#) in the *Basic Analysis Guide*.

## Menu Paths

**This command cannot be accessed from a menu.**

### RSTOFF, *Lab*, *OFFSET*

**Offsets node or element IDs in the FE geometry record.**

[SOLUTION: Analysis Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

#### *Lab*

The offset type:

#### **NODE**

Offset the node IDs.

#### **ELEM**

Offset the element IDs.

#### **OFFSET**

A positive integer value specifying the offset value to apply. The value must be greater than the number of nodes or elements in the existing superelement results file.

## Command Default

Issuing the **RSTOFF** command with no specified argument values applies no offsets.

## Notes

The **RSTOFF** command offsets node or element IDs in the FE geometry record saved in the `.rst` results file. Use the command when expanding superelements in a bottom-up [substructuring](#) analysis (where each superelement is generated individually in a [generation pass](#), and all superelements are assembled together in the [use pass](#)).

With appropriate offsets, you can write results files with unique node or element IDs and thus display the entire model even if the original superelements have overlapping element or node ID sets. (Such results files are incompatible with the `.db` database file saved at the generation pass.)



The offset that you specify is based on the original superelement node or element numbering, rather than on any offset specified via a **SESYMM** or **SETRAN** command. When issuing an **RSTOFF** command, avoid specifying an offset that creates conflicting node or element numbers for a superelement generated via a **SESYMM** or **SETRAN** command.

If you issue the command to set non-zero offsets for node or element IDs, you must bring the geometry into the database via the **SET** command so that ANSYS can display the results. You must specify appropriate offsets to avoid overlapping node or element IDs with other superelement results files.

The command is valid only in the first load step of a [superelement expansion pass](#).

## Menu Paths

**This command cannot be accessed from a menu.**

---

### RSURF, Options, Delopts, ETNUM

**Generates the radiosity surface elements (**SURF251/SURF252**) and stores them in the database.**

SOLUTION: Radiosity

MP ME ST PR <> <> <> <> <> <> <> <> PP <> EME MFS

#### Options

Command options:

##### **CLEAR**

Deletes radiosity surface elements and nodes. The set of elements and nodes to be deleted is defined by *Delopts*. *ETNUM* is ignored.

##### **DEFINE**

Creates the radiosity surface elements and nodes (default).

##### **STAT**

Shows the status/listing. Other command options are ignored.

#### *Delopts*

Deletion options

##### **ALL**

Deletes all radiosity surface elements and nodes.

##### **LAST**

Deletes radiosity surface elements and nodes created by the last **RSURF** command.

#### *ETNUM*

Element type number. Leave blank to indicate the next available number.

## Notes

This command generates the radiosity surface elements based on the **RSYMM** and **RDEC** parameters and stores them in the database. It works only on selected underlying elements that have RDSF flags on them. You can issue multiple **RSURF** commands to build the radiosity model. However, all **RSURF** commands must be issued after issuing the **RSYMM**, and after the model is complete (i.e., after all meshing operations are complete).

If you do issue multiple **RSURF** commands for different regions, you must first mesh the different regions, and then generate the radiosity surface elements on each meshed region individually. Use **RSURF,,,ETNUM** to assign a separate element type number to each region. This procedure allow you to identify the individual regions later in the multi-field analysis.

If the underlying solid elements are higher order, the radiosity surface elements are always lower order (4- or 3-node in 3-D or 2-node in 2-D). Decimation will always occur before any symmetry operations.

For 2-D axisymmetric YR models, the newly-generated nodes can have only positive Y coordinates.

If you have already issued **RSURF** for a surface and you issue **RSURF** again, ANSYS creates a new set of radiosity surface elements and nodes over the existing set, resulting in an erroneous solution.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Surface Mesh>Clear Radiation Surface Mesh**

**Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Surface Mesh>Generate Radiation Surface Mesh**

**Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Surface Mesh>Clear Radiation Surface Mesh**

**Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Surface Mesh>Generate Radiation Surface Mesh**

---

## **RSYMM**, *Option, CS, Axis, NSECT, CONDDVALUE*

**Defines the plane of symmetry or center of rotation for the radiosity method.**

**SOLUTION:Radiosity**

MP ME ST PR <> <> <> <> <> <> PP <> EME MFS

### *Option*

Command options:

#### **CLEAR**

Deletes all symmetry definitions. Other command options are ignored.

#### **DEFINE**

Defines the symmetry (default).

#### **STAT**

Shows the status/listing. Other command options are ignored.

#### **COND**

Turns condensation on or off in the radiosity solver for all defined radiation symmetries. *CS*, *AXIS*, and *NSECT* are ignored. Condensation is the process where equations for symmetrical elements are excluded from the linear equation system. Default is off.

### **CS**

Local coordinate system ( $\geq 11$ ) as defined using the **LOCAL** or **CS** commands or a global coordinate system (0). For planar reflection, the coordinate system origin must be on the plane of symmetry (POS)

and one of its axes must be normal to the POS. For cyclic reflection, the coordinate system origin must be coincident with the center of rotation (COR). Only Cartesian systems are valid.

**Axis**

Axis label of the coordinate system (CS) that is normal to the POS for planar reflection. For 2-D model planar reflections, valid labels are X or Y. For 3-D model planar reflections, valid labels are X, Y, or Z. Must be blank for cyclic reflection. For cyclic reflection, it is assumed that the Z axis is aligned with the axis of rotation.

**NSECT**

Number of cyclic reflections to be done ( $\geq 1$ ). This field must be blank or 0 for planar reflection. Default is blank.

**CONDVALUE**

Condensation key. Used only with *Option* = COND.

**ON**

Turns condensation on in the radiosity solver for all defined radiation symmetries.

**OFF**

Turns condensation off in the radiosity solver for all defined radiation symmetries (default).

**Notes**

This command is used to define the POS for planar reflection or the COR for cyclic reflection. The **RSYMM** command must be issued before **RSURF** and it may be issued multiple times to have more than one planar/cyclic reflection; however, the **RSURF** command processes them in the order they are issued.

For planar reflection, you must define a local coordinate system ( $\geq 11$ ) with its origin on the POS. One of its axes must be aligned so that it is normal to the plane. If possible, use the existing global coordinate system (0).

For cyclic reflection, you must define a local coordinate system ( $\geq 11$ ) with its origin coincident with the COR. Reflections occur about the local Z-axis in the counterclockwise direction. You must align the Z-axis properly. If possible, use the existing global coordinate system (0).

New surface elements generated inherit the properties of the original elements.

For 2-D axisymmetric models, **RSYMM** can be used only for symmetrization in the YR plane. It cannot be used for the theta direction. Use **V2DOPT** in that case.

For 2-D axisymmetric YR models, the newly-generated nodes can have only positive X coordinates.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Clear Symmetry**

**Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Condensation**

**Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Cyclic Symmetry**

Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Planar Symmetry

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Clear Symmetry

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Condensation

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Cyclic Symmetry

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Planar Symmetry

## RSYS, KCN

Activates a coordinate system for printout or display of element and nodal results.

[POST1:Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### KCN

The coordinate system to use for results output:

- 0**  
Global Cartesian coordinate system (default, except for spectrum analyses).
- 1**  
Global cylindrical coordinate system.
- 2**  
Global spherical coordinate system.
- > 10**  
Any existing local coordinate system.

### SOLU

Solution coordinate systems. For element quantities, these are the element coordinate system for each element. For nodal quantities, these are the nodal coordinate systems. If an element or nodal coordinate system is not defined, ANSYS uses the global Cartesian coordinate system. If you issue a **LAYER,N** command (where *N* refers to a layer number), the results appear in the layer coordinate system. (SOLU is the default for spectrum analyses.)

### LSYS

Layer coordinate system. For layered shell and solid elements, the results appear in their respective layer coordinate systems. For a specific layer of interest, issue a **LAYER,N** command (where *N* refers to a layer number). If a model has both nonlayered and layered elements, you can use **RSYS,SOLU** and **RSYS,LSYS** simultaneously (with **RSYS,SOLU** applicable to nonlayered elements and **RSYS,LSYS** applicable to layered elements). To reverse effects of the LSYS option, issue an **RSYS,0** command. LSYS is the default for spectrum analysis.

## Command Default

Activate global Cartesian ( $KCN = 0$ ) coordinate system. For spectrum or PSD analyses (**ANTYPE,SPECTR**), the default is to activate the solution coordinate system ( $KCN = SOLU$ ), which is the only available option.

## Notes

The **RSYS** command activates a coordinate system for printing or displaying element results data such as stresses and heat fluxes, and nodal results data such as degrees of freedom and reactions. ANSYS rotates the results data to the specified coordinate system during printout, display, or element table operations (such as **PRNSOL**, **PRESOL**, **PLNSOL**, and **ETABLE**). You can define coordinate systems with various ANSYS commands such as **LOCAL**, **CS**, **CLOCAL**, and **CSKP**.

If you issue **RSYS** with  $KCN > 10$  (indicating a local coordinate system), and the specified system is subsequently redefined, you must reissue **RSYS** for results to be rotated into the redefined system.

---

### Note

The default coordinate system for certain elements, notably shells, is not global Cartesian and is frequently not aligned at adjacent elements.

The use of **RSYS,SOLU** with these elements can make nodal averaging of component element results, such as  $SX$ ,  $SY$ ,  $SZ$ ,  $SXY$ ,  $SYZ$ , and  $SXZ$ , invalid and is not recommended.

The **RSYS** command has no effect on beam or pipe stresses, which ANSYS displays (via **/ESHAPE,1** and PowerGraphics) in the element coordinate system.

Element results such as stresses and heat fluxes are in the element coordinate systems when  $KCN = SOLU$ . Nodal requests for element results (for example, **PRNSOL,S,COMP**) average the element values at the common node; that is, the orientation of the node is not a factor in the output of element quantities. For nearly all **solid elements**, the default element coordinate systems are parallel to the global Cartesian coordinate system. For **shell elements** and the remaining solid elements, the default element coordinate system can differ from element to element. For layered shell and layered solid elements, ANSYS initially selects the element coordinate system when  $KCN = SOLU$ ; you can then select the layer coordinate system via the **LAYER** command.

Nodal results such as degrees of freedom and reactions can be properly rotated only if the resulting component set is consistent with the degree-of-freedom set at the node. (The degree-of-freedom set at a node is determined by the elements attached to the node.) For example, if a node does not have a  $UZ$  degree of freedom during solution, then any  $Z$  component resulting from a rotation does not print or display in POST1. Therefore, results at nodes with a single degree-of-freedom ( $UY$  only, for example) should not be rotated; that is, they should be viewed only in the nodal coordinate system or a system parallel to the nodal system. (The global Cartesian system--the **RSYS** command default--may not be parallel to the nodal system.) Results at nodes with a 2-D degree-of-freedom set ( $UX$  and  $UY$ , for example) should not be rotated out of the 2-D plane.

### PowerGraphics

For PowerGraphics, ANSYS plots **PLVECT** vector arrow displays (such temperature, velocity, and force) in the global Cartesian coordinate system (**RSYS** = 0). Subsequent operations revert to your original coordinate system.

### PGR File

When you generate a `.PGR` file in SOLUTION, you can use the **Results Viewer** to display your stresses only in the coordinate system in which you write your `.PGR` file. To view stresses in other coordinate systems, load your results file into the Results Viewer and regenerate the data.

## Large Deflections

If large deflection is active (**NLGEOM,ON**), ANSYS rotates the element component result directions by the amount of rigid body rotation.

ANSYS displays the element component results in the initial global coordinate system for the following elements: **SHELL181**, **SHELL281**, **ELBOW290**, **PLANE182**, **PLANE183**, **SOLID185**, **SOLID186**, **SOLID187**, **SOLID272**, **SOLID273**, **SOLID285**, **SOLSH190**, **SHELL208**, and **SHELL209**. All other element result transformations are, therefore, also relative to the initial global system. Nodal degree-of-freedom results are based on the initial (and not the updated) geometry. For all other element types, component results displayed in the co-rotated coordinate system include the element rigid body rotation from the initial global coordinate system, and all other element result transformations are relative to the rotated global system.

## LS-DYNA

You can use the **RSYS** command to rotate stress data for all explicit (ANSYS LS-DYNA) elements *except* **BEAM161**, **COMBI165**, and composite **SHELL163** (**KEYOPT(3) = 1**). In models that contain these element types combined with other explicit elements, you must unselect the unsupported elements before issuing the **RSYS** command. The command does not support strain data for any explicit element types. If you request strain results for explicit elements when **RSYS** is not set to the global Cartesian coordinate system (**KCN = 0**), ANSYS ignores the printing or plotting command. (ANSYS always rotates displacements into the results coordinate system, independent of the explicit element type.)

## Menu Paths

**Main Menu>General Postproc>Options for Outp  
Utility Menu>List>Results>Options**

## RTHICK, *Par*, *ILOC*, *JLOC*, *KLOC*, *LLOC*

**Defines variable thickness at nodes for shell elements.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### *Par*

Array parameter (indexed by node number) that expresses the function to be mapped. For example, func (17) should be the desired shell thickness at node 17.

### *ILOC*

Position in real constant set for thickness at node I of the element (default 1).

### *JLOC*

Position in real constant set for thickness at node J of the element (default 2).

### *KLOC*

Position in real constant set for thickness at node K of the element (default 3).

### *LLOC*

Position in real constant set for thickness at node L of the element (default 4).

---

## Notes

After **RTHICK**, the REAL number will match the ELEM number of each selected element. For example, R (*ILOC*) = func (I NODE), R (*JLOC*) = func (J NODE), etc.

R(*ILOC*), R(*JLOC*), R(*KLOC*), and R(*LLOC*) on a previously defined real constant will be overwritten. Any other real constants on a previously defined real constant set will remain unchanged. This command cannot be used for beam elements.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>Thickness Func**

**Main Menu>Preprocessor>Real Constants>Thickness Func**

**Main Menu>Solution>Load Step Opts>Other>Real Constants>Thickness Func**

---





---

## S Commands

### **SABS**, *KEY*

Specifies absolute values for element table operations.

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *KEY*

Absolute value key:

**0**

Use algebraic values in operations.

**1**

Use absolute values in operations.

### Command Default

Use algebraic values.

### Notes

Causes absolute values to be used in the **SADD**, **SMULT**, **SMAX**, **SMIN**, and **SSUM** operations.

### Menu Paths

Main Menu>General Postproc>Element Table>Abs Value Option

---

### **SADD**, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*, *CONST*

Forms an element table item by adding two existing items.

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *LabR*

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

#### *Lab1*

First labeled result item in operation.

#### *Lab2*

Second labeled result item in operation (may be blank).

#### *FACT1*

Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

**FACT2**

Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

**CONST**

Constant value.

**Notes**

Forms a labeled result (see **ETABLE** command) for the selected elements by adding two existing labeled result items according to the operation:

$$LabR = (FACT1 \times Lab1) + (FACT2 \times Lab2) + CONST$$

May also be used to scale results for a single labeled result item. If absolute values are requested [**SABS**,1], absolute values of *Lab1* and *Lab2* are used.

**Menu Paths**

**Main Menu>General Postproc>Element Table>Add Items**

---

**SALLOW**, *STRS1*, *STRS2*, *STRS3*, *STRS4*, *STRS5*, *STRS6*

**Defines the allowable stress table for safety factor calculations.**

POST1: Element Table

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

**STRS1**, **STRS2**, **STRS3**, ..., **STRS6**

Input up to six allowable stresses corresponding to the temperature points [**TALLOW**].

**Notes**

Defines the allowable stress table for safety factor calculations [**SFACT**,**SFCALC**]. Use the **STAT** command to list current allowable stress table. Repeat **SALLOW** to zero table and redefine points (6 maximum).

Safety factor calculations are not supported by PowerGraphics. Both the **SALLOW** and **TALLOW** commands must be used with the Full Model Graphics display method active.

**Menu Paths**

**Main Menu>General Postproc>Safety Factor>Allowable Strs>Constant**

**Main Menu>General Postproc>Safety Factor>Allowable Strs>Reset Stress**

**Main Menu>General Postproc>Safety Factor>Allowable Strs>Temp-depend**

---

**SAVE**, *Fname*, *Ext*, *--*, *Slab*

**Saves all current database information.**

[DATABASE: Set Up](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `DB` if *Fname* is blank.

***--***

Unused field.

***Slab***

Mode for saving the database:

**ALL**

Save the model data, solution data and post data (element tables, etc.). This value is the default.

**MODEL**

Save the model data (solid model, finite element model, loadings, etc.) only.

**SOLU**

Save the model data and the solution data (nodal and element results).

**Notes**

Saves all current database information to a file (`File.DB`). In interactive mode, an existing `File.DB` is first written to a backup file (`File.DBB`). In batch mode, an existing `File.DB` is replaced by the current database information with no backup. The command should be issued periodically to ensure a current file backup in case of a system "crash" or a "line drop." It may also be issued before a "doubtful" command so that if the result is not what was intended the database may be easily restored to the previous state. A save may be time consuming for large models. Repeated use of this command overwrites the previous data on the file (but a backup file is first written during an interactive run). When issued from within `POST1`, the nodal boundary conditions in the database (which were read from the results file) will overwrite the nodal boundary conditions existing on the database file.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>File>Save as**

**Utility Menu>File>Save as Jobname.db**

## SBCLIST

Lists solid model boundary conditions.

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### Notes

Lists all solid model boundary conditions for the selected solid model entities. See also [DKLIST](#), [DLLIST](#), [DALIST](#), [FKLIST](#), [SFLIST](#), [SFALIST](#), [BFLIST](#), [BFALIST](#), [BFVLIST](#), and [BFKLIST](#) to list items separately.

This command is valid in any processor.

### Menu Paths

Utility Menu>List>Loads>Solid Model Loads

## SBCTRAN

Transfers solid model loads and boundary conditions to the FE model.

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### Notes

Causes a manual transfer of solid model loads and boundary conditions to the finite element model. Loads and boundary conditions on unselected keypoints, lines, areas, and volumes are not transferred. Boundary conditions and loads will not be transferred to unselected nodes or elements. The **SBCTRAN** operation is also automatically done upon initiation of the solution calculations [[SOLVE](#)].

This command is also valid in PREP7.

### Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>All Solid Lds

Main Menu>Solution>Define Loads>Operate>Transfer to FE>All Solid Lds

## SDELETE, *SFIRST*, *SLAST*, *SINC*, *KNOCLEAN*, *LCHK*

Deletes sections from the database.

[PREP7: Cross Sections](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *SFIRST*

First section ID to be deleted; defaults to first available section in the database.

### *SLAST*

Last section ID to be deleted; defaults to last available section in the database.

**SINC**

Increment of the section ID; defaults to 1.

**KNOCLEAN**

Pretension element cleanup key (pretension sections only).

**0**

Perform cleanup of pretension elements (delete pretension elements and reconnect elements split during PSMESH).

**1**

Do not perform cleanup.

**LCHK**

Specifies the level of element-associativity checking:

**NOCHECK**

No element-associativity check occurs. This option is the default.

**WARN**

When a section, material, or real constant is associated with an element, ANSYS issues a message warning that the necessary entity has been deleted.

**CHECK**

The command terminates, and no section, material, or real constant is deleted if it is associated with an element.

## Notes

Deletes one or more specified sections and their associated data from the ANSYS database.

## Menu Paths

**Main Menu>Preprocessor>Sections>Delete Section**

**SE**, *File*, *--*, *--*, *TOLER*

**Defines a superelement.**

[PREP7: Superelements](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**File**

Jobname (case-sensitive, eight-character maximum) of file containing superelement. Defaults to the current Jobname.

*--*, *--*

Unused fields.

**TOLER**

Tolerance used to determine if use pass nodes are noncoincident with master nodes having the same node numbers. Defaults to 0.0001. Use pass nodes will always be replaced by master nodes of the same node number. However, if a use pass node is more than *TOLER* away from the corresponding master node, a warning is generated.

## Notes

Defines a superelement by reading in the superelement matrices and master nodes from the superelement matrix file. The matrix file (`File.SUB`) must be available from the substructure generation pass. The proper element type (`MATRIX50`) must be active [`TYPE`] for this command. A scratch file called `File.SORD` showing the superelement names and their corresponding element numbers is also written.

## Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>From .SUB File

---

**SECCONROLS**, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*, *VAL10*, *VAL11*, *VAL12*, *VAL13*

Supplements or overrides default section properties.

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

*VAL1*, *VAL2*, *VAL3*, . . . , *VAL13*

Values, such as the length of a side or the numbers of cells along the width, that describe the geometry of a section. See the "Notes" (p. 1466) section of this command description for details about these values for the various section types.

## Notes

The **SECCONROLS** command is divided into these operation types: [Beams](#), [Pipes](#), [Shells](#), and [Reinforcings](#).

Values are associated with the most recently issued **SECTYPE** command. The data required is determined by the section type and is different for each type.

**SECCONROLS** overrides the program-calculated transverse-shear stiffness.

The command does not apply to thermal shell elements [SHELL131](#) and [SHELL132](#) or thermal solid elements [SOLID278](#) and [SOLID279](#).

## Beams

### Type: BEAM

Data to provide in the value fields (*VAL1* through *VAL4*):

*TXZ* = User transverse shear stiffness.

- = Unused field.

*TXY* = User transverse shear stiffness.

*ADDMAS* = Added mass per unit length.

## Pipes

### Type: PIPE

Data to provide in the value field (*VAL1*):

*ADDMAS* = Added mass per unit length.

## Shells

### Type: SHELL

Data to provide in the value fields (*VAL1* through *VAL7*):

*E<sub>11</sub>* = User transverse shear stiffness.

*E<sub>22</sub>* = User transverse shear stiffness.

*E<sub>12</sub>* = User transverse shear stiffness.

*ADDMAS* = Added mass per unit area.

*MEMSCF* = Hourglass control membrane scale factor.

*BENSCF* = Hourglass control bending scale factor.

*DRLSTIF* = Drill stiffness scale factor.

#### Subtype: FIBERSIM

For **FiberSIM** analyses, provide data for these *additional* value fields (*VAL8* through *VAL13*):

*ELFSTIF* = Elastic foundation stiffness.

*THKTOL* = Maximum allowable distance between the ANSYS element centroid and the FiberSIM triangle, normal to the FiberSIM triangle. Defaults to the sum of the defined layers (which *does not* represent the laminate thickness.)

*EDGTOL* = Maximum allowable distance from the ANSYS element centroid and an edge of the FiberSIM triangle, in the plane of the FiberSIM triangle. This value is used only when the element centroid is outside of all of the FiberSIM triangles. Defaults to 0.0. A non-zero value may increase running time slightly. ANSYS recommends specifying only a very small tolerance value (as small as possible).

*ANGTOL* = Maximum allowable angle tolerance in degrees between the normals of the ANSYS element and the FiberSIM triangle. The default value is 10.

*NEL* = Activates debugging output (via the **LAYLIST** command) for the first *NEL* elements after a new section is called. The default value is 0.

*TKFLG* = Layer thickness flag (valid only with FiberSIM version 5.1 or later):

0 - Determine the layer thickness based on the FiberSIM file data.

1 - Use the layer thickness specified via the **SECDATA** command.

In an analysis that uses **FiberSIM** data to define sections, issue the **SECCONTROLS** command *after* defining the geometry for your section (that is, after issuing all necessary **SECDATA** commands).

## Reinforcing

### Type: REINF

#### Subtype: DISCRETE

Data to provide in the value field (*VAL1*):

*TENSKEY* = Flag specifying tension-and-compression, tension-only, or compression-only fibers:

0 = Tension *and* compression.

1 = Tension only.

-1 = Compression only. The default value is 0.

The value specified applies to all fibers defined in the current section.

## Menu Paths

Main Menu>Preprocessor>Sections>Beam>Sect Control

Main Menu>Preprocessor>Sections>Pipe>Add

Main Menu>Preprocessor>Sections>Pipe>Edit

Main Menu>Preprocessor>Sections>Reinforcing>Sect Control

Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit

---

**SECDATA**, VAL1, VAL2, VAL3, VAL4, VAL5, VAL6, VAL7, VAL8, VAL9, VAL10, VAL11, VAL12

Describes the geometry of a section.

PREP7: Cross Sections

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

VAL1, VAL2, VAL3, . . . , VAL10

Values, such as thickness or the length of a side or the numbers of cells along the width, that describe the geometry of a section. The terms VAL1, VAL2, etc. are specialized for each type of cross-section.

## Notes

The **SECDATA** command defines the data describing the geometry of a section. The command is divided into these section types: [Beams](#), [Pipes](#), [Axisymmetric](#), [Taper](#), [Shells](#), [Pretension](#), [Joints](#), [Reinforcing](#), and [Contact](#).

The data input on the **SECDATA** command is interpreted based on the most recently issued **SECTYPE** command. The data required is determined by the section type and subtype, and is different for each one.

In an analysis using [FiberSIM](#) data (**SECTYPE**,,SHELL,FIBERSIM), the **SECDATA** command *does not* define the order of the layers.

---

## Beams

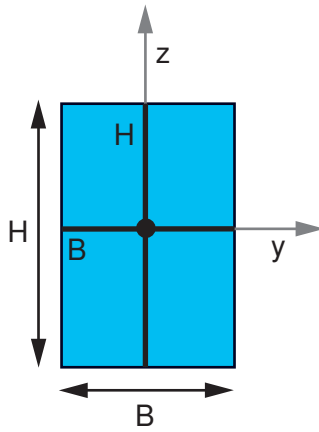
### Type: BEAM

Beam sections are referenced by beam elements such as [BEAM188](#) or [BEAM189](#). Not all **SECOFFSET** location values are valid for each subtype.

Type: BEAM, Subtype: RECT

Type: BEAM, Subtype: QUAD





Data to provide in the value fields:

$B, H, Nb, Nh$

where

$B$  = Width

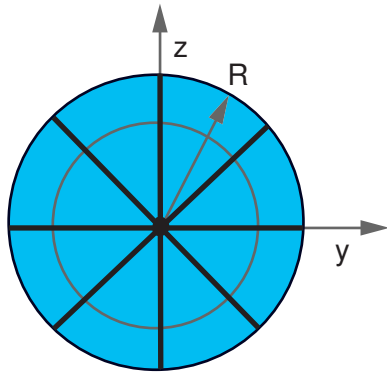
$H$  = Height

$Nb$  = Number of cells along width; default = 2

$Nh$  = Number of cells along height; default = 2

$Nb * Nh$

**Type: BEAM, Subtype: CSOLID**



Data to provide in the value fields:

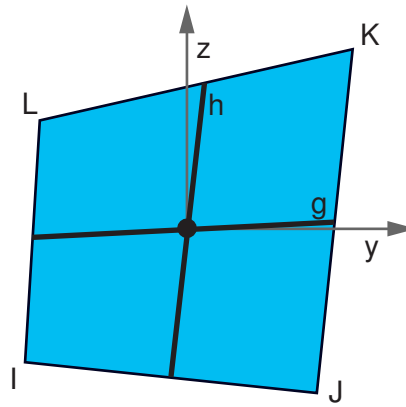
$R, N, T$

where

$R$  = Radius

$N$  = Number of divisions around the circumference;  $N \geq 8$ , default = 8

$T$  = Number of divisions through the radius; default = 2



Data to provide in the value fields:

$yI, zI, yJ, zJ, yK, zK, yL, zL, Ng, Nh$

$yI, zI, yJ, zJ, yK, zK, yL, zL$  = Coordinate location of various points

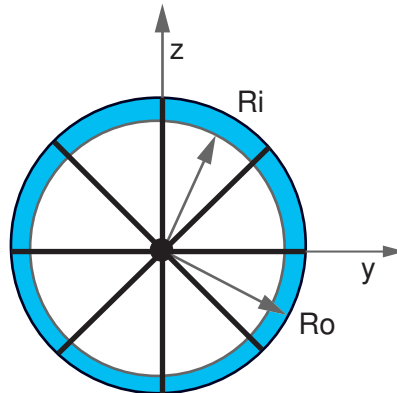
$Ng$  = Number of cells along  $g$ ; default = 2

$Nh$  = Number of cells along  $h$ ; default = 2

$Ng * Nh$

Degeneration to triangle is permitted by specifying the same coordinates for cells along an edge.

**Type: BEAM, Subtype: CTUBE**



Data to provide in the value fields:

$Ri, Ro, N$

where

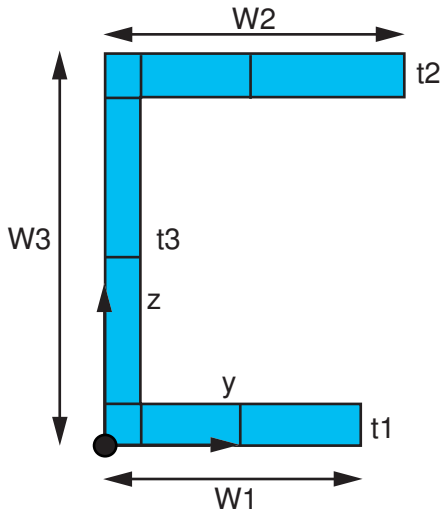
$Ri$  = Inner radius of the tube

$Ro$  = Outer radius of the tube

$N$  = Number of cells along the circumference;  $N \geq 8$ , default = 8

This subtype is similar to type PIPE. However, elements using PIPE account for internal or external pressures, whereas elements using CTUBE do not.

**Type: BEAM, Subtype: CHAN**



Data to provide in the value fields:

$W1, W2, W3, t1, t2, t3$

where

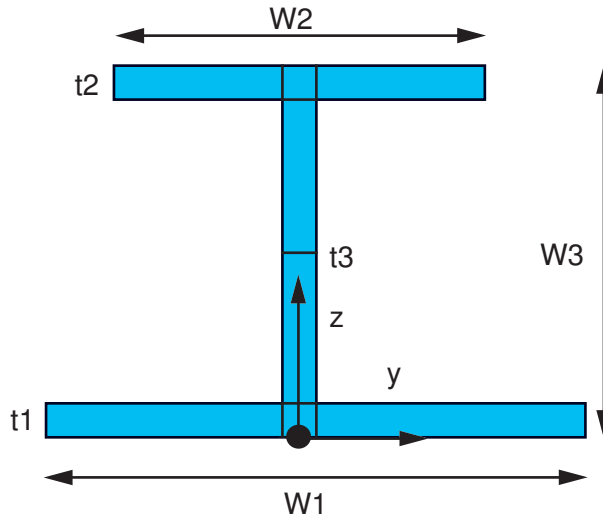
$W1, W2$  = Lengths of the flanges

$W3$  = Overall depth

$t1, t2$  = Flange thicknesses

$t3$  = Web thicknesses

**Type: BEAM, Subtype: I**



Data to provide in the value fields:

$W1, W2, W3, t1, t2, t3$

where

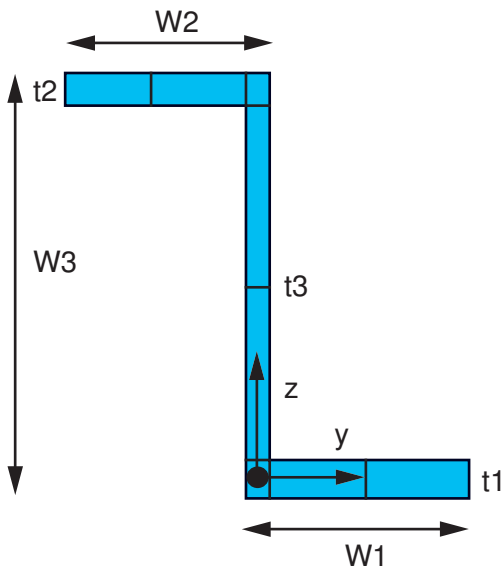
$W1, W2$  = Width of the top and bottom flanges

$W3$  = Overall depth

$t1, t2$  = Flange thicknesses

$t3$  = Web thicknesses

**Type: BEAM, Subtype: Z**



Data to provide in the value fields:

$W1, W2, W3, t1, t2, t3$

where

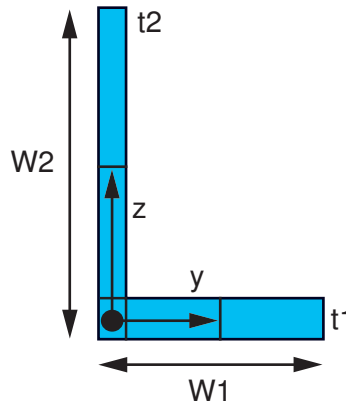
$W1, W2$  = Flange lengths

$W3$  = Overall depth

$t1, t2$  = Flange thicknesses

$t3$  = Stem thicknesses

**Type: BEAM, Subtype: L**



Data to provide in the value fields:

$W1, W2, t1, t2$

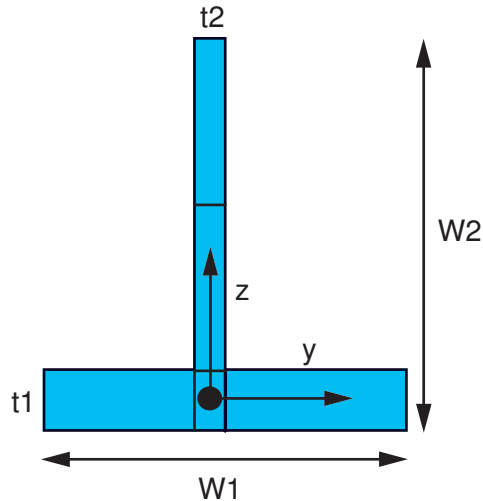
where

$W1, W2$  = Leg lengths

$t1, t2$  = Leg thicknesses

If  $W2$  is a negative value, the section will be flipped.

**Type: BEAM, Subtype: T**



Data to provide in the value fields:

$W1, W2, t1, t2$

where

$W1$  = Flange width

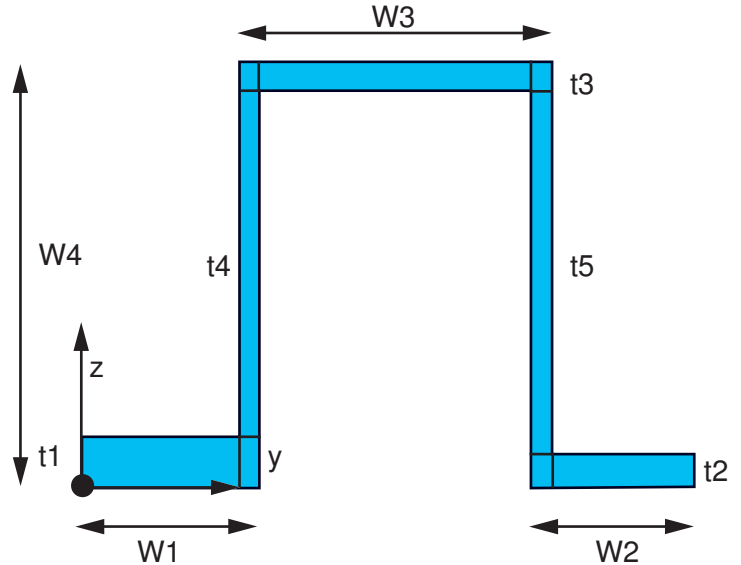
$W2$  = Overall depth

$t1$  = Flange thicknesses

$t2$  = Stem thicknesses

If  $W2$  is a negative value, the section will be flipped.

**Type: BEAM, Subtype: HATS**



Data to provide in the value fields:

$W1, W2, W3, W4, t1, t2, t3, t4, t5$

where

$W1, W2$  = Width of the brim

$W3$  = Width of the top of the hat

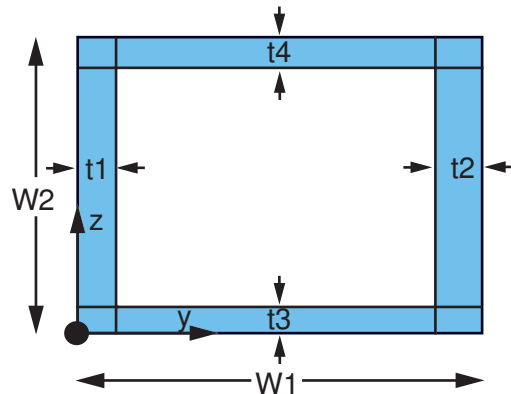
$W4$  = Overall depth

$t1, t2$  = Thickness of the brim

$t3$  = Thickness of the top of the hat

$t4, t5$  = Web thicknesses

**Type: BEAM, Subtype: HREC**



Data to provide in the value fields:

$W1, W2, t1, t2, t3, t4$

where

$W1$  = Outer width of the box

$W2$  = Outer height of the box

$t1, t2, t3, t4$  = Wall thickness

**Type: BEAM, Subtype: ASEC**

Arbitrary: User-supplied integrated section properties instead of basic geometry data.

Data to provide in the value fields:

$A, I_{yy}, I_{yz}, I_{zz}, I_w, J, CG_y, CG_z, SH_y, SH_z$

where

$A$  = Area of section

$I_{yy}$  = Moment of inertia about the y axis

$I_{yz}$  = Product of inertia

$I_{zz}$  = Moment of inertia about the z axis

$I_w$  = Warping constant

$J$  = Torsional constant

$CG_y$  = Y coordinate of centroid

$CG_z$  = Z coordinate of centroid

$SH_y$  = Y coordinate of shear center

$SH_z$  = Z coordinate of shear center

$TK_z$  = Thickness along Z axis

$TK_y$  = Thickness along Y axis

**SECPLOT** cannot display an ASEC plot.

**Type: BEAM, Subtype: MESH**

User-defined mesh.

Data required is created by the **SECWRITE** command and is read into ANSYS by the **SECREAD** command.

When a user mesh is input via **SECREAD**, ANSYS calculates the area, second moments of inertia, centroid, torsion constant, warping rigidity, and shear center.

If you redefine a material for a composite cross-section after creating the section, you must re-issue the **SECREAD** command.

---

## Pipes

### Type: PIPE

Pipe sections are referenced by the **PIPE288**, **PIPE289**, and **ELBOW290** elements.

Data to provide in the value fields:

$D_o, T_w, N_c, S_s, N_t, M_{int}, M_{ins}, T_{ins}$

where

$D_o$  = Outside diameter of pipe. Does not include  $T_{ins}$ .

$T_w$  = Wall thickness.

$N_c$  = Number of cells along the circumference. This value must be 8 or greater. The default value is 8.

$S_s$  = Section number of the shell representing the pipe wall. Valid with **ELBOW290** only.

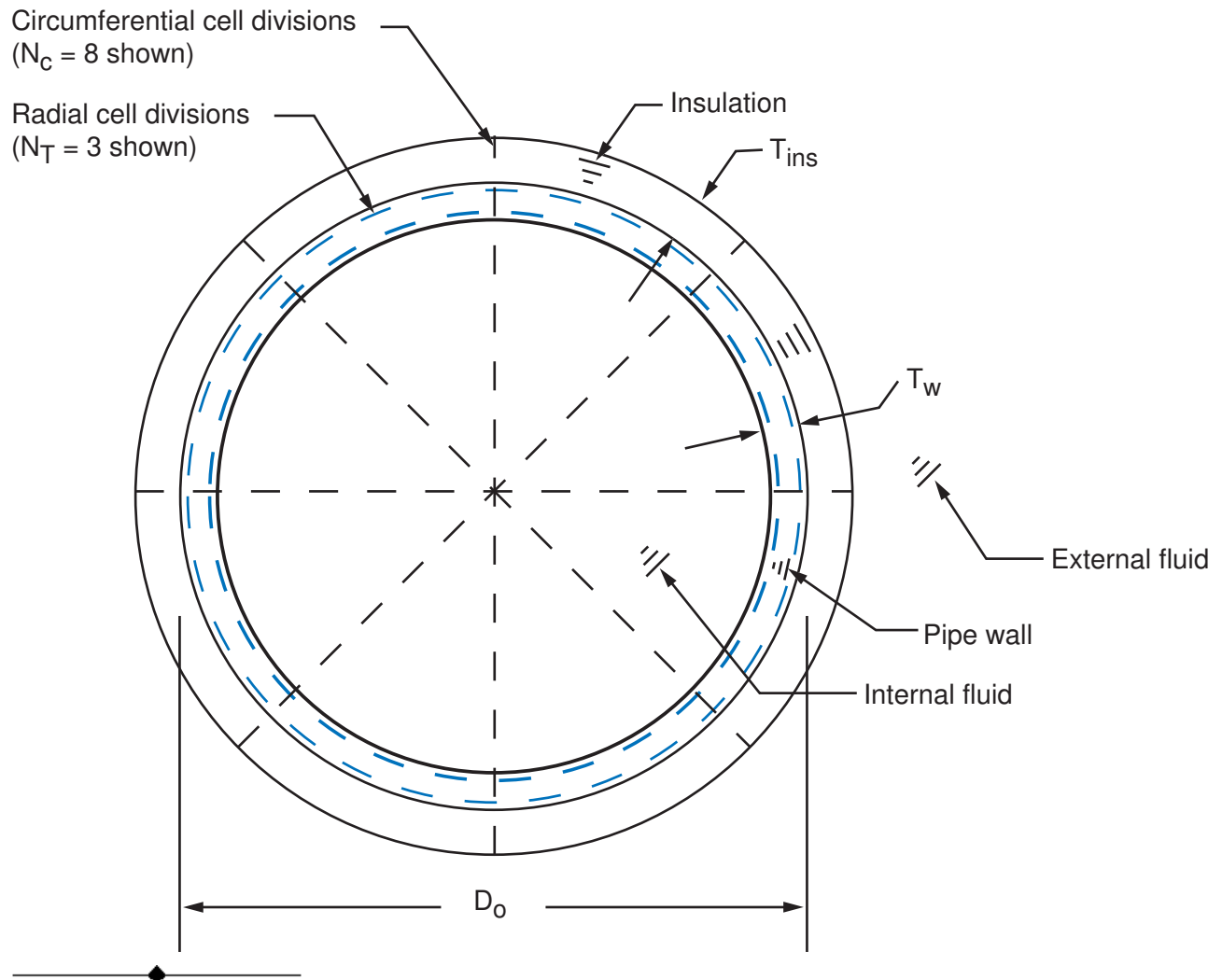
$N_t$  = Number of cells through the pipe wall. Valid values are 1, 3, 5, 7, and 9. The default value is 1. Cells are graded such that they are thinner on the inner and outer surfaces. Valid with **PIPE288** and **PIPE289** only.

$M_{int}$  = Material number of fluid inside of the pipe. The default value is 0 (no fluid). This value is used to input the density of the internal fluid. Valid with **PIPE288** and **PIPE289** only.

$M_{ins}$  = Material number of material external to the pipe (such as insulation, biofouling, or armoring). The default value is 0 (no external material). This value is used to input the density of the external material. Valid with **PIPE288** and **PIPE289** only.

$T_{ins}$  = Thickness of material external to the pipe, such as insulation. The default value is 0 (no external material). Valid with **PIPE288** and **PIPE289** only.

External material ( $M_{ins}$ ) adds mass and increases hydraulic diameter, but does not add to stiffness. ( $M_{ins}$  is valid for **PIPE288** and **PIPE289** only.)



## Taper

### Type: TAPER

Taper sections are referenced by [BEAM188](#), and [BEAM189](#) elements. After specifying the tapered section type ( **SECTYPE,,TAPER**), issue separate **SECDATA** commands to define each end of the tapered beam.

Data to provide in the value fields:

*Sec\_IDn*, *XLOC*, *YLOC*, *ZLOC*

where

*Sec\_IDn* = Previously defined beam section at ends 1 (I) and 2 (J).

*XLOC*, *YLOC*, *ZLOC* = The location of *Sec\_IDn* in the global Cartesian coordinate system.

## General Axisymmetric

### Type: AXIS

General axisymmetric sections are referenced by the [SURF159](#), [SOLID272](#), and [SOLID273](#) elements. Use this command to locate the axisymmetric axis.

Data to provide in the value fields:

**Pattern 1 (two points):**

1,  $X1, Y1, Z1, X2, Y2, Z2$

where  $X1, Y1, Z1, X2, Y2, Z2$  are global Cartesian coordinates.

**Pattern 2 (coordinate system number plus axis [1 = x, 2 = y, 3 = z]):**

2,  $csys, axis$

where  $csys$  is a Cartesian coordinate system.

**Pattern 3 (origin plus direction):**

3,  $XO, YO, ZO, xdir, ydir, zdir$

where  $XO, YO, ZO$  are global Cartesian coordinates and  $xdir, ydir,$  and  $zdir$  are direction cosines.

## Shells

### Type: SHELL

Shell sections are referenced by the [SHELL131](#), [SHELL132](#), [SHELL181](#), [SOLID185](#) Layered Solid, [SOLID186](#) Layered Solid, [SOLSH190](#), [SHELL208](#), [SHELL209](#), [SOLID278](#) Layered Solid, [SOLID279](#) Layered Solid, and [SHELL281](#) elements.

Data to provide in the value fields:

$TK, MAT, THETA, NUMPT, LayerName$

where

$TK$  = Thickness of shell layer. Zero thickness (not valid for [SHELL131](#) and [SHELL132](#)) indicates a dropped layer. The sum of all layer thicknesses must be greater than zero. The total thickness can be tapered via the [SECFUNCTION](#) command.

$MAT$  = Material ID for layer (any current-technology material model is available for [SHELL181](#), [SOLID185](#) Layered Solid, [SOLID186](#) Layered Solid, [SOLSH190](#), [SHELL208](#), [SHELL209](#), [SOLID278](#) Layered Solid, and [SOLID279](#) Layered Solid, including UserMat).  $MAT$  is required for a composite (multi-layered) laminate. For a homogeneous (single-layered) shell, the default is the [MAT](#) command setting. Use the [TREF](#) and/or the [MP](#), [REFT](#) commands to address multiple reference temperatures.

$THETA$  = Angle (in degrees) of layer element coordinate system with respect to element coordinate system (ESYS).

$NUMPT$  = Number of integration points in layer. The GUI permits 1, 3, 5, 7, or 9 points (default = 3).

However a higher odd number may be specified in the command. The integration rule used is Simpson's Rule. ( $NUMPT$  is not used by [SHELL131](#) and [SHELL132](#).)

$LayerName$  = The layer name (up to 72 characters) in an analysis using [FiberSIM](#) data ([SECTYPE](#), [SHELL](#), [FIBERSIM](#)). The layer name is case-sensitive and must match the  $ply$  name in the [FiberSIM](#) .xml file. (In a non-[FiberSIM](#) analysis, this value serves only as a comment in the input.)

In an analysis using [FiberSIM](#) data:

- The  $THETA$  value is ignored,
- The order of the **SECDATA** commands does not matter as the layer order comes from the .xml file. Therefore, repeated values of  $LayerName$  should not be used.
- Every  $LayerName$  used on the .xml file should be on a separate **SECDATA** command.

Issue separate **SECDATA** commands to define each of the layers in the section.

## Pretension

### Type: PRETENSION

Pretension sections are referenced by the [PRETS179](#) element.

Data to provide in the value fields:

*node, nx, ny, nz*

where

*node* = Pretension node number.

*nx* = Orientation in global Cartesian x direction.

*ny* = Orientation in global Cartesian y direction.

*nz* = Orientation in global Cartesian z direction.

The following usage is typical:

```
SECTYPE, 1, PRETENSION
SECDATA, 13184, 0.000, 0.000, 1.000
SECMODIF, 1, NAME, example
SLOAD, 1, PL01, TINY, FORC, 100.00 , 1, 2
```

The *PRETENSION* section options of **SECTYPE** and **SECDATA** are documented mainly to aid in the understanding of data written by **CDWRITE**. ANSYS recommends that you generate pretension sections using **PSMESH**.

## Joints

### Type: JOINT

Joint sections are referenced by [MPC184](#) joint elements.

Data to provide in the value fields:

*length1, length2, length3, angle1, angle2, angle3*

where

*length1-3* = Reference lengths used in the constitutive calculations.

*angle1-3* = Reference angles used in the constitutive calculations.

The following table shows the lengths and angles to be specified for different kinds of joints.

Joint	Subtype	Reference lengths and angles
Revolute	REVO	angle1 for x-axis revolute joint angle3 for z-axis revolute joint
Universal	UNIV	angle1 and angle3
Slot	SLOT	length1
Point-in-plane	PINP	length2 and length3,
RevoluteTranslational	PRIS	length1

Joint	Subtype	Reference lengths and angles
Cylindrical	CYLI	length1 and angle1 for x-axis cylindrical joint length3 and angle3 for z-axis cylindrical joint
Planar	PLAN	length2, length3, and angle1 for x-axis planar joint length1, length2, and angle3 for z-axis planar joint
Weld	WELD	(not used)
Orient	ORIE	(not used)
Spherical	SPHE	(not used)
General	GENE	length1, length2, length3, angle1, angle2, angle3 -- Depends upon which "unconstrained" relative degrees of freedom are available.
Screw	SCRE	length3 and angle3

The reference length and angle specifications correspond to the free relative degrees of freedom in a joint element for which constitutive calculations are performed. These values are used when stiffness and/or damping are specified for the joint elements.

If the reference lengths and angles are not specified, they are calculated from the default or starting configuration for the element.

See [MPC184](#) or the individual joint element descriptions for more information on joint element constitutive calculations.

## Reinforcing

### Type: REINF

Reinforcing sections are referenced by [REINF264](#) and [REINF265](#) elements.

### Type: REINF, Subtype: DISCRETE

This subtype defines discrete reinforcing fibers with arbitrary orientations. Issue separate **SECDATA** commands to define each reinforcing fiber.

Data to provide in the value fields:  
where

*MAT*, *A*, *PATT*, *V1*, *V2*, *V3*, *V4*, *V5*

*MAT* = Material ID for the fiber. (Refer to the [REINF264](#) documentation for available material models.)

The material ID is required for all fibers, and no default for this value is available.

*A* = Cross-section area of the reinforcing fiber.

*PATT* = Input pattern code indicating how the location of this fiber is defined with respect to the base element. Available input patterns are *LAY<sub>n</sub>*, *EDG<sub>o</sub>*, and *BEAM*.

*V1*, *V2*, *V3*, *V4*, *V5* = Values to define the location of the reinforcing fiber (depending on the *PATT* pattern code used), as shown:



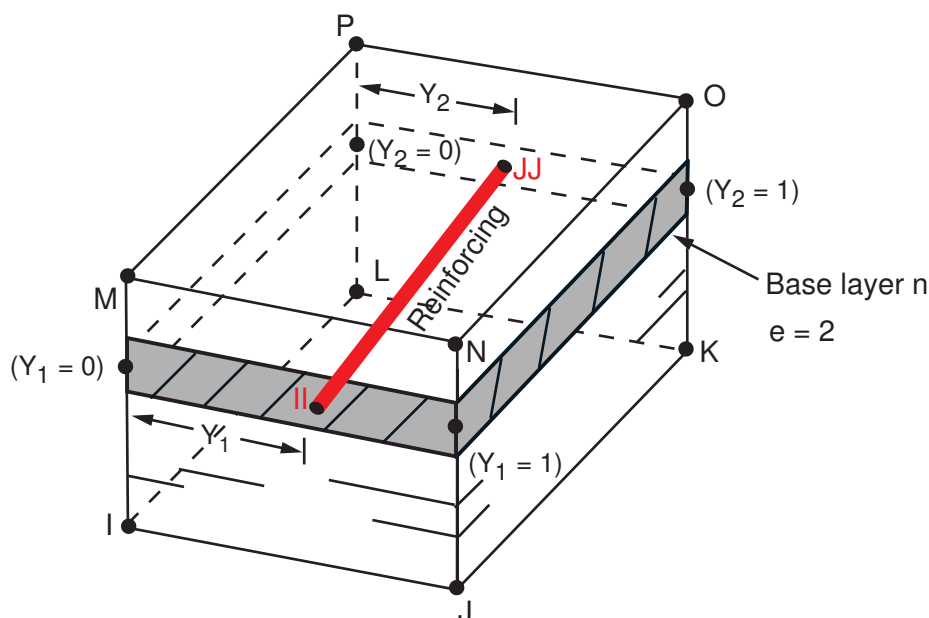
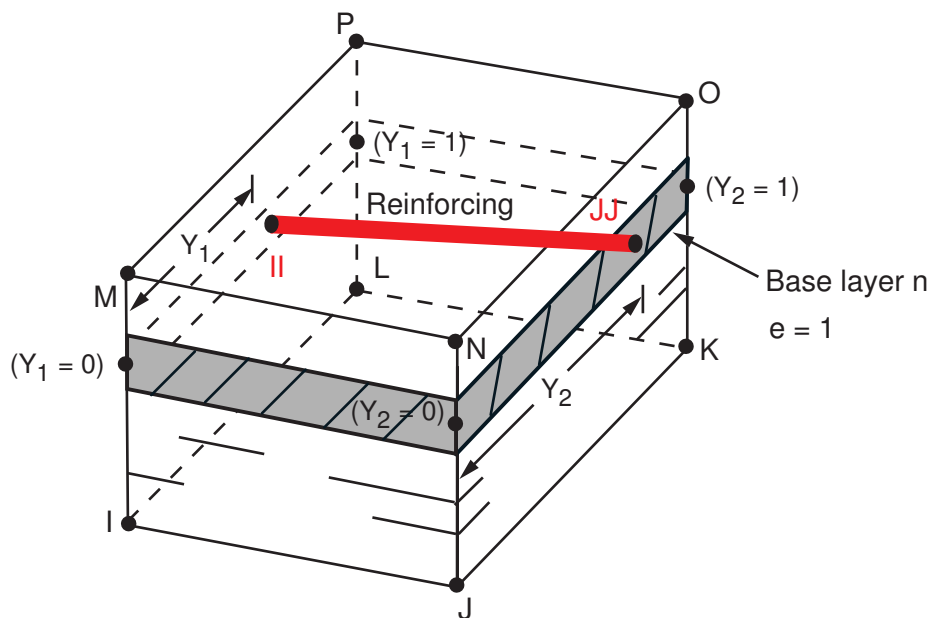
**PATT : LAY $n$** 

**Description:** The discrete reinforcing fiber is placed in the middle of a layer in a layered base element. The orientation of the fiber within the layer is adjustable via offsets with respect to a specified element edge.

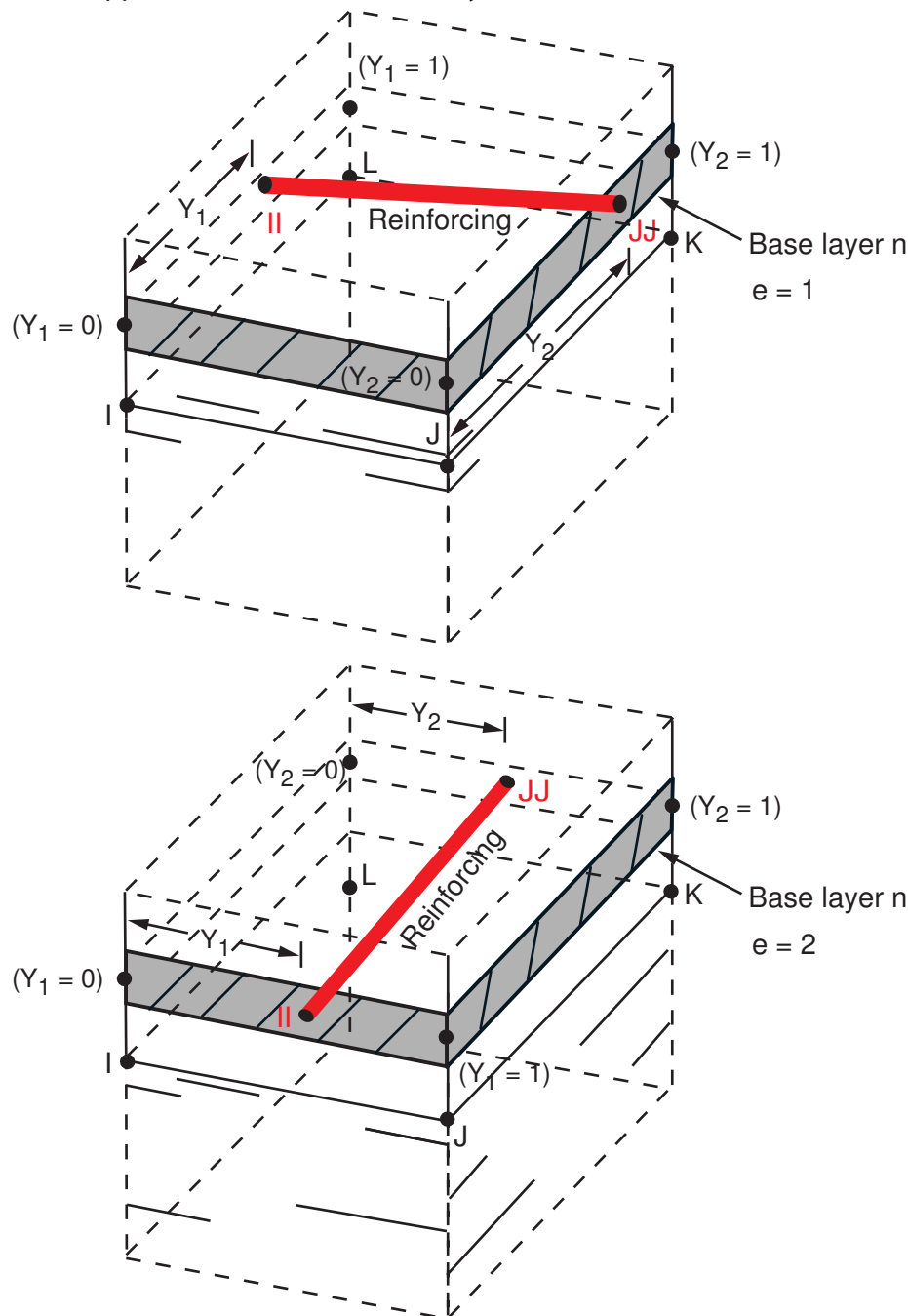
**Required input:**

- V1 (or  $n$ ) -- The number of the layer in the base element on which to apply the reinforcing fiber. The default value is 1.
- V2 (or  $e$ ) -- The number to indicate the element edge to which the offsets are measured. The default value is 1.
- V3 and V4 (or  $Y1$  and  $Y2$ ) -- The normalized distances from the fiber to the two ends of the specified element edge. Valid values for  $Y1$  and  $Y2$  are 0.0 through 1.0. The default value of  $Y1$  is 0.5. The default value of  $Y2$  is  $Y1$ .

When applied to 8-node or 20-node layered solid elements:



When applied to 4-node or 8-node layered shell elements:



### **PATT : EDGo**

**Description:** The orientation of the discrete reinforcing fiber is similar to one of the specified element edges. The fiber orientation can be further adjusted via offsets with respect to the specified element edge.

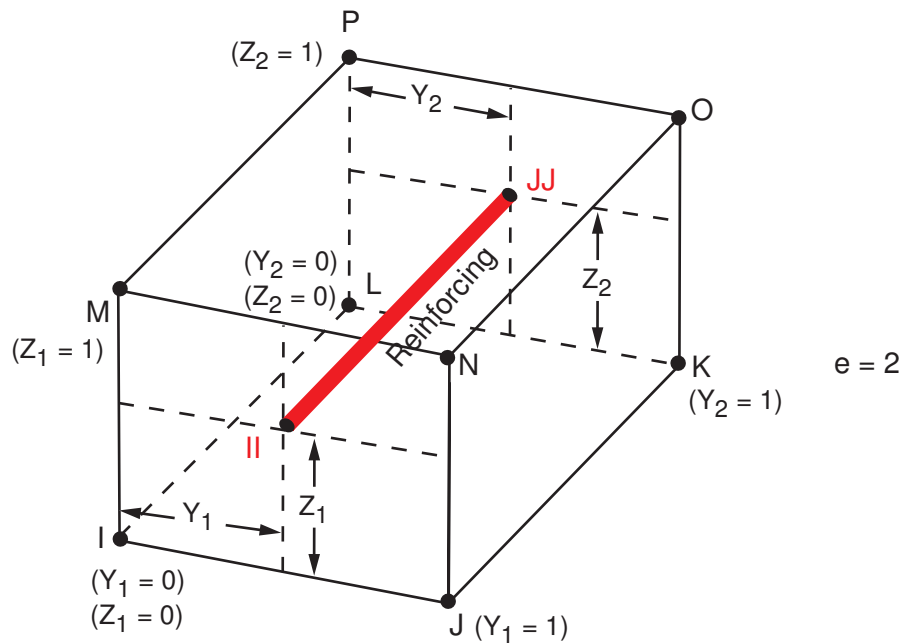
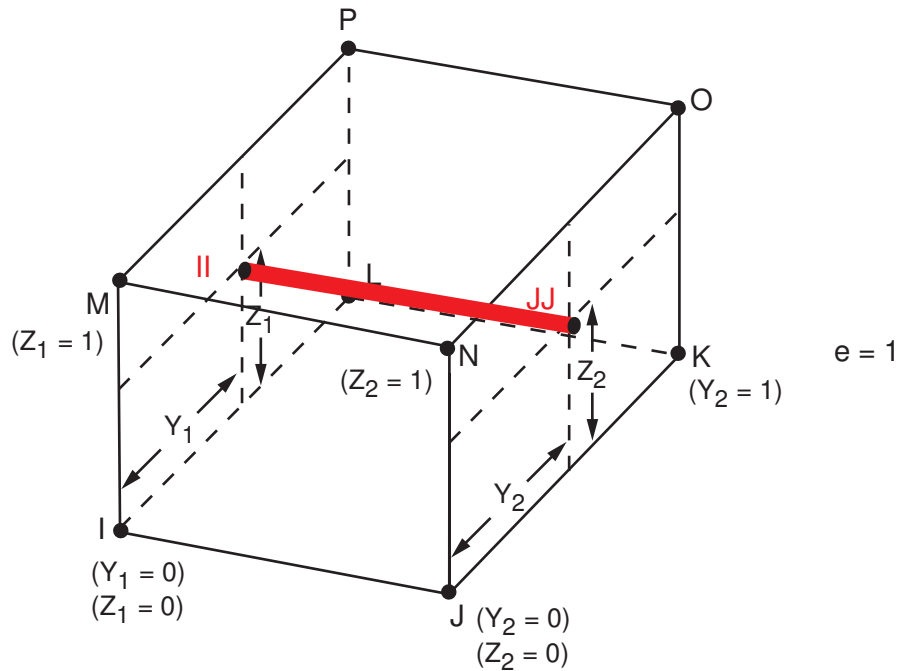
### **Required input:**

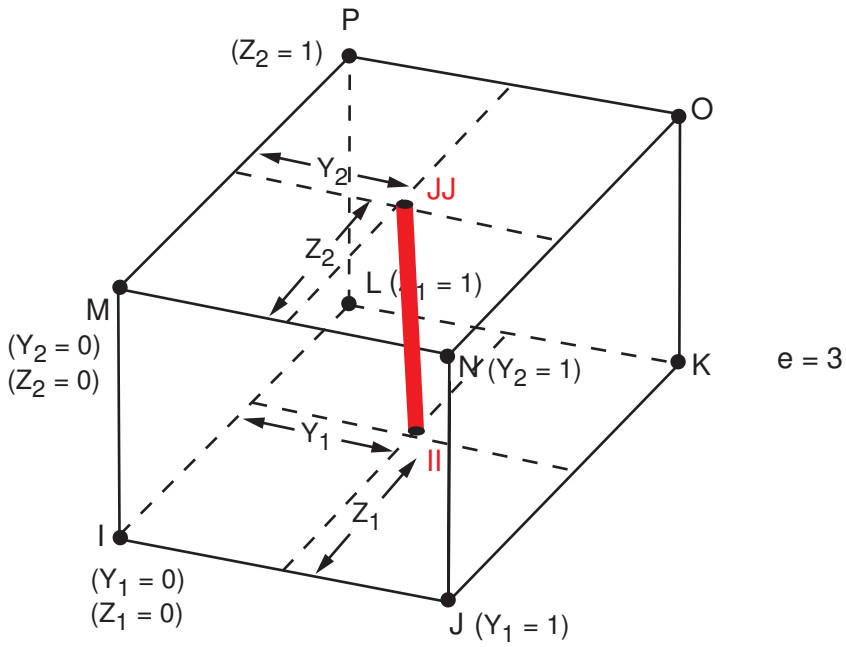
- V1 (or  $e$ ) -- The number to indicate the element edge to which the offsets are measured. The default value is 1.
- V2 and V3 (or  $Y1$  and  $Z1$ ) -- The normalized distances from the fiber to the first end of the specified element edge. Valid values for  $Y1$  and  $Z1$  are 0.0 through 1.0. The default value for  $Y1$  and  $Z1$  is 0.5.

$V4$  and  $V5$  (or  $Y2$  and  $Z2$ ) – The normalized distances from the fiber to the second end of the specified element edge. Value values for  $Y2$  and  $Z2$  are 0.0 through 1.0. The default value for  $Y2$  is  $Y1$ , and the default value for  $Z2$  is  $Z1$ .

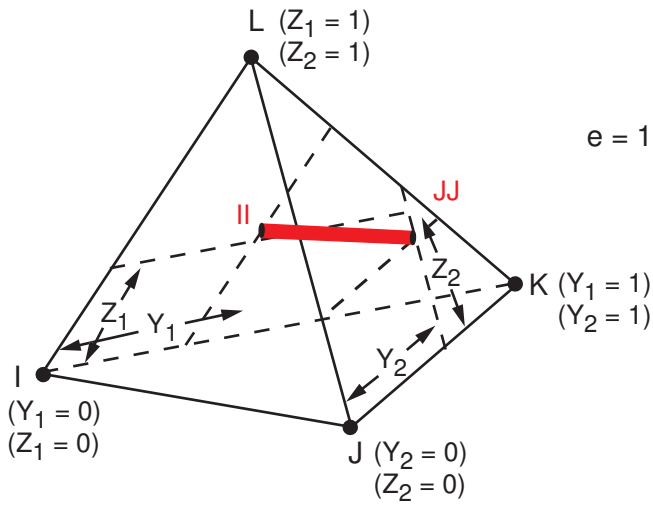
If the base element is a beam or link, ANSYS ignores values  $V2$  through  $V5$  and instead places the reinforcing in the center of the beam or link.

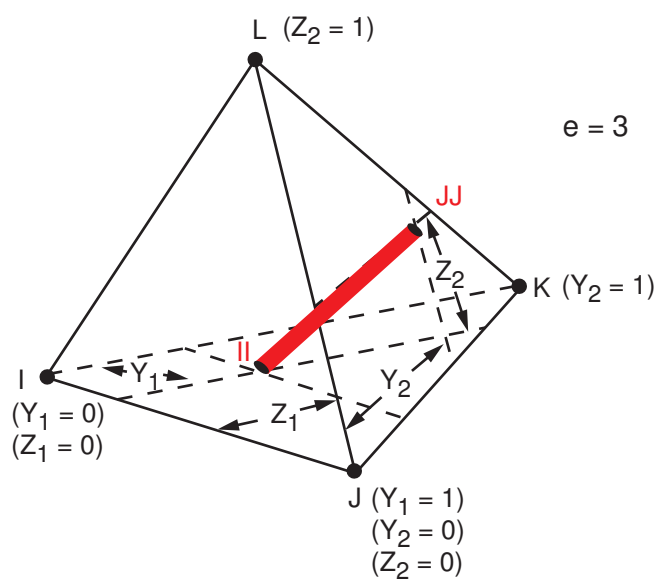
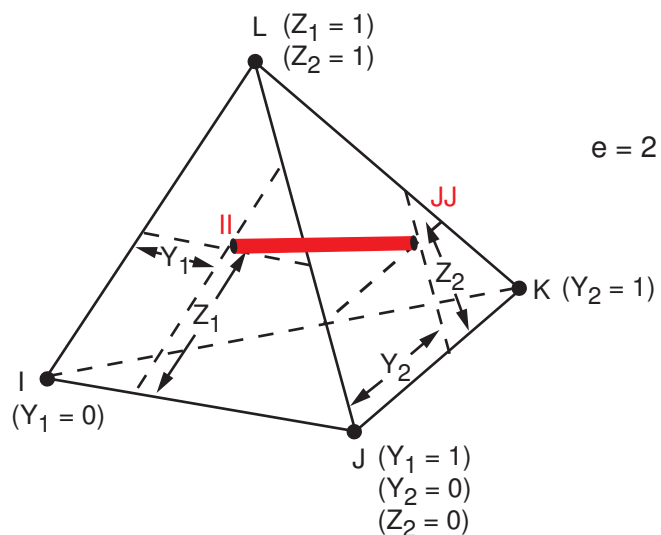
When applied to 8-node or 20-node solid elements:



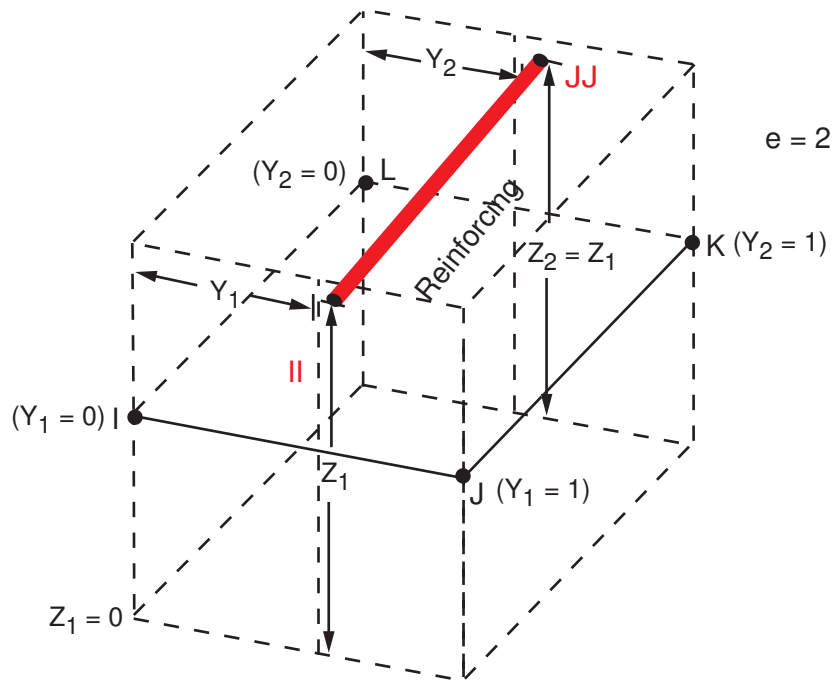
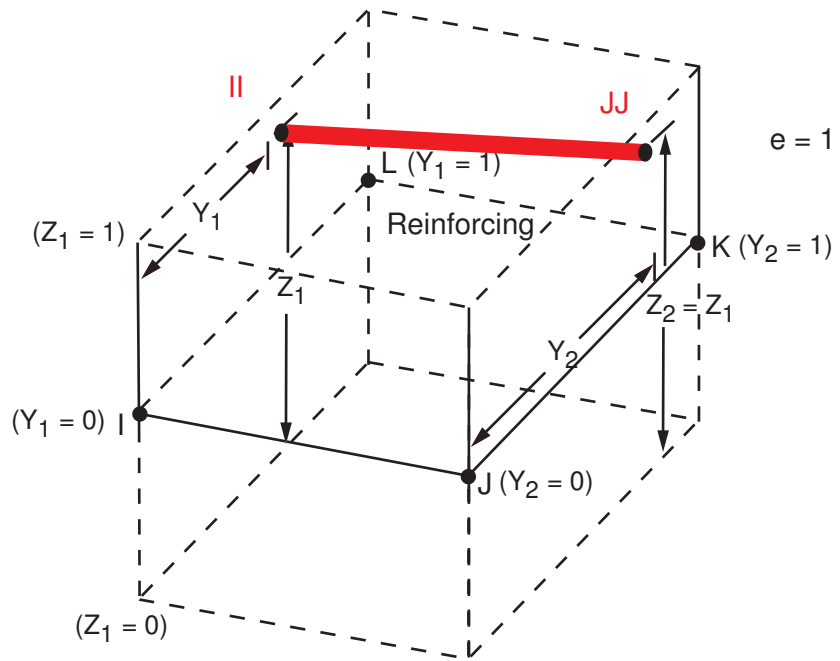


When applied to tetrahedral elements:

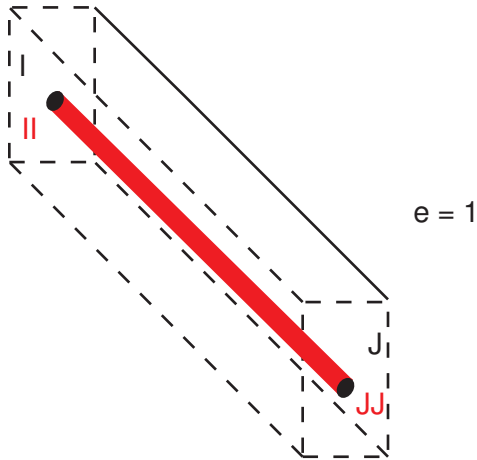




When applied to 3-D shell elements:



When applied to beam or link elements:



**PATT : BEAM**

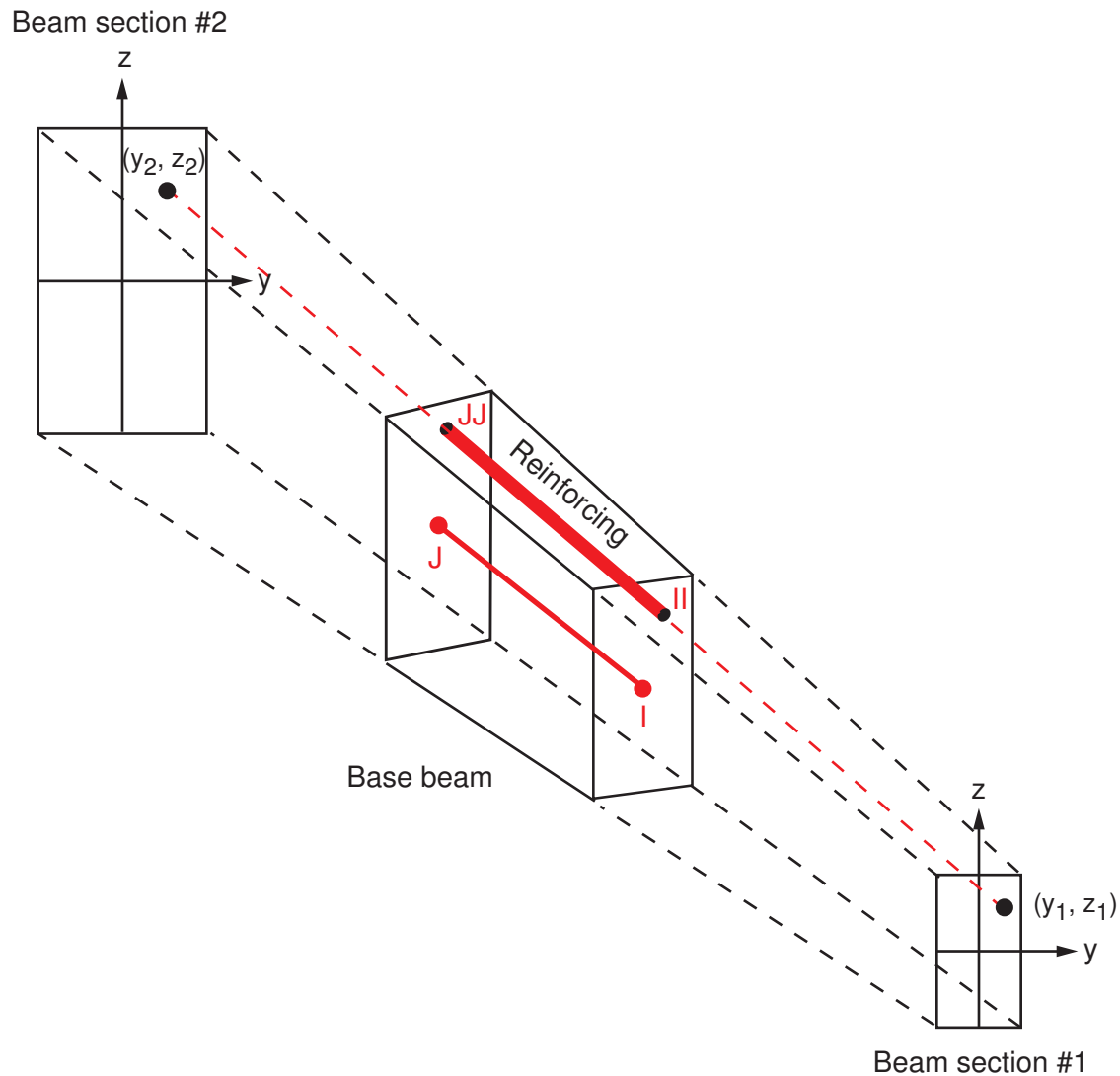
**Description:** Use this specialized input pattern for defining reinforcing in regular constant and tapered beams.

**Required input:**

$V1$  and  $V2$  (or  $Y1$  and  $Z1$ ) -- Y and Z offsets with respect to the section origin in the first beam section referred by the base beam element. The default value for  $Y1$  and  $Z1$  is 0.0.

$V3$  and  $V4$  (or  $Y2$  and  $Z2$ ) -- Y and Z offsets with respect to the section origin in the second beam section referred by the base beam element. The default value for  $Y2$  is  $Y1$ , and the default value for  $Z2$  is  $Z1$ .

(Because  $V3$  and  $V4$  values apply only to tapered beams, ANSYS ignores them if the base beam has a constant section.)



**Type: REINF, Subtype: SMEAR**

This subtype defines reinforcing fibers in homogenous layered forms. Issue separate **SECDATA** commands to define each of the layers of reinforcing fibers with uniform cross-section area and spacing.

Data to provide in the value fields:

*MAT*, *A*, *S*, *KCN*, *THETA*, *PATT*, *V1*, *V2*, *V3*, *V4*, *V5*

where

*MAT* = Material ID for layer. (Refer to the [REINF265](#) documentation for available material models.) The material ID is required for all layers, and no default for this value is available.

*A* = Cross-section area of a single reinforcing fiber.

*S* = Distance between two adjacent reinforcing fibers.

*KCN* = Local coordinate system reference number for this layer. (See the **LOCAL** command documentation for more information about local coordinate systems.) If this value is not provided, the layer adopts a default layer reference system. For the 2-D smeared reinforcing element [REINF263](#), *KCN* input is ignored and a default layer reference system is adopted.



*THETA* = Angle (in degrees) of the final layer coordinate system with respect to the default layer system or the layer system specified in the *KCN* field. This value is ignored for *REINF263* when that element is embedded in 2-D plane strain or plane stress base elements.

*PATT* = Input pattern code indicating how the location of this layer is defined with respect to the base element. Available input patterns are *LAYn*, *EDGo*, and *ELEf*.

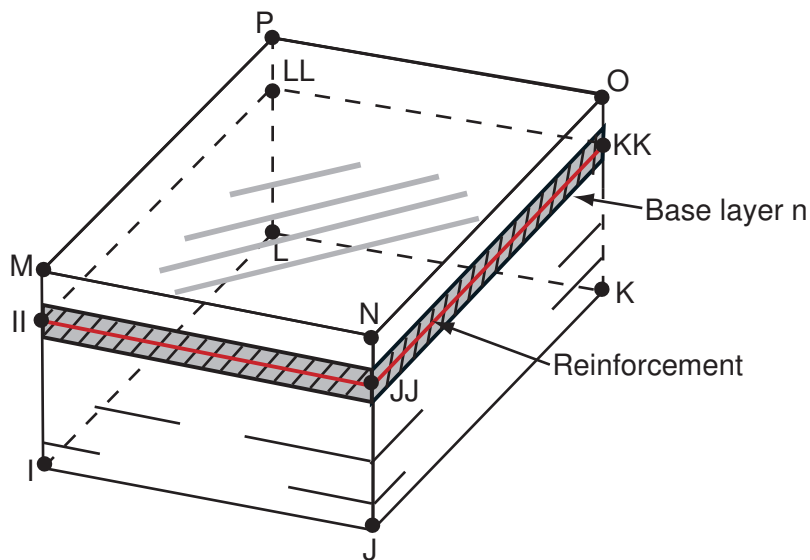
*V1*, *V2*, *V3*, *V4*, *V5* = Values to define the location of the reinforcing layer, as shown:

***PATT* : *LAYn***

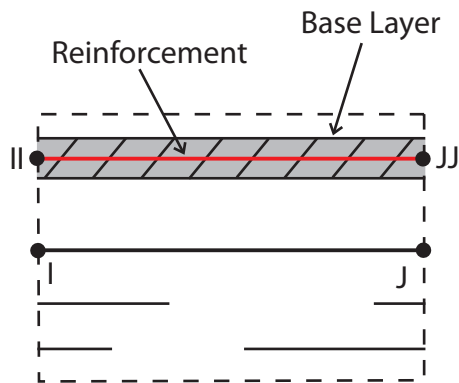
**Description:** The smeared reinforcing layer is placed in the middle of a layer in a layered base element.

**Required input:** *V1* (or *n*) -- The number of the layer in the base element on which to apply the reinforcing layer. The default value is 1.

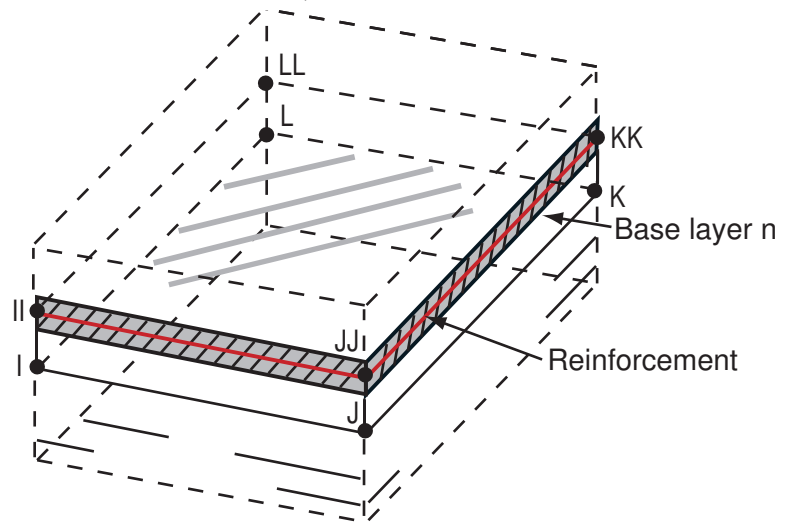
When applied to layered solid elements:



When applied to 2-D axisymmetric shells:



When applied to 3-D layered shells:



***PATT* : *EDGo***

**Description:** This pattern applies only to 2-D smeared reinforcing element [REINF263](#). The smeared reinforcing layer is represented by a line in 2-D. The orientation of the 2-D smeared reinforcing layer is similar to one of the specified element edges. The fiber orientation can be further adjusted via offsets with respect to the specified element edge.

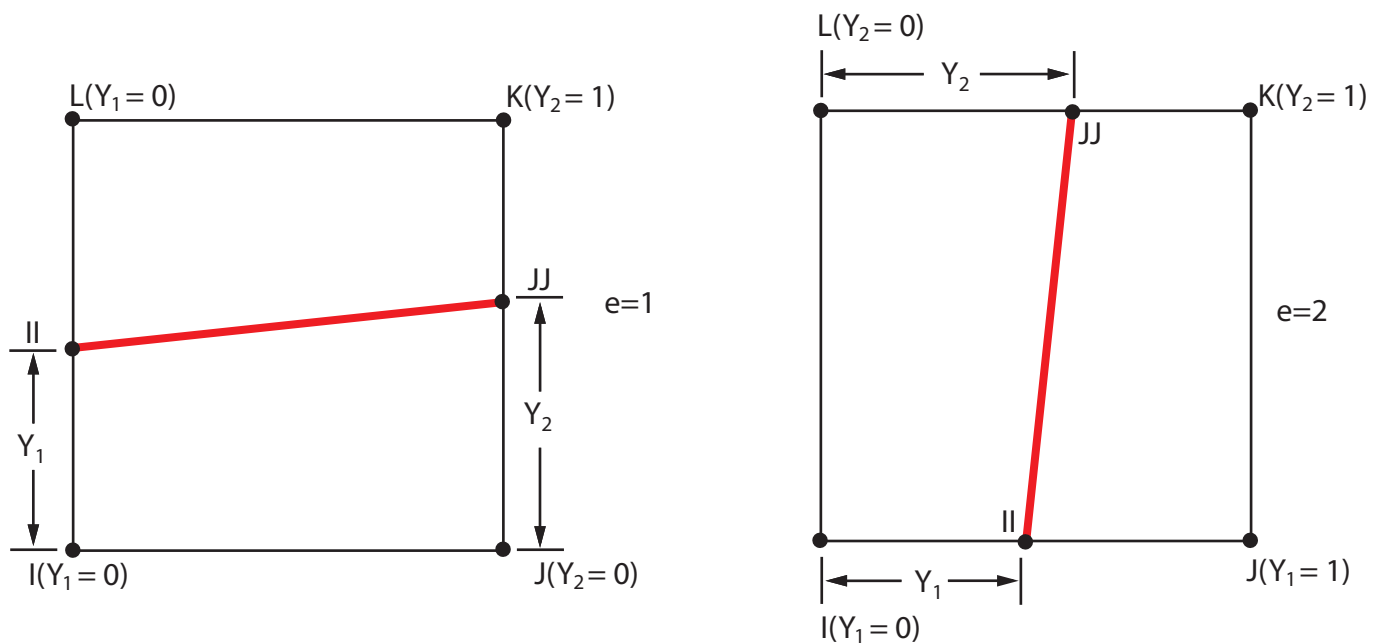
**Required input:**

$V1$  (or  $e$ ) -- The number to indicate the element edge to which the offsets are measured. The default value is 1.

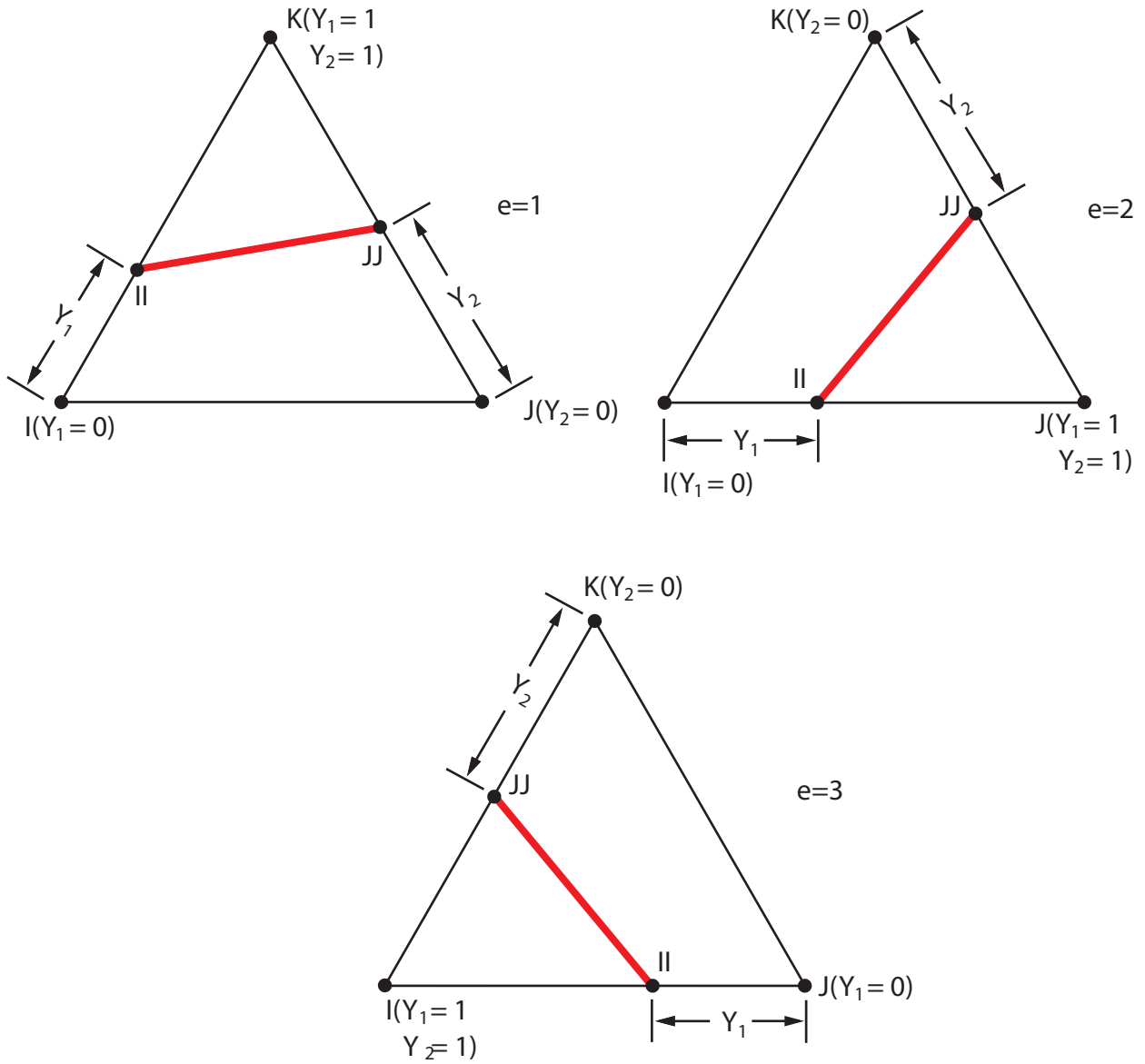
$V2$  (or  $Y1$ ) -- The normalized distances from the reinforcing layer to the first end of the specified element edge. Valid values for  $Y1$  are 0.0 through 1.0. The default value for  $Y1$  is 0.5.  $V3$  (or  $Z1$ ) input is ignored.

$V4$  (or  $Y2$ ) -- The normalized distances from the reinforcing layer to the second end of the specified element edge. Value values for  $Y2$  are 0.0 through 1.0. The default value for  $Y2$  is  $Y1$ .  $V5$  (or  $Z2$ ) input is ignored.

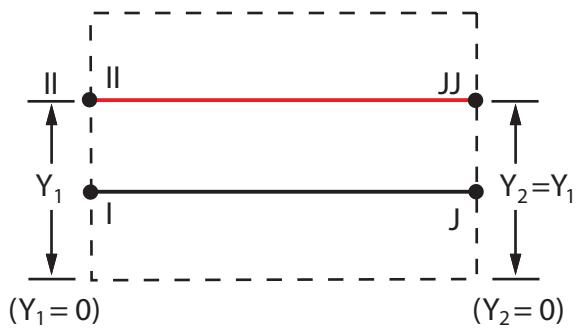
When applied to 2-D 4-node or 8-node solid elements:



When applied to 2-D 3-node or 6-node triangular solid elements:



When applied to 2-node or 3-node axisymmetric shell elements:



**PATT : ELEF**

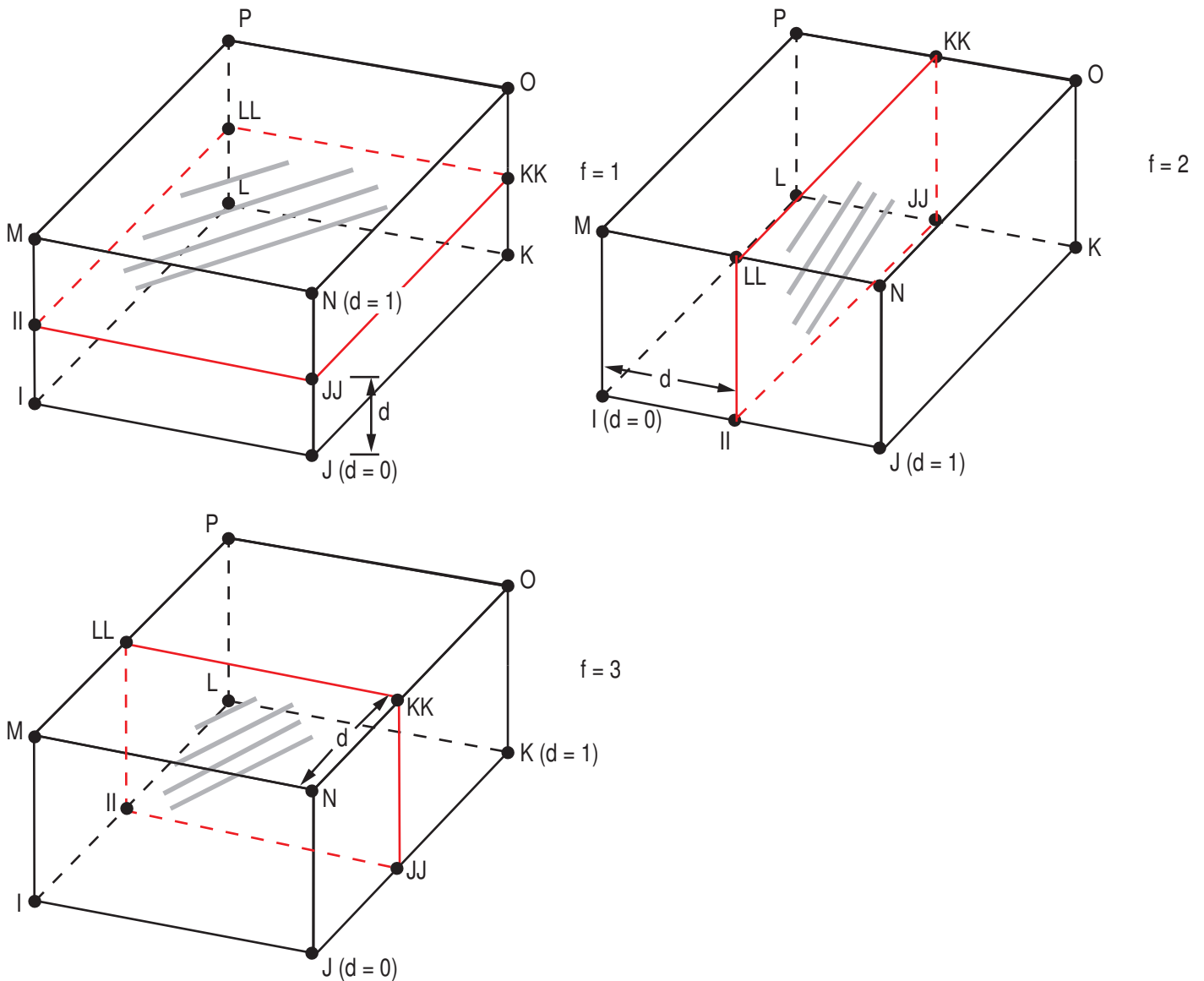
**Description:** The smeared reinforcing layer is oriented parallel to one of three adjacent element faces. (This pattern does not apply to 2-D smeared reinforcing element [REINF263](#).)

**Required input:**

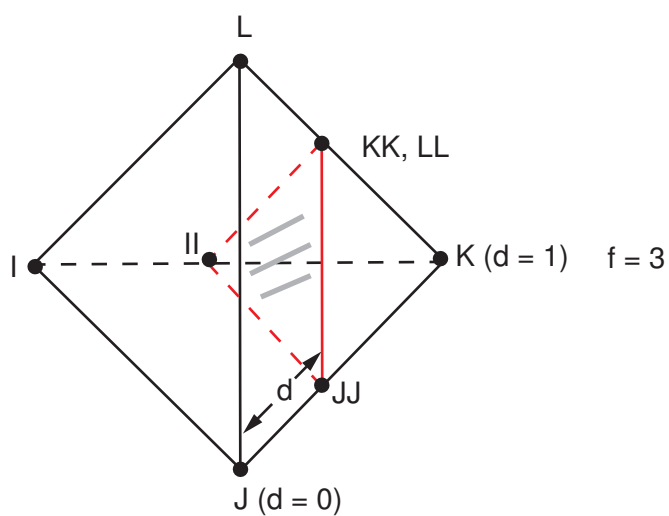
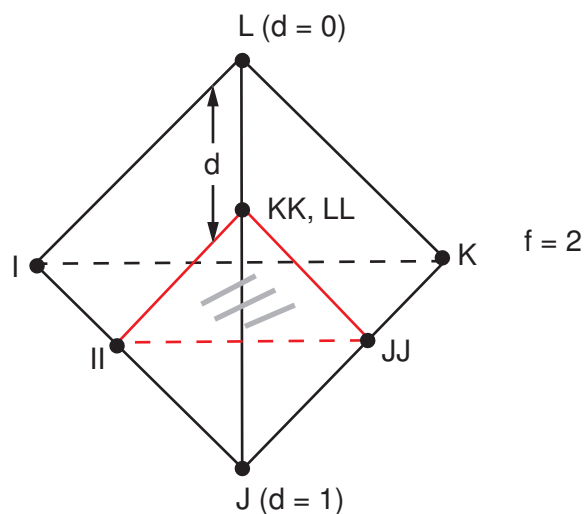
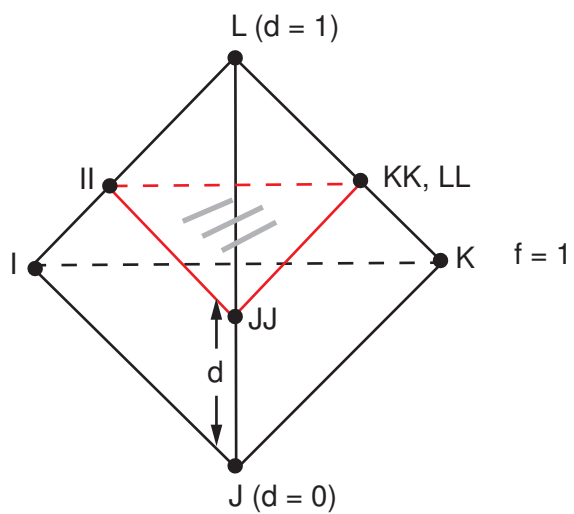
$V1$  (or  $f$ ) -- The number to indicate the base element face. The default value is 1.

$V2$  (or  $d$ ) -- The normalized distance from the layer to the specified base element face. Valid values for  $d$  are 0.0 through 1.0. The default value is 0.5.

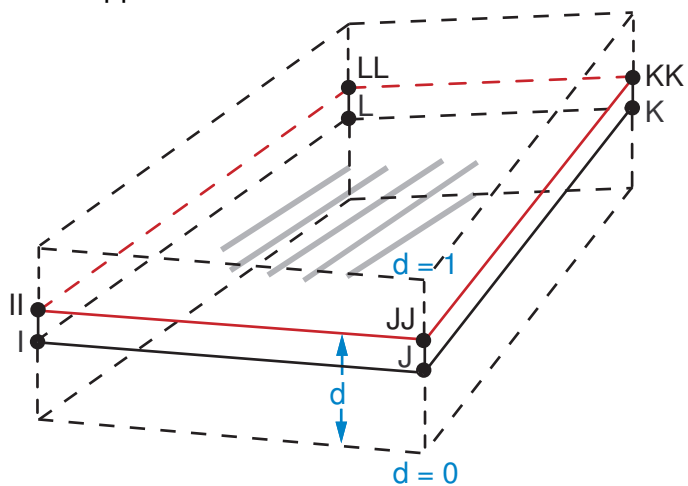
When applied to 8-node or 20-node solid elements:



When applied to tetrahedral elements:



When applied to 3-D shell elements:



## CONTACT

### Type: CONTACT

Contact sections are referenced by [CONTA173](#), [CONTA174](#), and [TARGE170](#) elements for geometry correction. This geometry correction is applicable to cases where the original meshes of contact elements or target elements are located on a portion of a spherical or revolute surface.

### Type: CONTACT, Subtype: SPHERE

Data to provide in the value fields for *Subtype* = SPHERE:  
*X0, Y0, Z0* (sphere center location in Global Cartesian coordinates)

### Type: CONTACT, Subtype: CYLINDER

Data to provide in the value fields for *Subtype* = CYLINDER:  
*X1, Y1, Z1, X2, Y2, Z2* (two ends of cylindrical axis in Global Cartesian coordinates)

---

## Menu Paths

- Main Menu>Preprocessor>Sections>Axis>Add
- Main Menu>Preprocessor>Sections>Axis>Edit
- Main Menu>Preprocessor>Sections>Beam>Common Sections
- Main Menu>Preprocessor>Sections>Beam>Taper Sections>By Picked Nodes
- Main Menu>Preprocessor>Sections>Beam>Taper Sections>By XYZ Location
- Main Menu>Preprocessor>Sections>Contact>Add
- Main Menu>Preprocessor>Sections>Contact>Edit
- Main Menu>Preprocessor>Sections>Joints>Add / Edit
- Main Menu>Preprocessor>Sections>Pipe>Add
- Main Menu>Preprocessor>Sections>Pipe>Edit
- Main Menu>Preprocessor>Sections>Reinforcing>Add / Edit
- Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit

---

## SECFUNCTION, *TABLE*, *KCN*

Specifies shell section thickness as a tabular function.

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *TABLE*

Table name or array parameter reference for specifying thickness.

### *KCN*

Local coordinate system reference number for this tabular function evaluation.

## Notes

The **SECFUNCTION** command is associated with the section most recently defined via the **SECTYPE** command.

A table (*TABLE*) can define tabular thickness as a function of coordinates. Alternatively, you can use an array parameter (indexed by node number) that expresses the function to be mapped. (For example, func (17) should be the desired shell thickness at node 17.) To specify a table, enclose the table or array name in percent signs (%) (**SECFUNCTION,%tablename%**). Use the **\*DIM** command to define a table.

The table or array defines the total shell thickness at any point in space. In multilayered sections, the total thickness and each layer thickness are scaled accordingly.

The [Function Tool](#) is a convenient way to define your thickness tables. For more information, see "[Using the Function Tool](#)" in the *Basic Analysis Guide*.

If you do not specify a local coordinate system (*KCN*), ANSYS interprets the table in global XYZ coordinates. When *KCN* = NODE, ANSYS interprets *TABLE* as an array parameter (indexed by node number) that expresses the function to be mapped. For information about local coordinate systems, see the documentation for the **LOCAL** command.

## Menu Paths

**Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit**

**SECJOINT**, *Kywrđ*, *Val1*, *Val2*, *Val3*, *Val4*, *Val5*, *Val6*

**Defines local coordinate systems at joint element nodes and other data for joint elements.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

### *Kywrđ*

Keyword that indicates the type of joint element data being defined.

### **LSYS** or blank

Define local coordinate systems at the nodes that form the [MPC184](#) joint element.

### **RDOF**

Define the relative degrees of freedom to be fixed for an [MPC184-General](#) joint element.

### **PITC**

Define the pitch of an [MPC184-Screw](#) joint element.

### **FRIC**

Define the geometric quantities required for Coulomb frictional behavior in the [MPC184-Revolute](#) or [MPC184-Translational](#) joint element.

### *Val1*, *Val2*, *Val3*, *Val4*, *Val5*, *Val6*

The meaning of *Val1* through *Val6* changes, depending on the value of *Kywrđ*.

If *Kywrđ* = LSYS (or blank), *Val1* and *Val2* are Identifiers of the local coordinate systems at nodes I and J, respectively, of the joint element. *Val3* through *Val6* are not used.

If *Kywrđ* = RDOF, *Val1* through *Val6* are the relative degrees of freedom to be fixed for a general joint element. Input 1 for UX, 2 for UY, 3 for UZ, 4 for ROTX, 5 for ROTY, and 6 for ROTZ. You may input the DOFs in any order.

If *Kywrđ* = PITC, *Val1* is the pitch of the screw joint element; pitch is defined as the ratio of relative axial displacement (length units) to relative rotation (in radians). *Val2* through *Val6* are not used.

If *Kywrđ* = FRIC, *Val1* through *Val3* are defined as follows.

**For Revolute Joint:**

*Val1* = outer radius

*Val2* = inner radius

*Val3* = effective length

**For Translational Joint:**

*Val1* = effective length

*Val2* = effective radius

## Notes

Use this command to define additional section data for MPC184 joint elements. To overwrite the current values, issue another **SECJOINT** command with the same *Kywrđ* value. The data input on this command is interpreted based on the most recently issued **SECTYPE** command.

## Menu Paths

Main Menu>Preprocessor>Sections>Joints>Add / Edit

---

**/SECLIB**, *Option*, *Path*

Sets the default section library path for the **SECREAD** command.

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Option*

**READ**

Sets the read path (default).

**STATUS**

Reports the current section library path setting to the Jobname.LOG file.

### *Path*

Defines the directory path from which to read section library files.

## Notes

When the **SECREAD** command is issued without a directory path, the command searches for a section library in the following order:

- The user's home directory
- The current working directory
- The path specified by the **/SECLIB** command

## Menu Paths

Main Menu>Preprocessor>Sections>Section Library>Library Path

---



**SECLOCK**, *dof*, *MINVALUE*, *MAXVALUE*, *dof*, *MINVALUE*, *MAXVALUE*, *dof*, *MINVALUE*, *MAXVALUE*

**Specifies locks on the components of relative motion in a joint element.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**dof**

Local degree of freedom to be locked.

**MINVALUE**

Low end of the range of allowed movement for the specified DOF.

**MAXVALUE**

High end of the range of allowed movement for the specified DOF.

## Notes

Specify up to three DOFs to be locked. Locks are activated when the limit values are reached, and further motion in that DOF is frozen. If necessary, you may repeat the command.

## Menu Paths

**Main Menu>Preprocessor>Sections>Joints>Add / Edit**

**SECMODIF**, *SECID*, *Kywrđ*

**Modifies a pretension section**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

If *Kywrđ* = NORM, command format is **SECMODIF**,*SECID*, NORM, *NX*, *NY*, *NZ*, *KCN*

**SECID**

Unique section number. This number must already be assigned to a section.

**NORM**

Keyword specifying that the command will modify the pretension section normal direction.

***NX*, *NY*, *NZ***

Specifies the individual normal components to modify.

***KCN***

Coordinate system number. This can be either 0 (Global Cartesian), 1 (Global Cylindrical) 2 (Global Spherical), 4 (Working Plane), 5 (Global Y Axis Cylindrical) or an arbitrary reference number assigned to a coordinate system.

If *Kywrđ* = NAME, the command format is **SECMODIF**,*SECID*, NAME, *Name*

**SECID**

Unique section number. This number must already be assigned to a section.

**NAME**

Action key that instructs the command to change the name of the specified pretension section.

**Name**

The new name to be assigned to the pretension section.

**Notes**

The **SECMODIF** command either modifies the normal for a specified pretension section, or changes the name of the specified pretension surface.

**Menu Paths**

**Main Menu>Preprocessor>Sections>Pretension>Modify Name**  
**Main Menu>Preprocessor>Sections>Pretension>Modify Normal**

---

**SECNUM**, *SECID*

Sets the element section attribute pointer.

[PREP7: Cross Sections](#)

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

**SECID**

Defines the section ID number to be assigned to the subsequently-defined elements. Defaults to 1. See [SECTYPE](#) for more information about the section ID number.

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes**

---

**SECOFFSET**, *Location, OFFSET1, OFFSET2, CG-Y, CG-Z, SH-Y, SH-Z*

Defines the section offset for cross sections.

[PREP7: Cross Sections](#)

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

*Location, OFFSET1, OFFSET2, CG-Y, CG-Z, SH-Y, SH-Z*

The location of the nodes in the section. All are dependent on the type. See the ["Notes"](#) (p. 1494) section of this command description for details about these values for the various section types.

**Notes**

The **SECOFFSET** command is divided into three types: [Beams](#), [Pipes](#), and [Shells](#).

The offsets defined by the **SECOFFSET** command are associated with the section most recently defined using the [SECTYPE](#) command. Not all **SECOFFSET** location values are valid for each subtype.

For the thermal shell elements, [SHELL131](#) and [SHELL132](#), the node offset specified by **SECOFFSET** is used in thermal contact analyses. Otherwise, the **SECOFFSET** command has no effect on the solution for these elements and is used only for visualization purposes.

This command is not valid with thermal solid elements [SOLID278](#) and [SOLID279](#).

## Beams

### Type: *BEAM*

Argument data to provide:

*Location, OFFSETY, OFFSETZ, CG-Y, CG-Z, SH-Y, SH-Z*

#### *Location*

##### **CENT** --

Beam node will be offset to centroid (default).

##### **SHRC** --

Beam node will be offset to shear center.

##### **ORIGIN** --

Beam node will be offset to origin of the cross section.

##### **USER** --

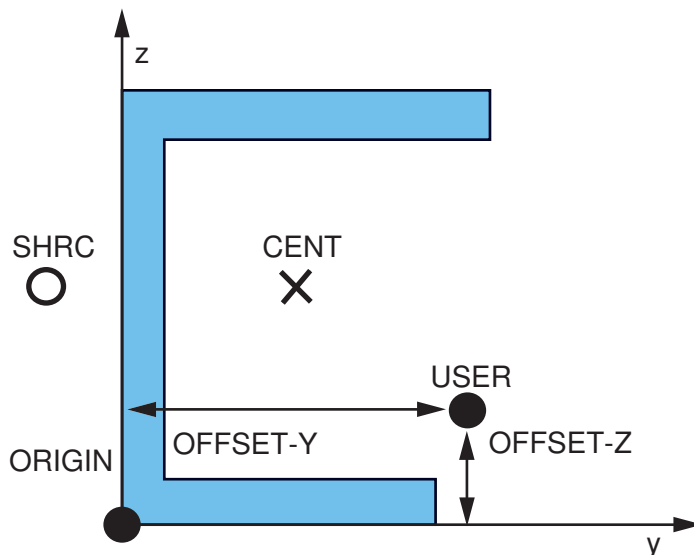
Beam node will be offset to the location specified by the *OFFSETY* and *OFFSETZ* arguments.

#### *OFFSETY, OFFSETZ*

Values that locate the node with respect to the default origin of the cross section when the *Location* argument is set to *USER*. Valid only when *USER* is set.

The following figure illustrates the offsets for a channel cross section, and shows the relative locations of *SHRC* and *CENT*.

**Figure 5 Offsets for a CHAN Section Subtype**



#### *CG-Y, CG-Z, SH-Y, SH-Z*

Override the program-calculated centroid and shear centroid locations.

**Note**

This option should only be used by *advanced users* modeling composite cross sections.

**Pipes****Type: PIPE**

Argument data to provide:

*OFFSETY, OFFSETZ*

***OFFSETY, OFFSETZ***

Values that locate the node with respect to the center of the pipe.

**Shells****Type: SHELL**

Argument data to provide:

*Location, OFFSET*

***Location***

**TOP --**

Shell node will be offset to top of the section.

**MID --**

Shell node will be offset to midplane of the section (default).

**BOT --**

Shell node will be offset to bottom of the section.

**USER --**

Shell node will be offset to the location specified by the *OFFSET* argument.

***OFFSET***

Value that locates the node with respect to the default origin (midplane) of the section. Valid only when *Location* = USER.

For shells, an offset alters only the *reference surface* of the shell elements (that is, how the program measures displacements: at the middle, top, or bottom of the physical shell). It does not change the physical dimensions of the shell itself; the volume and mass remain constant when an offset is specified.

**Menu Paths**

**Main Menu>Preprocessor>Sections>Beam>Common Sections**

**Main Menu>Preprocessor>Sections>Beam>Custom Sections>Read Sect Mesh**

**Main Menu>Preprocessor>Sections>Pipe>Add**

**Main Menu>Preprocessor>Sections>Pipe>Edit**

**Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit**

**SECQPLOT**, *SECID*, *VAL1*, *VAL2*, *VAL3*

**Plots the geometry of a beam, pipe, shell, or reinforcing section to scale.**

PREP7: Cross Sections

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**SECID**

The section ID number (as defined via the **SECTYPE** command).

**VAL1, VAL2, VAL3**

Values that control the information to be plotted. See the "Notes" (p. 1497) section of this command description for more information. For clarity, the labels *VAL1*, *VAL2*, and *VAL3* are renamed according to the section type.

**Notes**

The **SECQPLOT** command is valid only for "Beams and Pipes" (p. 1497), "Shells" (p. 1498), and "Reinforcings" (p. 1499). Custom sections created using FiberSIM data do not support **SECQPLOT**.

**SECQPLOT** cannot display the plot of an ASEC (arbitrary section) subtype.

**Beams and Pipes**

Plots the geometry of the beam or pipe section to scale depicting the centroid, shear center, and origin. **SECQPLOT** also lists various section properties such as  $I_{yy}$ ,  $I_{yz}$ , and  $I_{zz}$ .

Data to be supplied in the value fields:

**MESHKEY**

Beam or pipe section mesh display options:

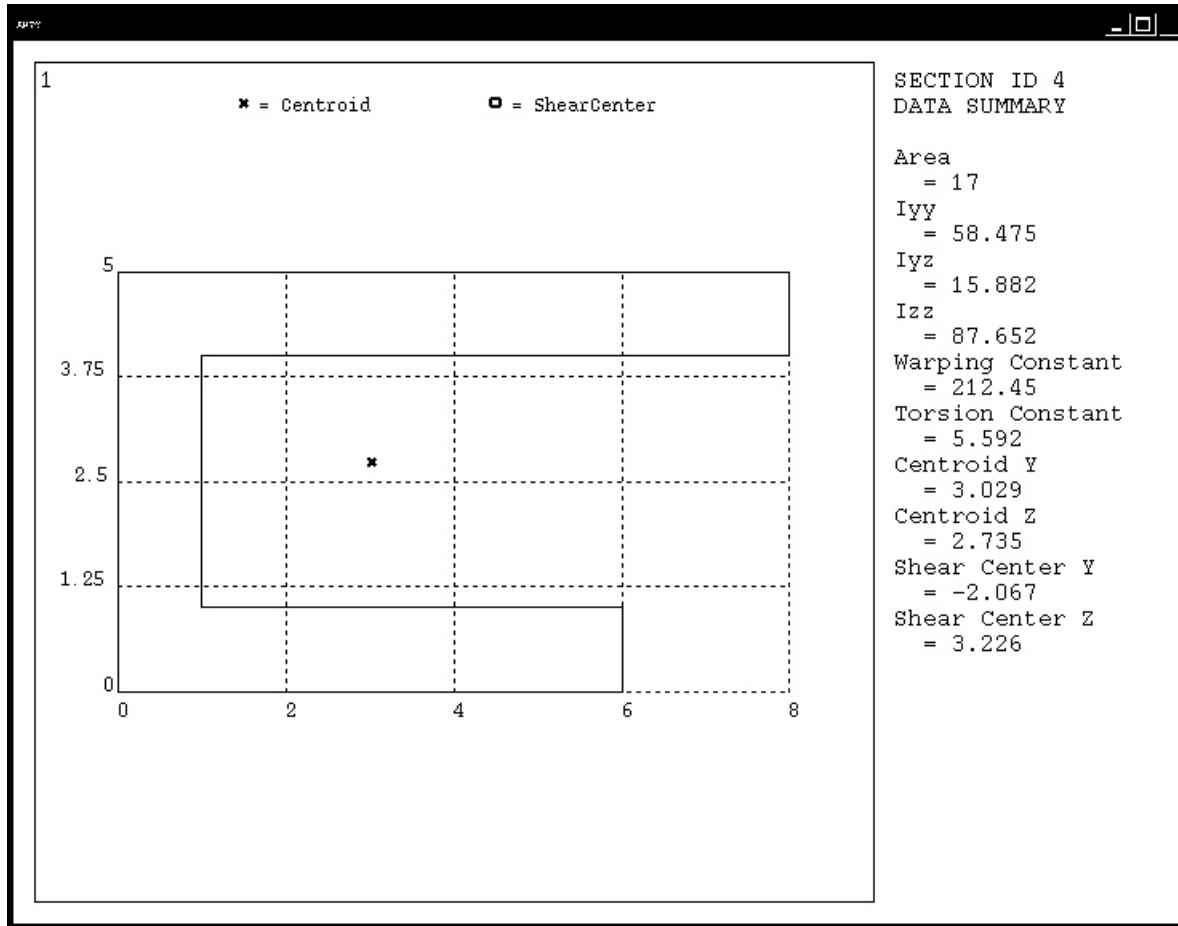
- 0** -- Display section outline only.
- 1** -- Display beam or pipe section mesh.
- 2** -- Display the section mesh with node numbers.
- 3** -- Display the section mesh with cell numbers.
- 4** -- Display the section mesh with material numbers and colors.
- 5** -- Display the section mesh with material colors only.
- 6** -- Display the section mesh with the RST node numbers. RST nodes are section corner nodes where results are available. This is applicable when the averaged results format (KEYOPT(15) = 0 for **BEAM188**, **BEAM189**, **PIPE288**, and **PIPE289**) is used.

**7 --**

Display the section mesh with the RST cell numbers. RST cells are section cells where results are available. This is applicable when the non-averaged results format (KEYOPT(15) = 1 for [BEAM188](#), [BEAM189](#), [PIPE288](#), and [PIPE289](#)) is used.

Options 2 through 6 do not depict centroid and shear center, nor do they list section properties.

Following is a sample section plot for the beam section type:



**Shells**

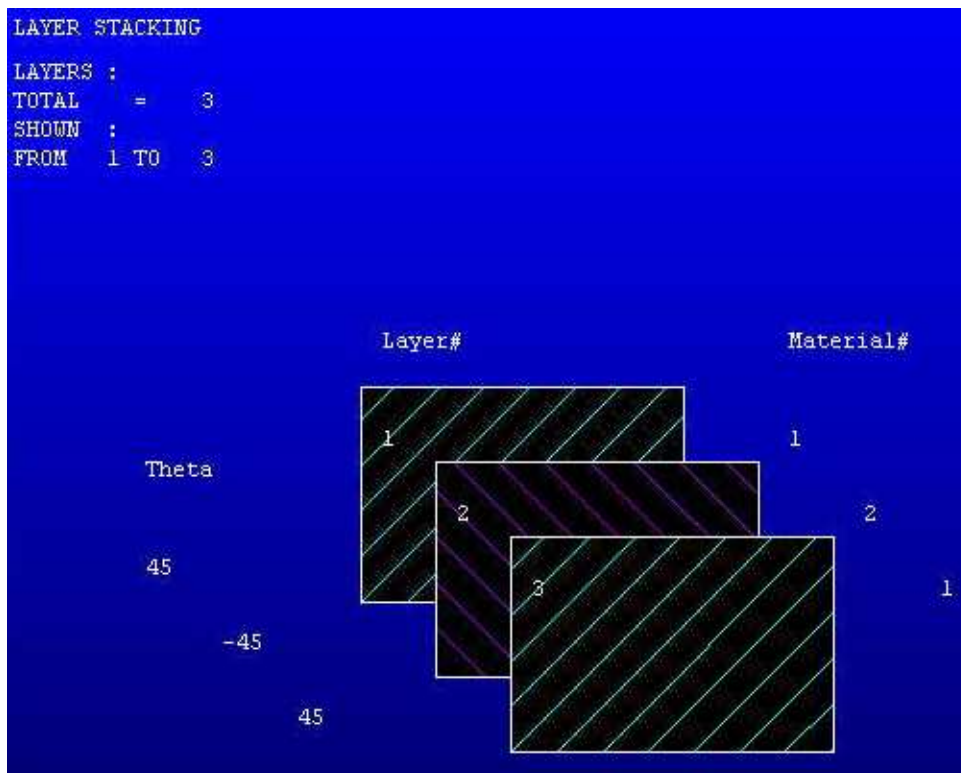
Plots the layer arrangement of the shell section showing the layer material and orientation.

Data to be supplied in the value fields:

**LAYR1, LAYR2**

The range of layer numbers to be displayed. If *LAYR1* is greater than *LAYR2*, a reversed order display is produced. Up to 20 layers may be displayed at the same time. *LAYR1* defaults to 1. *LAYR2* defaults to *LAYR1* if *LAYR1* is input or to the number of layers (or to 19+*LAYR1*, if smaller) if *LAYR1* is not input.

Following is a sample section plot for the shell section type:



## Reinforcings

Plots the arrangement of a reinforcing section within the base element.

Data to be supplied in the value fields:

### **REINF1, REINF2, OVERLAY**

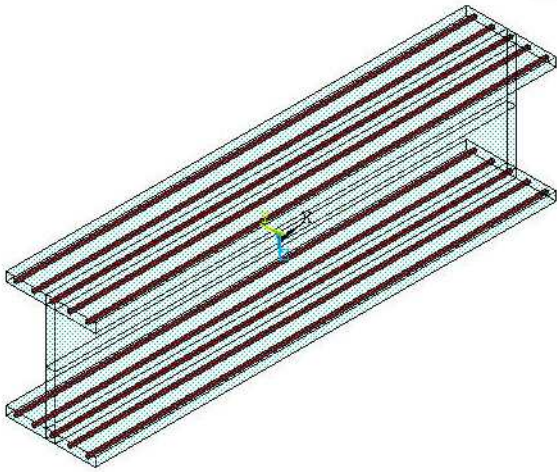
*REINF1, REINF2* -- The numerical range of reinforcings to be displayed. The default *REINF1* value is 1. The default *REINF2* value is the number of reinforcings.

*OVERLAY* -- The section ID of the base element within which to display the reinforcing section. The section appears translucent and the reinforcing section is solid. Valid values are:

- *SOLID* -- Display a translucent solid block over the reinforcing section
- *SECID* -- A number corresponding to a specific section ID of the base element.

If no *OVERLAY* value is specified, ANSYS displays the reinforcing section only.

Following is a sample section plot for the reinforcing section type:



For more information about reinforcing, see the documentation for the **SECDATA** command, and the **REINF264** and **REINF265** elements.

## Menu Paths

**Main Menu>Preprocessor>Sections>Beam>Plot Section**

**Main Menu>Preprocessor>Sections>Pipe>Plot Section**

**Main Menu>Preprocessor>Sections>Reinforcing>Plot Section**

**Main Menu>Preprocessor>Sections>Shell>Lay-up>Plot Section**

---

**SECRETAD**, *Fname*, *Ext*, --, *Option*

**Reads a custom section library or a user-defined section mesh into ANSYS.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### ***Fname***

Section library file name and directory path containing the section library file (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

When the **SECRETAD** command is given without a directory path, the command searches for a section library in the following order:

- The user's home directory
- The current working directory
- The path specified by the **/SECLIB** command

The file name defaults to `Jobname` if *Fname* is left blank.

### ***Ext***

Filename extension (8 character maximum).

The extension defaults to `SECT` if *Ext* is left blank.



--

Unused field.

**Option****LIBRARY**

Reads in a library of sections and their associated section data values; the default. A section library may be created by editing the section-defining portions of the `Jobname.LOG` file and saving it with a `.SECT` suffix.

**MESH**

Reads in a user mesh section file containing the cell connectivity, cell flags, and nodal coordinates for the current beam section of subtype MESH as defined by **SECTYPE**. See the "Notes" (p. 1501) section of this command description for details about user mesh section files. **SECWRITE** builds mesh files based on 2-D models you create.

**Notes**

The **SECRETAD** command operates on the section specified via the most recently issued **SECTYPE** command. Issue a separate **SECRETAD** command for each section ID that you want to read into ANSYS.

For shell sections incorporating **FiberSIM** data, the `.xml` file containing the **FiberSIM** data must exist in its original format. ANSYS cannot use the data if you have altered the file in any way (for example, by opening the file in an XML editor, making changes, and then saving it again).

**Sample User Section Cell Mesh File**

Here are excerpts from a sample user section mesh file for a section with 75 nodes, 13 cells, and 9 nodes per cell for a two-hole box section. Illustrations of the two-hole box section and the cell mesh for it appear later in this command description.

```

First Line:      75  13

Cells Section:
                   1  3  25  23  2  14  24  12  13  1
                   3  5  27  25  4  16  26  14  15  1
                   5  7  29  27  6  18  28  16  17  1
                   7  9  31  29  8  20  30  18  19  1
                   9 11  33  31 10  22  32  20  21  1
                   23 25  45  43 24  36  44  34  35  1
                   ...
                   49 51  73  71 50  62  72  60  61  1
                   51 53  75  73 52  64  74  62  63  1
                   0   0.0           0.0
                   0   0.025         0.0
                   0   0.05          0.0
                   0   5.0175        0.0

Nodes Section:
                   ...
                   0   19.98          10.00
                   0   20.00          10.00

```

The mesh file is divided into three sections: the First Line, the Cells Section, and the Nodes Section. Here are brief descriptions of the contents of each.

**First Line:** The First Line defines the number of nodes and the number of cells for the mesh.

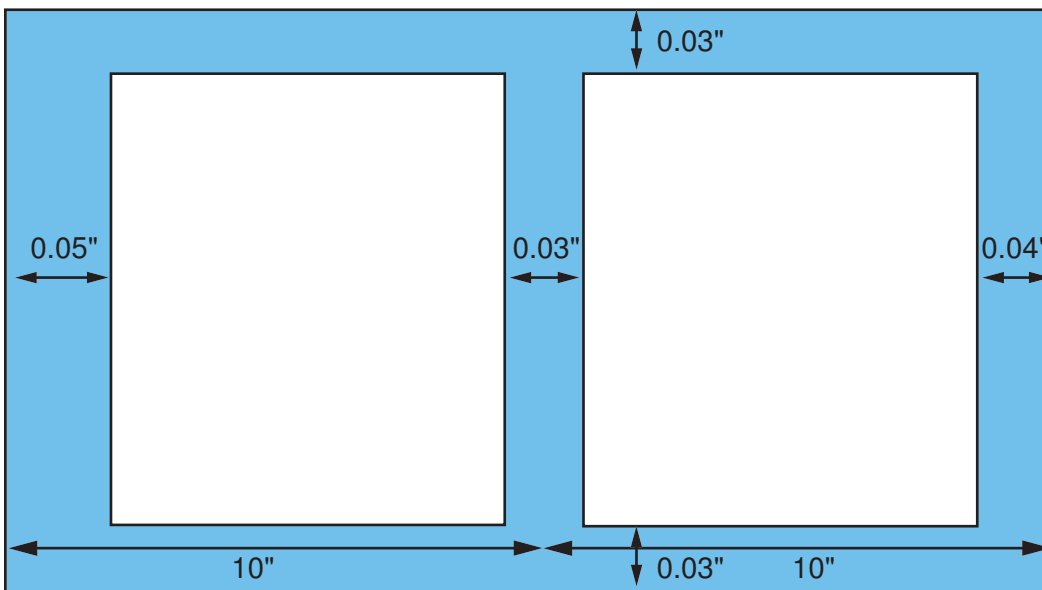
**Cells Section:** The Cells Section contains as many lines as there are cells. In this example, there are thirteen cells, so there are thirteen lines in this section. In each line, the number "1" that follows the cell connectivity information is the material number.

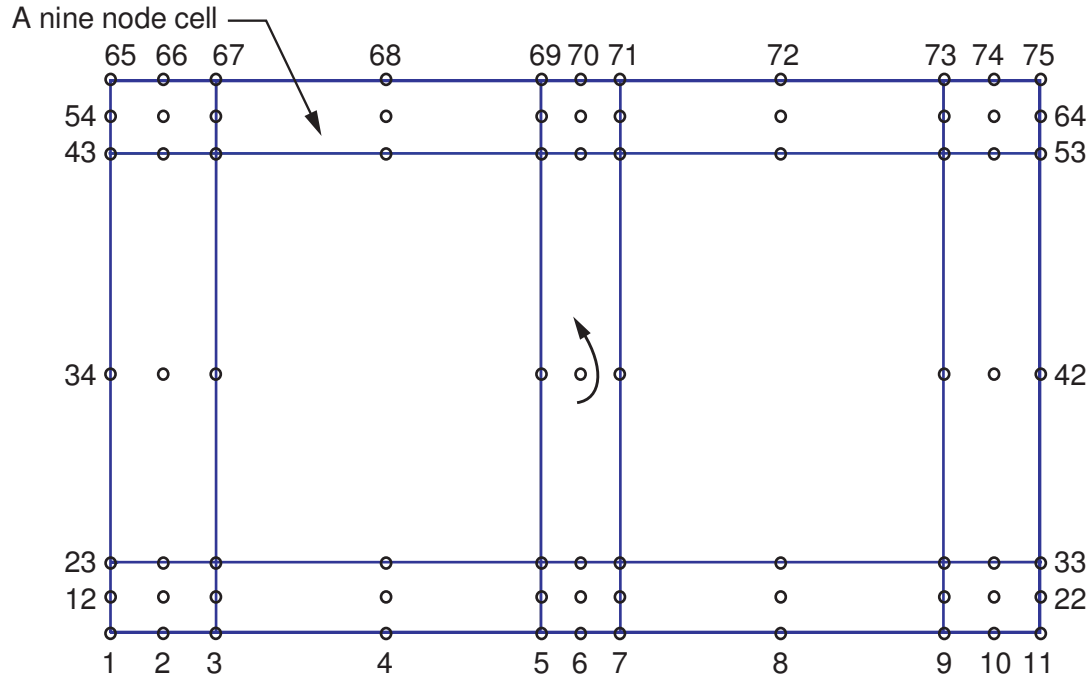
Cell nodal connectivity must be given in a counterclockwise direction, with the center node being the ninth node. For details, see *Figure 7* (p. 1503).

**Nodes Section:** The Nodes Section contains as many lines as there are nodes. In this example, there are 75 nodes, so there are a total of 75 lines in this section. Each node line contains the node's boundary flag, the Y coordinate of the node, and the Z coordinate of the node. Currently, all node boundary flags appear as 0s in a cell mesh file (as illustrated in *Figure 6* (p. 1502)). Since all node boundary flags are 0, **SECREAD** ignores them when it reads a cell mesh file into ANSYS.

There cannot be any gaps in the node numbering of a cell mesh. The nodes in a cell mesh must be numbered consecutively, with the first node having a node number of 1, and the last node having a node number that is equal to the maximum number of nodes in the cell mesh.

**Figure 6 Two-hole Box Section**



**Figure 7 Cell Mesh for the Two-hole Box Section**

## Menu Paths

**Main Menu>Preprocessor>Sections>Beam>Custom Sections>Read Sect Mesh**  
**Main Menu>Preprocessor>Sections>Section Library>Import Library**

**SECSTOP**, *dof*, *MINVALUE*, *MAXVALUE*, *dof*, *MINVALUE*, *MAXVALUE*, *dof*, *MINVALUE*, *MAXVALUE*

**Specifies stops on the components of relative motion in a joint element.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**dof**

Local degree of freedom to be stopped.

**MINVALUE**

Low end of the range of allowed movement for the specified DOF.

**MAXVALUE**

High end of the range of allowed movement for the specified DOF.

## Notes

Stops restrict motion in a DOF; motion beyond the MINVALUE or MAXVALUE is prevented (motion away from a limit is allowed). You can specify up to three stops. If necessary, you can repeat the command.

## Menu Paths

**Main Menu>Preprocessor>Sections>Joints>Add / Edit**

**SECTYPE**, *SECID*, *Type*, *Subtype*, *Name*, *REFINEKEY*

**Associates section type information with a section ID number.**

PREP7: Cross Sections

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

**SECID**

Section identification number.

**Type****BEAM**

Defines a beam section.

**TAPER**

Defines a tapered beam section. The sections at the end points must be topologically identical.

**GENB**

Defines a nonlinear general (temperature-dependent) beam section.

**COMB**

Defines a composite (temperature-dependent) beam section.

**PIPE**

Defines a pipe section.

**AXIS**

Define the axis for a general axisymmetric section.

**SHELL**

Defines a shell section.

**GENS**

Defines a preintegrated general (temperature-dependent) shell section.

**PRETENSION**

Defines a pretension section.

**JOINT**

Defines a joint section.

**REINF**

Defines a reinforcing section.

**CONTACT**

Defines a contact section.

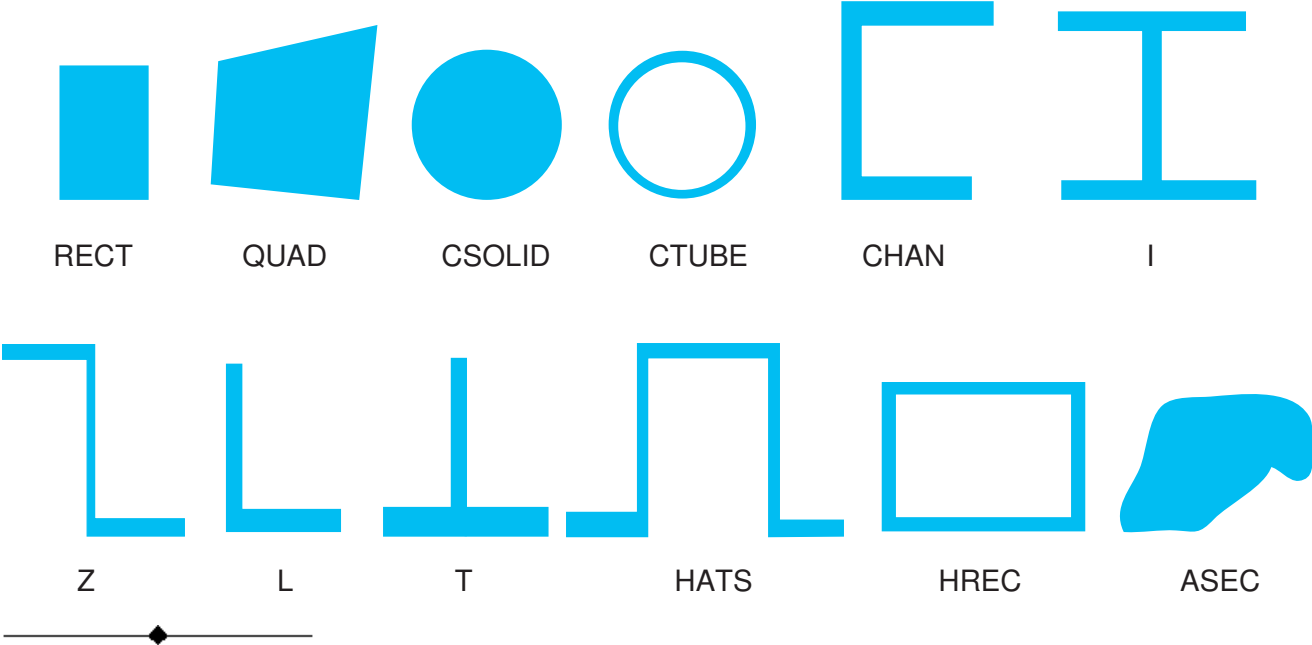
**Subtype**

When *Type* = BEAM, the possible beam sections that can be defined for *Subtype* are:

RECT	Rectangle
QUAD	Quadrilateral
CSOLID	Circular solid
CTUBE	Circular tube
CHAN	Channel
I	I-shaped section
Z	Z-shaped section

L	L-shaped section
T	T-shaped section
HATS	Hat-shaped section
HREC	Hollow rectangle or box
ASEC	Arbitrary section -- integrated cross-section inertia properties supplied by user
MESH	User-defined mesh -- see the <b>SECREAD</b> command for more information about this data

The following figure shows the shape of each cross section subtype:



When *Type* = GENB, the possible **nonlinear general beam sections** that can be defined for *Subtype* are:

ELASTIC	The generalized-stress/generalized-strain relationship is elastic (linear or nonlinear).
PLASTIC	The generalized-stress/generalized-strain relationship is elasto-plastic (and allows for permanent deformation).

When *Type* = COMB, the only possible composite-beam section that can be defined for *Subtype* is:

MATRIX	Matrix.
--------	---------

When *Type* = SHELL, an optional, special-purpose *Subtype* is available:

FIBERSIM	Specifies that layer data from the <b>FiberSIM</b> product will be used to generate the section.
----------	--

When *Type* = JOINT, the possible joint sections that can be defined for *Subtype* are:

UNIV	Universal joint
REVO	Revolute joint
SLOT	3-D Slot joint
PINP	Point-in-plane joint
PRIS	Translational joint
CYLI	Cylindrical joint
PLAN	Planar joint
WELD	Weld joint
ORIE	Orient joint
SPHE	Spherical joint
GENE	General joint
SCRE	Screw joint
BUSH	Bushing joint

When *Type* = REINF, the possible reinforcing sections that can be defined for *Subtype* are:

DISC	Discrete reinforcing. The reinforcing fibers are arbitrarily oriented and modeled individually.
SMEAR	Smearred reinforcing. The reinforcing fibers are homogenous and defined as a membrane.

When *Type* = CONTACT, the possible contact sections that can be defined for *Subtype* are:

SPHERE	Geometry correction for a portion of a spherical (or nearly spherical) surface.
CYLINDER	Geometry correction for a portion of a revolute (or nearly revolute) surface.

#### **Name**

An eight-character name for the section. *Name* can be a string such as "W36X210" or "HP13X73" for beam sections. Section name can consist of letters and numbers, but cannot contain punctuation, special characters, or spaces.

#### **REFINEKEY**

Sets mesh refinement level for thin-walled beam sections. Valid values are 0 (the default - no mesh refinement) through 5 (high level of mesh refinement). This value has meaning only when *Type* = BEAM.

## Notes

**SECTYPE** sets the section ID number, section type, and subtype for a section. If the section ID number is not specified, ANSYS increments the highest section ID number currently defined in the database by one. A previously-defined section with the same identification number will be redefined. The geometry data describing this section type is defined by a subsequent **SECDATA** command. Define the offsets by a subsequent **SECOFFSET** command. The **SLIST** command lists the section properties, and the **SECPLLOT** command displays the section to scale. The **SECNUM** command assigns the section ID number to a subsequently-defined beam element.

### Beam Section Considerations

For a beam section (*Type* = BEAM), a subsequent **SECDATA** command builds a numeric model using a nine-node cell for determining the properties (*Ixx*, *Iyy*, etc.) of the section and for the solution to the Poisson's equation for torsional behavior. See [Beam Analysis and Cross Sections](#) in the *Structural Analysis Guide* for examples using the section commands.

For a tapered beam section (*Type* = TAPER), two subsequent **SECDATA** commands are required (one for each end section). Section ends must be topologically identical (same *Subtype*, number of cells and material IDs).

For a nonlinear general beam section (*Type* = GENB), the *Subtype* and *REFINEKEY* options do not apply. Subsequent commands are necessary to define the section: **BSAX**, **BSM1**, **BSM2**, **BSTQ**, **BSS1**, **BSS2**, **BSMD**, and **BSTE** are available. All other section commands are ignored for this section type.

For a preintegrated composite-beam section (*Type* = COMB), the *REFINEKEY* options do not apply. Subsequent commands are necessary to define the section: **CBTMP**, **CBMX**, **CBMD**, and **CBTE** are available. All other section commands are ignored for this section type.

### Shell Section Considerations

For a preintegrated general shell section (*Type* = GENS), the *Subtype* and *REFINEKEY* options do not apply. Subsequent commands are necessary to define the section: **SSPA**, **SSPB**, **SSPD**, **SSPE**, **SSMT**, **SSBT**, and **SSPM** are available. All other section commands are ignored for this section type.

### Pretension Section Considerations

The PRETENSION section options of the **SECTYPE** and **SECDATA** commands are documented primarily to aid your understanding of the data written by the **CDWRITE** command. ANSYS, Inc. recommends that you generate pretension sections via the **PSMESH** command.

### Displaying Elements with Section Definition

To display elements with shapes determined from the section definition, issue the **/ESHAPE** command.

## Menu Paths

**Main Menu>Preprocessor>Sections>Axis>Add**

**Main Menu>Preprocessor>Sections>Axis>Edit**

**Main Menu>Preprocessor>Sections>Beam>Common Sections**

**Main Menu>Preprocessor>Sections>Beam>Composite Sections**

**Main Menu>Preprocessor>Sections>Beam>Custom Sections>Read Sect Mesh**

**Main Menu>Preprocessor>Sections>Beam>NL Generalized**

Main Menu>Preprocessor>Sections>Beam>Taper Sections>By Picked Nodes  
Main Menu>Preprocessor>Sections>Beam>Taper Sections>By XYZ Location  
Main Menu>Preprocessor>Sections>Contact>Add  
Main Menu>Preprocessor>Sections>Contact>Edit  
Main Menu>Preprocessor>Sections>Joints>Add / Edit  
Main Menu>Preprocessor>Sections>Pipe>Add  
Main Menu>Preprocessor>Sections>Pipe>Edit  
Main Menu>Preprocessor>Sections>Reinforcing>Add / Edit  
Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit  
Main Menu>Preprocessor>Sections>Shell>Pre-integrated

---

**SECWRITE**, *Fname*, *Ext*, --, *ELEM\_TYPE*

Creates an ASCII file containing user mesh section information.

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname` if *Fname* is left blank.

***Ext***

Filename extension (8 character maximum).

The extension defaults to `SECT` if *Ext* is left blank.

--

Unused field.

***ELEM\_TYPE***

Element type attribute pointer (**ET**) for the elements that are part of the section. See **SECREAD** for a detailed description.

## Notes

Before creating a user mesh file, you must create a model using 2-D meshing. Use [PLANE183](#) or [MESH200](#) with `KEYOPT(1) = 7` (quadrilateral with 8 nodes option) to model the cells. **SECWRITE** creates an ASCII file that contains information about the nodes and cells that describe a beam section. For detailed information on how to create a user mesh file, see [Creating Custom Cross Sections with a User-defined Mesh](#) in the *Structural Analysis Guide*.

## Menu Paths

Main Menu>Preprocessor>Sections>Beam>Custom Sections>Write From Areas

---



**SED**, *SEDX*, *SEDY*, *SEDZ*, *Cname*

**Defines the excitation direction for response spectrum and PSD analyses.**

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***SEDX*, *SEDY*, *SEDZ***

Global Cartesian coordinates of a point that defines a line (through the origin) corresponding to the excitation direction. For example: 0.0, 1.0, 0.0 defines global Y as the spectrum direction.

***Cname***

The component name corresponding to the group of excited nodes. Only applies to base excitation multi-point response spectrum analysis (**SPOPT**, MPRS) and power spectral density analysis (**SPOPT**, PSD). Defaults to no component.

## Notes

In single-point response spectrum analysis (**SPOPT**, SPRS), the excitation direction without rocking (**ROCK**) is normalized to one so that the *SEDX*, *SEDY*, and *SEDZ* values do not scale the spectrum. The excitation direction with rocking is not normalized. The *SEDX*, *SEDY*, and *SEDZ* values must be consistent with the *OMX*, *OMY*, and *OMZ* values on the **ROCK** command. The calculated direction then scales the spectrum. For more information, see [Participation Factors and Mode Coefficients](#).

In multi-point response spectrum analysis (**SPOPT**, MPRS) and power spectral density analysis (**SPOPT**, PSD), the excitation direction is normalized to one so that the *SEDX*, *SEDY*, and *SEDZ* values do not scale the spectrum. The component name (*Cname*) is required. The constraints corresponding to the excitation direction are applied to the component nodes.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>DDAM Options**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Settings**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Settings**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Settings**  
**Main Menu>Solution>Load Step Opts>Spectrum>DDAM Options**  
**Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Settings**  
**Main Menu>Solution>Load Step Opts>Spectrum>PSD>Settings**  
**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Settings**

**SEDLIST**, *Sename*, *KOPT*

**Lists the DOF solution of a superelement after the use pass.**

[PREP7: Superelements](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***Sename***

Name of the superelement on `File.DSUB` to be listed. If a number, it is the element number of the superelement as used in the use pass. If ALL, list results for all superelements.

**KOPT**

List key:

**0**

List summary data only.

**1**

List full contents. Be aware that the listing may be extensive.

## Notes

Lists the degree of freedom solution of a superelement after the substructure use pass. Results may be listed for any superelement on `File.DSUB`.

This command is valid in any processor.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>General Postproc>List Results>Superelem DOF**

**Utility Menu>List>Results>Superelem DOF Solu**

---

**SEEXP**, *Sename*, *Usefil*, *Imagky*, *Expopt*

**Specifies options for the substructure expansion pass.**

[SOLUTION: Analysis Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**Sename**

The name (case-sensitive) of the superelement matrix file created by the [substructure](#) generation pass (*Sename*.SUB). Defaults to the initial jobname `File`. If a number, it is the element number of the superelement as used in the use pass.

**Usefil**

The name of the file containing the superelement degree-of-freedom (DOF) solution created by the substructure use pass (*Usefil*.DSUB).

**Imagky**

Key to specify use of the imaginary component of the DOF solution. Applicable only if the use pass is a harmonic (**ANTYPE**,HARMIC) analysis:

**OFF**

Use real component of DOF solution (default).

**ON**

Use imaginary component of DOF solution.

---

### Note

If all solutions are to be expanded (**NUMEXP**,ALL), *Imagky* is ignored and both the real and imaginary solutions are expanded.

**Expopt**

Key to specify whether the superelement (**ANTYPE**,SUBSTR) expansion pass (**EXPASS**,ON) should transform the geometry:

**OFF**

Do not transform node or element locations (default).

**ON**

Transform node or element locations in the FE geometry record of the .rst results file.

**Notes**

Specifies options for the expansion pass of the substructure analysis (**ANTYPE**,SUBSTR). If used in SOLUTION, this command is valid only within the first load step.

If you specify geometry transformation (*Expopt* = ON), you must retrieve the transformation matrix (if it exists) from the specified .SUB file. The command updates the nodal X, Y, and Z coordinates to represent the transformed node locations. The *Expopt* option is useful when you want to expand superelements created from other superelements (via **SETRAN** or **SESYMM** commands). For more information, see [Superelement Expansion in Transformed Locations](#) and [Plotting or Printing Mode Shapes](#).

This command is also valid in **/PREP7**.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Expand Superelem**  
**Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Expand Superelem**

**/SEG**, *Label*, *Aviname*, *DELAY*

**Allows graphics data to be stored in the local terminal memory.**

[GRAPHICS: Set Up](#)

[DISPLAY: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Label**

Storage key:

**SINGL**

Store subsequent display in a single segment (overwrites last storage).

**MULTI**

Store subsequent displays in unique segments [**ANIM**].

**DELET**

Delete all currently stored segments.

**OFF**

Stop storing display data in segments.

**STAT**

Display segment status.

**PC**

This option only applies to PC versions of ANSYS and only when animating via the AVI movie player (i.e., **/DEVICE,ANIM,2**). This command appends frames to the `File.AVI`, so that the animation goes in both directions (i.e., forward--backward--forward). You must have a current animation file to use this option.

***Aviname***

Name of the animation file that will be created when each frame is saved. The `.AVI` extension is applied automatically. Defaults to `Jobname.AVI` if no filename is specified.

***DELAY***

Delay factor between each frame, in seconds. Defaults to 0.015 seconds if no value is specified.

**Command Default**

No segment storage.

**Notes**

Allows graphics data to be stored in the terminal local memory (device-dependent). Storage occurs concurrently with the display.

Although the information from your graphics window is stored as an individual segment, you cannot plot directly (**GPLOT**) from the segment memory.

For the DISPLAY program, the *Aviname* and *DELAY* fields are ignored.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>PlotCtrls>Redirect Plots>Delete Segments**

**Utility Menu>PlotCtrls>Redirect Plots>Segment Status**

**Utility Menu>PlotCtrls>Redirect Plots>To Segment Memory**

**SEGEN**, *Mode*, *nSuper*, *mDof*, *stopStage*

**Automatically generate superelements.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***Mode***

Specify action to take (must be specified as one of the following):

**AUTO**

Turn on feature.

**OFF**

Turn off feature.

***nSuper***

Number of superelements to create. The minimum number of superelements is 2, and the maximum number of superelements is 999. Note that the number of requested superelements may not be the same as the number of defined superelements (see "Notes" (p. 1513) for more details).

***mDof***

Specifies whether to use the master DOF defined by the user.

**YES**

Use master DOF defined by the user with the **M** command.

**NO**

Use the master DOF defined by the automatic generation process. Be aware that this option can generate a large number of master DOFs (see "Notes" (p. 1513) for more details).

***stopStage***

Specifies when to stop the automatic superelement generation process.

**PREVIEW**

Preview the superelements only; stop after creating the domains which will become the superelements, and after creating master DOF on the interfaces between each domain.

**GEN**

Create (generate) the superelements.

## Command Default

No command default. If the command is issued, a *Mode* must be specified. By default, if the AUTO mode is chosen, the **SEGEN** command automatically creates 10 superelements and defines master DOFs at all DOFs on the interfaces between superelements.

## Notes

This command can be used to quickly generate a set of superelements. Each superelement is created in a separate file (jobnameXXX.sub, where XXX is a positive number from 1 to 999).

Due to the heuristics in the automatic domain decomposer, which is used to define the domains that will become superelements, the number of defined superelements may exceed the number of requested superelements. Use the *mDof* and *stopStage* options to determine exactly how many superelements will be created, the interfaces between each superelement, and where master DOF will be defined. With the **/PNUM,DOMAIN** command, you can graphically (visually) preview the elements in each superelement. Then, if required, you can add additional master DOF to (or remove from) the boundaries of the superelements. Use the **SEGEN** command again with *stopStage* = GEN to actually create the superelements.

ANSYS automatically defines master DOF at each of the following: all interface DOF between superelements, all DOF attached to contact elements (**TARGE169** to **CONTA177**), and all DOF associated with nodes having a point load defined. Note that for regular superelements, all interface DOFs must be defined as master DOFs for the correct solution to be obtained. However, for CMS superelements, some of the interface DOFs can be removed without a significant loss of accuracy.

For the case when *mDof* = YES, you should select the preview option first (*stopStage* = PREVIEW) to verify exactly how many superelements will be created and where the superelement boundaries are located. If more superelements will be created than were requested, you should define master DOF on the interface(s) between all superelements.

This command is valid only for substructuring analyses (**ANTYPE**,**SUBSTR**). Use **SEOPT** to specify any options for all of the superelements (e.g., which matrices to reduce), and possibly **CMSOPT** for any CMS substructuring analysis. Note that the created superelements will follow the current **/FILNAME** instead of *SENAME* from **SEOPT**. Also, only one load vector will be written to each *.SUB* file. Multiple load steps are not supported with the automatic superelement generation process.

During the actual creation of the superelements, the output is redirected to *jobname.autotemp*.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>Analysis Options**

**SELIST**, *Sename*, *KOPT*, *KINT*

**Lists the contents of a superelement matrix file.**

[PREP7: Superelements](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *Sename*

The name (case-sensitive) of the superelement matrix file created by the substructure generation pass (*Sename*.*SUB*). Defaults to the current *Jobname*. If a number, it is the element number of the superelement as used in the use pass.

### *KOPT*

List key:

- 0**  
List summary data only.
- 1**  
List contents, except load vectors and matrices.
- 2**  
List contents, except matrices.
- 3**  
List full contents. Be aware that the listing may be extensive.

### *KINT*

Integer printout format key:

- OFF**  
Default.
- ON**  
Long format for large integers.

## Notes

This command is valid in any processor.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Utility Menu>List>Other>Superelem Data

---

### SELM

Specifies "Superelements" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Utility Menu>List>Status>Preprocessor>Superelements

---

### SELTOL, *Toler*

Sets the tolerance for subsequent select operations.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Toler*

Tolerance value. If blank, restores the default tolerance logic.

### Notes

For selects based on non-integer numbers (e.g. coordinates, results, etc.), items within the range  $V_{MIN} - T_{oler}$  and  $V_{MAX} + T_{oler}$  are selected, where  $V_{MIN}$  and  $V_{MAX}$  are the range values input on the xSEL commands (**ASEL**, **ESEL**, **KSEL**, **LSEL**, **NSEL**, and **VSEL**).

The default tolerance logic is based on the relative values of  $V_{MIN}$  and  $V_{MAX}$  as follows:

- If  $V_{MIN} = V_{MAX}$ ,  $T_{oler} = 0.005 \times V_{MIN}$ .
- If  $V_{MIN} = V_{MAX} = 0.0$ ,  $T_{oler} = 1.0E-6$ .
- If  $V_{MAX} \neq V_{MIN}$ ,  $T_{oler} = 1.0E-8 \times (V_{MAX} - V_{MIN})$ .

This command is typically used when  $V_{MAX} - V_{MIN}$  is very large so that the computed default tolerance is therefore large and the xSEL commands selects more than what is desired.

*Toler* remains active until respecified by a subsequent **SELTOL** command. A **SELTOL** < blank > resets back to the default *Toler* logic.

## Menu Paths

This command cannot be accessed from a menu.

## SENERGY, OPT, ANTYPE

Determines the stored magnetic energy or co-energy.

POST1: Magnetics Calculations

MP <> <> <> <> <> <> <> EM EH <> PP <> EME <>

### OPT

Item to be calculated:

- 0 Stored magnetic energy.
- 1 Stored magnetic co-energy.

### ANTYPE

Analysis type:

- 0 Static or transient.
- 1 Harmonic.

## Notes

**SENERGY** invokes an ANSYS macro which calculates the stored magnetic energy or co-energy for all selected elements. (For a harmonic analysis, the macro calculates a time-averaged (rms) stored energy.) A summary table listing the energy by material number is produced. The energy density is also calculated and stored on a per-element basis in the element table [**ETABLE**] with the label MG\_ENG (energy density) or MG\_COENG (co-energy density). The macro erases all other items in the element table [**ETABLE**] and only retains the energy density or co-energy density. Use the **PLETAB** and **PRETAB** commands to plot and list the energy density. The macro is valid for static and low-frequency magnetic field formulations. The macro will not calculate stored energy and co-energy for the following cases:

- Orthotropic nonlinear permanent magnets.
- Orthotropic nonlinear permeable materials.
- Temperature dependent materials.

**SENERGY** is restricted to MKSA units.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.



## Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Co-Energy  
Main Menu>General Postproc>Elec&Mag Calc>Element Based>Energy

**SEOPT**, *Sename*, *SEMATR*, *SEPR*, *SESST*, *EXPMTH*

**Specifies substructure analysis options.**

[SOLUTION: Analysis Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### ***Sename***

The name (case-sensitive) assigned to the superelement matrix file. The matrix file will be named *Se-name*.SUB. This field defaults to *Fname* on the **/FILENAME** command.

### ***SEMATR***

Matrix generation key:

- 1**  
Generate stiffness (or conductivity) matrix (default).
- 2**  
Generate stiffness and mass (or conductivity and specific heat) matrices.
- 3**  
Generate stiffness, mass and damping matrices.

### ***SEPR***

Print key:

- 0**  
Do not print superelement matrices or load vectors.
- 1**  
Print both load vectors and superelement matrices.
- 2**  
Print load vectors but not matrices.

### ***SESST***

Stress stiffening key:

- 0**  
Do not save space for stress stiffening in a later run.
- 1**  
Save space for the stress stiffening matrix (calculated in a subsequent generation run after the expansion pass).

### ***EXPMTH***

Expansion method for expansion pass:

#### **BACKSUB**

Save necessary factorized matrix files for backsubstitution during subsequent expansion passes (default). This normally results in a large usage of disk space

**RESOLVE**

Do not save factorized matrix files. Global stiffness matrix will be reformed during expansion pass. This option provides an effective way to save disk space usage. This option cannot be used if the use pass uses large deflections (**NLGEOM,ON**).

**Notes**

Specifies substructure analysis options (**ANTYPE,SUBSTR**). If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>Analysis Options**

---

**SESYMM**, *Sename*, *Ncomp*, *INC*, *File*, *Ext*, --

**Performs a symmetry operation on a superelement within the use pass.**

[PREP7: Superelements](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***Sename***

The name (case-sensitive) of the superelement matrix file created by the substructure generation pass (*Sename*.SUB). Defaults to the current `JOBNAME`. If a number, it is the element number of a previously defined superelement in the current use pass.

***Ncomp***

Symmetry key:

**X**

X symmetry (default).

**Y**

Y symmetry.

**Z**

Z symmetry.

***INC***

Increment all nodes in the superelement by *INC*.

***File***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

*This field must be input.*

***Ext***

Filename extension (8 character maximum).

The extension defaults to SUB.

--  
Unused field.

## Notes

Performs a symmetry operation on a superelement within the substructure use pass by reversing the sign of component *NComp* in the global Cartesian coordinate system. The node numbers are incremented by *INC*. The new superelement is written to *File.SUB* in the current directory (by default). All master node nodal coordinate systems must be global Cartesian (no rotated nodes allowed).

The maximum number of transformations for a given superelement is five (including **SETRAN**, **SESYMM**, and the large rotation transformation if **NLGEOM** is ON in the use pass).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>By Reflection

**\*SET**, *Par*, *VALUE*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*, *VAL10*

Assigns values to user-named parameters.

[APDL: Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Par*

An alphanumeric name used to identify this parameter. *Par* may be up to 32 characters, beginning with a letter and containing only letters, numbers, and underscores. Examples: ABC A3X TOP\_END. ANSYS command names, function names, label names, component and assembly names, etc., should not be used. Parameter names beginning with an underscore (e.g., \_LOOP) are reserved for use by ANSYS and should be avoided. Parameter names ending in an underscore are not listed by the **\*STATUS** command. Array parameter names must be followed by a subscript, and the entire expression must be 32 characters or less. Examples: A(1,1) NEW\_VAL(3,2,5) RESULT(1000). There is no character parameter substitution for the *Par* field. Table parameters that are used in command fields (where constant values are normally given) are limited to 32 characters.

### *VALUE*

Numerical value or alphanumeric character string (up to 32 characters enclosed in single quotes) to be assigned to this parameter. Examples: A(1,3)=7.4 B='ABC3'. May also be a parameter or a parametric expression. Examples: C=A(1,3) A(2,2)=(C+4)/2. If blank, delete this parameter. Example: A= deletes parameter A.

### *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*, *VAL10*

If *Par* is an array parameter, values *VAL2* through *VAL10* (up to the last nonblank value) are sequentially assigned to the succeeding array elements of the column. Example: **\*SET**,A(1,4),10,11 assigns A(1,4)=10, A(2,4)=11. **\*SET**,B(2,3),'file10','file11' assigns B(2,3)='file10', B(3,3)='file11'.

## Notes

Assigns values to user-named parameters that may be substituted later in the run. The equivalent (and recommended) format is

```
Par = VALUE,VAL2,VAL3, ..., VAL10
```

which may be used in place of **\*SET**,*Par*, ... for convenience.

This command is valid in any processor.

## Parameter Definitions

Parameters (numeric or character) may be scalars (single valued) or arrays (multiple valued in one, two, or three dimensions). An unlimited number of parameter names may be defined in any ANSYS run. For very large numbers of parameters, it is most efficient to define them in alphabetical order.

Parameter values may be redefined at any time. Array parameters may also be assigned values within a do-loop [**\*DO**] for convenience. Internally programmed do-loop commands are also available with the **\*Vxx** commands (**\*VFILL**). Parameter values (except for parameters ending in an underscore) may be listed with the **\*STATUS** command, displayed with the **\*VLOT** command (numeric parameters only), and modified with the **\*VEDIT** command (numeric parameters only).

Older ANSYS-supplied macro files may use parameter names that do not begin with an underscore. Using these macros embedded in your own macros may cause conflicts if the same parameter names are used.

Parameters can also be resolved in comments created by the **/COM** command (see **/COM** for complete documentation). A parameter can be deleted by redefining it with a blank *VALUE*. If the parameter is an array, the entire array is deleted. Parameters may also be defined by a response to a query with the **\*ASK** command or from an "ANSYS-supplied" value with the **\*GET** command.

## Array Parameters

Array parameters must be dimensioned [**\*DIM**] before being assigned values unless they are the result of an array operation or defined using the **implied loop** convention. Scalar parameters that are not defined are initialized to a "near" zero value. Numeric array parameters are initialized to zero when dimensioned, and character array parameters are initialized to blank. An existing array parameter must be deleted before it can be redimensioned. Array parameter names must be followed by a subscript list (enclosed in parentheses) identifying the element of the array. The subscript list may have one, two, or three values (separated by commas). Typical array parameter elements are A(1,1), NEW\_VAL(3,2,5), RESULT(1000). Subscripts for defining an array element must be integers (or parameter expressions that evaluate to integers). Non-integer values are rounded to the nearest integer value. All array parameters are stored as 3-D arrays with the unspecified dimensions set to 1. For example, the 4th array element of a 1-dimensional array, A(4), is stored as array element A(4,1,1). Arrays are patterned after standard FORTRAN conventions.

## Numerical Parameter Substitution

If the parameter name *Par* is input in a numeric argument of a command, the numeric value of the parameter (as assigned with **\*SET**, **\*GET**, =, etc.) is substituted into the command at that point. Substitution occurs only if the parameter name is used between blanks, commas, parentheses, or arithmetic operators (or any combination) in a numeric argument. Substitution can be prevented by enclosing the parameter name *Par* within single quotes ( ' ), if the parameter is alone in the argument; if the parameter is part of an arithmetic

expression, the entire expression must be enclosed within single quotes to prevent substitution. In either case the character string will be used instead of the numeric value (and the string will be taken as 0.0 if it is in a numeric argument).

A forced substitution is available in the text fields of the **/TITLE**, **/STITLE**, **/TLABEL**, **/AN3D**, **/SYP** (*ARG1--ARG8*), and **\*ABBR** commands by enclosing the parameter within percent (%) signs. Also, parameter substitution may be forced within the file name or extension fields of commands having these fields by enclosing the parameter within percent (%) signs. Array parameters [**\*DIM**] must include a subscript (within parentheses) to identify the array element whose value is to be substituted, such as A(1,3). Out-of-range subscripts result in an error message. Non-integer subscripts are allowed when identifying a TABLE array element for substitution. A proportional linear interpolation of values among the nearest array elements is performed before substitution. Interpolation is done in all three dimensions.

---

### Note

Interpolation is based upon the assigned index numbers which must be defined when the table is filled [**\*DIM**].

## Character Parameter Substitution

Most alphanumeric arguments permit the use of character parameter substitution. When the parameter name *Par* input, the alphanumeric value of the parameter is substituted into the command at that point. Substitution can be suppressed by enclosing the parameter name within single quotes ( ' ). Forced substitution is available in some fields by enclosing the parameter name within percent (%) signs. Valid forced substitution fields include command name fields, *Fname* (filename) or *Ext* (extension) arguments, **\*ABBR** command (*Abbr* arguments), **/TITLE** and **/STITLE** commands (*Title* argument) and **/TLABEL** command (*Text* argument). Character parameter substitution is also available in the **\*ASK**, **/AN3D**, **\*CFWRITE**, **\*IF**, **\*ELSEIF**, **\*MSG**, **\*SET**, **\*USE**, **\*VREAD**, and **\*VWRITE** commands. Character array parameters must include a subscript (within parentheses) to identify the array element whose value is to be substituted.

## Parameter Expressions

If a parameter *operation expression* is input in a numeric argument, the numeric value of the expression is substituted into the command at that point. Allowable operation expressions are of the form

E1oE2oE3 ...oE10

where E1, E2, etc. are expressions connected by operators (o). The allowable operations (o) are

+ - \* / \*\* < >

For example, A+B\*\*C/D\*E is a valid operation expression. The \* represents multiplication and the \*\* represents exponentiation.

---

### Note

Exponentiation of a negative number (without parentheses) to an integer power follows standard FORTRAN hierarchy conventions; that is, the positive number is exponentiated and then the sign is attached. Thus, -4\*\*2 is evaluated as -16. If parentheses are applied, such as (-4)\*\*2, the result is 16.

A parameter is evaluated as a number within parentheses before exponentiation. Exponentiation of a negative number to a non-integer power is performed by exponentiating the positive number and prepending the minus sign, for example,  $-4^{2.3}$  is  $-(4^{2.3})$ . The  $<$  and  $>$  operators allow conditional substitution. For example,  $E1 < E2$  substitutes the value of  $E1$  if the comparison is true or the value of  $E2$  if the comparison is false.

Spaces should not be used around operation symbols since " \*" (a space and a star) makes the rest of the line a comment. Operation symbols (or symbols and signs) may not be immediately adjacent to each other. Parentheses may be used to separate symbols and signs, to determine a hierarchy of operations, or for clarity. For example,  $A^{(-B)}$  must be used instead of  $A^{-B}$ . Numbers ending with  $+0nn$  or  $-0nn$  are assumed to be of exponential form (as written on files by some computer systems) so that  $123-002$  is  $123E-2$  while  $123-2$  is  $121$ . This form of exponential data should not be input directly. The default hierarchy follows the standard FORTRAN conventions, namely:

- operations in parentheses (innermost first)
- then exponentiation (right to left)
- then multiplication or division (left to right)
- then unary association (such as  $+A$  or  $-A$ )
- then addition or subtraction (left to right)
- then logical evaluations (left to right).

Expressions (E) may be a constant, a parameter, a function, or another operation expression (of the form  $E1oE2oE3 \dots oE10$ ). Functions are of the form  $FTN(A)$  where the argument (A) may itself be of the form  $E1oE2oE3 \dots oE10$ . Operations are recursive to a level of four deep (three levels of internally nested parentheses). Iterative floating point parameter arithmetic should not be used for high precision input because of the accumulated numerical round off-error. Up to 10 expressions are accepted within a set of parenthesis.

Valid functions (which are based on standard FORTRAN functions where possible) are:

SIN(X)	Sine
COS(X)	Cosine
TAN(X)	Tangent
ASIN(X)	Arcsine
ACOS(X)	Arccosine
ATAN(X)	Arctangent
ATAN2(Y,X)	Arctangent (Y/X) with the sign of each component considered
SINH(X)	Hyperbolic sine
COSH(X)	Hyperbolic cosine
TANH(X)	Hyperbolic tangent
SQRT(X)	Square root
ABS(X)	Absolute value
SIGN(X,Y)	Absolute value of X with sign of Y. Y=0 results in positive sign
NINT(X)	Nearest integer
MOD(X,Y)	Remainder of X/Y Y=0 returns zero (0)
EXP(X)	Exponential

LOG( <i>X</i> )	Natural log
LOG10( <i>X</i> )	Common log
RAND( <i>X</i> , <i>Y</i> )	Random number, where <i>X</i> is the lower bound, and <i>Y</i> is the upper bound
GDIS( <i>X</i> , <i>Y</i> )	Random sample of Gaussian distributions, where <i>X</i> is the mean, and <i>Y</i> is the standard deviation
LWCASE( <i>CPARM</i> )	Lowercase equivalent of character parameter <i>CPARM</i>
UPCASE( <i>CPARM</i> )	Uppercase equivalent of character parameter <i>CPARM</i>
VALCHR( <i>CPARM</i> )	Numeric value of character parameter <i>CPARM</i> (If <i>CPARM</i> is a numeric parameter, returns 0.0)
CHRVAL( <i>PARM</i> )	Character value of numerical parameter <i>PARM</i> . For ABS( <i>PARM</i> ) < 10, character value format is F8.5; for $10 \leq \text{ABS}(\text{PARM}) < 1000$ , format is F8.3; for $1,000 \leq \text{ABS}(\text{PARM}) < 10,000,000$ , format is F8.0. For $10,000,000 \leq \text{PARM} < 100,000,000$ , format is also F8.0. Otherwise result is 0.0 and is not a character value.
IBSET( <i>b1</i> , <i>n2</i> )	Set the <i>n2</i> bit in value <i>b1</i> (bits are numbered from 0 to 31)
IBCLR( <i>b1</i> , <i>n2</i> )	Clear the <i>n2</i> bit in value <i>b1</i>
BTEST( <i>b1</i> , <i>n2</i> )	Test the <i>n2</i> bit in value <i>b1</i> (return true (1.0) if bit is set)
BITAND( <i>b1</i> , <i>b2</i> )	Bitwise AND of value <i>b1</i> and <i>b2</i>
BITOR( <i>b1</i> , <i>b2</i> )	Bitwise OR of value <i>b1</i> and <i>b2</i>
BITXOR( <i>b1</i> , <i>b2</i> )	Bitwise XOR of value <i>b1</i> and <i>b2</i>
BITSET( <i>b1</i> , <i>b2</i> )	Set the <i>b2</i> bits in <i>b1</i>
BITCLEAR( <i>b1</i> , <i>b2</i> )	Clear the <i>b2</i> bits in <i>b1</i>

Function arguments (*X*,*Y*, etc.) must be enclosed within parentheses and may be numeric values, parameters, or expressions. Input arguments for angular functions must evaluate to radians by default. Output from angular functions are also in radians by default. See the **\*AFUN** command to use degrees instead of radians for the angular functions. See the **\*VFUN** command for applying these parameter functions to a sequence of array elements. Additional functions, called "get functions" are described with the **\*GET** command. For SINH, COSH, TANH, and EXP, if  $X > 80$ , the value returned is zero.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Section**  
**Main Menu>Solution>Define Loads>Delete>Structural>Section**  
**Utility Menu>Parameters>Scalar Parameters**

**SET**, *Lstep*, *Sbstep*, *Fact*, *KIMG*, *TIME*, *ANGLE*, *NSET*, *ORDER*

**Defines the data set to be read from the results file.**

POST1: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lstep*

Load step number of the data set to be read (defaults to 1):

***N***

Read load step *N*.

**FIRST**

Read the first data set (*Sbstep* and *TIME* are ignored).

**LAST**

Read the last data set (*Sbstep* and *TIME* are ignored).

**NEXT**

Read the next data set (*Sbstep* and *TIME* are ignored). If at the last data set, the first data set will be read as the next.

**PREVIOUS**

Read the previous data set (*Sbstep* and *TIME* are ignored). If at the first data set, the last data set will be read as the previous.

**NEAR**

Read the data set nearest to *TIME* (*Sbstep* is ignored). If *TIME* is blank, read the first data set.

**LIST**

Scan the results file and list a summary of each load step. (*KIMG*, *TIME*, *ANGLE*, and *NSET* are ignored.)

***Sbstep***

Substep number (within *Lstep*). Defaults to the last substep of the load step (except in a buckling or modal analysis). For a buckling (**ANTYPE**,BUCKLE) or modal (**ANTYPE**,MODAL) analysis, *Sbstep* corresponds to the mode number. Specify *Sbstep* = LAST to store the last substep for the specified load step (that is, issue a **SET**,*Lstep*,LAST command).

If *Lstep* = LIST, *Sbstep* = 0 or 1 lists the basic step information. *Sbstep* = 2 also lists the basic step information, but includes the load step title, and labels imaginary data sets if they exist.

***Fact***

Scale factor applied to data read from the file. If zero (or blank), a value of 1.0 is used. A nonzero factor excludes non-summable items. Harmonic velocities or accelerations may be calculated from the displacement results from a Modal (**ANTYPE**,MODAL) or Harmonic Response (**ANTYPE**,HARMIC) analyses. If *Fact* = VELO, the harmonic velocities (*v*) are calculated from the displacements (*d*) at a particular frequency (*f*) according to the relationship  $v = 2 \pi f d$ . Similarly, if *Fact* = ACEL, the harmonic accelerations (*a*) are calculated as  $a = (2 \pi f)^2 d$ .

If *Lstep* = LIST in an analysis employing **rezoning**, *Fact* can restrict the output to a file type such as *.rst* or *.rsnn*. For example, to see only the portion of the data set associated with the *.rs01* file, issue a **SET**,LIST,,RS01 command.

***KIMG***

Used only with complex results (harmonic and complex modal analyses).

**0 or REAL**

Store the real part of complex solution (default).

**1, 2 or IMAG**

Store the imaginary part of a complex solution.

**3 or AMPL**

Store the amplitude

**4 or PHAS**

Store the phase angle. The angle value, expressed in degrees, will be between -180° and +180°.



**TIME**

Time-point identifying the data set to be read. For the harmonic response analyses, time corresponds to the frequency. For the buckling analysis, time corresponds to the load factor. Used only in the following cases: If *Lstep* = NEAR, read the data set nearest to *TIME*. If both *Lstep* and *Sbstep* are zero (or blank), read data set at time = *TIME*. Do not use *TIME* to identify the data set to be read if you used the arc-length method (**ARCLEN**) in your solution. If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets (except in **rezoning**). Solution items not written to the results file (**OUTRES**) for either data set will result in a null item after data set interpolation. If *TIME* is beyond the last time point on the file, the last time point will be used.

**ANGLE**

Circumferential location (0.0 to 360°). Defines the circumferential location for the harmonic calculations used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the harmonic elements (**PLANE25**, **PLANE75**, **PLANE78**, **FLUID81**, **PLANE83**, and **SHELL61**) of the load case. See the *Theory Reference for the Mechanical APDL and Mechanical Applications* for details.

**Note**

The factored values of applied constraints and loads will overwrite any values existing in the database. If *ANGLE* = NONE, all harmonic factors are set to 1 and postprocessing will yield the solution output. When using *ANGLE* = NONE with *MODE* > 0, the combined stresses and strains are not valid. The default value of *ANGLE* is 0.0, but if the **SET** command is not used, the effective default is NONE.

**NSET**

Data set number of the data set to be read. If a positive value for *NSET* is entered, *Lstep*, *Sbstep*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **SET,LIST**.

**ORDER**

Sorts the harmonic index results in ascending order of eigenfrequencies or buckling load multipliers. This value applies to **cyclic symmetry** buckling and modal analyses only, and is valid only when *Lstep* = FIRST, LAST, NEXT, PREVIOUS, NEAR or LIST.

**Notes**

Defines the data set to be read from the results file into the database. Various operations may also be performed during the read operation. The database must have the model geometry available (or use the **RESUME** command before the **SET** command to restore the geometry from *Jobname*.DB). Values for applied constraints [**D**] and loads [**F**] in the database will be replaced by their corresponding values on the results file, if available. (See the description of the **OUTRES** command.) In a single load step analysis, these values are usually the same, except for results from harmonic elements. (See the description of the *ANGLE* value above.)

In an interactive run, the sorted list (ORDER option) is also available for results-set reading via a GUI pick option.

During **rezoning**, *Lstep* values NEXT, PREVIOUS, and NEAR are not available.

In ANSYS, you can postprocess results without issuing a **SET** command if the solution results were saved to the database file (*Jobname*.DB). Distributed ANSYS, however, can only postprocess using the results file (*Jobname*.RST) and cannot use the *Jobname*.DB file since no solution results are written to the database. Therefore, you must issue a **SET** command before postprocessing in Distributed ANSYS.

When postprocessing amplitudes or phases (*KIMG* = AMPL or PHAS):

- The only appropriate coordinate system is the solution coordinate system (**RSYS**, SOLU). When displaying the displacement amplitudes, use a contour display (**PLNSOL** command). Because a deformed shape display (**PLDISP** command) could lead to a non-physical shape, the displacement scaling is off by default (**/DSCALE,,OFF**).
- The conversion is not valid for derived results such as principal stress/strain, equivalent stress/strain, and USUM.
- Cyclic symmetry results are not supported.

## Menu Paths

**Main Menu>General Postproc>List Results>Detailed Summary**  
**Main Menu>General Postproc>List Results>Detailed Summary (Freq ordered)**  
**Main Menu>General Postproc>Read Results>By Load Step**  
**Main Menu>General Postproc>Read Results>By Pick**  
**Main Menu>General Postproc>Read Results>By Pick (Freq ordered)**  
**Main Menu>General Postproc>Read Results>First Set**  
**Main Menu>General Postproc>Read Results>Last Set**  
**Main Menu>General Postproc>Read Results>Next Set**  
**Main Menu>General Postproc>Read Results>Previous Set**  
**Main Menu>General Postproc>Results Summary**  
**Main Menu>General Postproc>Results Summary (Freq ordered)**  
**Utility Menu>List>Results>Load Step Summary**

**SETFGAP**, *GAP*, *ROPT*, *--*, *PAMB*, *ACF1*, *ACF2*, *PREF*, *MFP*

**Updates or defines the real constant table for squeeze film elements.**

[PREP7: Real Constants](#)

MP ME <> <> <> <> <> <> <> <> <> PP <> EME MFS

### **GAP**

Gap separation.

### **ROPT**

Real constant set option.

**0**

Creates separate real constant sets for each selected element with the specified real constant values (default).

**1**

Updates existing real constant sets. The gap separation is updated from displacement results in the database. Other real constants are updated as specified in the command input parameters.

**--**

Unused field

### **PAMB**

Ambient pressure.

### **ACF1, ACF2**

Accommodation factor 1 and 2.

**PREF**

Reference pressure for mean free path.

**MFP**

Mean free path.

## Notes

This command is used for large signal cases to update the gap separation real constant on a per-element basis. Issue this command prior to solution using the default *ROPT* value to initialize real constant sets for every fluid element. After a solution, you can re-issue the command to update the real constant set for a subsequent analysis. See "[Thin Film Analysis](#)" for more information on thin film analyses.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>ThinFilm**

**Main Menu>Preprocessor>Real Constants>ThinFilm**

**Main Menu>Solution>Load Step Opts>Other>Real Constants>ThinFilm**

**SETRAN**, *Sename*, *KCNTO*, *INC*, *File*, *Ext*, *--*, *DX*, *DY*, *DZ*, *NOROT*

**Creates a superelement from an existing superelement.**

[PREP7: Superelements](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**Sename**

The name (case-sensitive) of the file containing the original superelement matrix created by the generation pass (*Sename*.SUB). The default is the current *Jobname*. If *Sename* is a number, it is the element number of a previously defined superelement in the current use pass.

**KCNTO**

The reference number of the coordinate system to where the superelement is to be transferred. The default is the global Cartesian system. Transfer occurs from the active coordinate system.

**INC**

The node offset. The default is zero. All new element node numbers are offset from those on the original by *INC*.

**File**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

*This field requires input.*

**Ext**

Filename extension (8 character maximum).

The extension defaults to SUB.

**--**

Unused field.

***DX, DY, DZ***

Node location increments in the global Cartesian coordinate system. Defaults to zero.

***NOROT***

Node rotation [key](#):

**0**

The nodal coordinate systems of the transferred superelement rotate into the *KCNTO* system. (That is, the nodal coordinate systems rotate with the superelement.) The superelement matrices remain unchanged. This value is the default.

**1**

The nodal coordinate systems do not rotate. (That is, they remain fixed in their original global orientation.) The superelement matrices and load vectors are modified if any rotations occur.

## Notes

The **SETRAN** command creates a superelement from an existing superelement and writes the new element to a file. You can then issue an **SE** command to read the new element (during the [use pass](#)).

You can create a superelement from an original by:

- Transferring the original's geometry from the active coordinate system into another coordinate system (*KCNTO*)
- Offsetting its geometry in the global Cartesian coordinate system (*DX, DY, and DZ* )
- Offsetting its node numbers (*INC*).

A combination of methods is valid. If you specify both the geometry transfer and the geometry offset, the transfer occurs first.

If you specify rotation of the transferred superelement's nodal coordinate systems into the *KCNTO* system (*NOROT* = 0), the rotated nodes cannot be coupled via the **CP** command; in this case, issue the **CE** command instead. If you specify *no* rotation of the nodal coordinate systems (*NOROT* = 1) for models with displacement degrees of freedom, and *KCNTO* is not the active system, the superelement *SeName* must have six MDOF at each node that has MDOF; therefore, only elements with all six structural DOFs are valid in such cases.

There is no limit to the number of copies that can be made of a superelement, provided the copies are all generated from the same original superelement. However, nested copies are limited to five. In other words, the total number of different *SeName* usages on the **SETRAN** and **SESYMM** commands is limited to five.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>By CS Transfer**

**Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>By Geom Offset**

---

**SEXP**, *LabR*, *Lab1*, *Lab2*, *EXP1*, *EXP2*

**Forms an element table item by exponentiating and multiplying.**

[POST1: Element Table](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**LabR**

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

**Lab1**

First labeled result item in operation.

**Lab2**

Second labeled result item in operation (may be blank).

**EXP1**

Exponent applied to *Lab1*.

**EXP2**

Exponent applied to *Lab2*.

## Notes

Forms a labeled result item (see **ETABLE** command) for the selected elements by exponentiating and multiplying two existing labeled result items according to the operation:

$$LabR = (|Lab1|^{EXP1}) \times (|Lab2|^{EXP2})$$

Roots, reciprocals, and divides may also be done with this command.

## Menu Paths

**Main Menu>General Postproc>Element Table>Exponentiate**

**SF**, *Nlist*, *Lab*, *VALUE*, *VALUE2*

**Specifies surface loads on nodes.**

[SOLUTION: FE Surface Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

**Nlist**

Nodes defining the surface upon which the load is to be applied. Use the label ALL or P, or a component name. If ALL, all selected nodes **[NSEL]** are used (default). If P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**Lab**

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the [Element Reference](#).

Discipline	Surface Load Label	Label Description
Structural	PRES	pressure

Discipline	Surface Load Label	Label Description
	FREQ	frequency (harmonic analyses only)
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
	RAD	radiation
	RDSF	surface-to-surface radiation
Acoustic fluid	FSI[2]	fluid-structure interaction flag
	IMPD	impedance boundary flag
Magnetic	MXWF	Maxwell force flag
	MCI	magnetic circuit interface
Electric	CHRG	surface charge density
	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for <a href="#">INFIN110</a> and <a href="#">INFIN111</a>
High-frequency electromagnetic	PORT	number 1-50 for a waveguide exterior port
	SHLD	surface shielding properties
	MXWF	Maxwell surface flag of equivalent source surface
	IMPD	surface impedance
Field-surface interface	FSIN[3]	field-surface interface number
Poromechanics	FFLX	Fluid flow flux

1. Thermal labels CONV and HFLUX are mutually exclusive.
2. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the [FLUID29](#), [FLUID30](#), [FLUID220](#), and [FLUID221](#) elements.
3. For a fluid-solid interaction analysis, apply the field-surface interface flag (Label = FSIN) twice: once for the fluid side ([FLUID141](#) or [FLUID142](#) elements) and once for the solid side.

#### VALUE

Surface load value or table name reference for specifying tabular boundary conditions.

If  $Lab = PRES$ ,  $VALUE$  is the real component of the pressure.

If  $Lab = CONV$ ,  $VALUE$  is typically the film coefficient and  $VALUE2$  (below) is typically the bulk temperature. If  $VALUE = -N$ , the film coefficient may be a function of temperature and is determined from the HF property table for material  $N$  [MP]. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user-defined for some elements.

If  $Lab = MCI$ ,  $VALUE$  indicates current direction (-1; current flow into the element face (IN), +1; current flow out of the element face (OUT)). If  $Lab = RAD$ ,  $VALUE$  is surface emissivity.

If  $Lab = PORT$ ,  $VALUE$  is a port number representing a waveguide exterior port. The port number must be an integer between 1 and 50.

If  $Lab = SHLD$ ,  $VALUE$  is surface conductivity.

If  $Lab = IMPD$ ,  $VALUE$  is resistance in ohms/square.

If  $Lab = RDSF$ ,  $VALUE$  is the emissivity value; the following conditions apply: If  $VALUE$  is between 0 and 1, apply a single value to the surface. If  $VALUE = -N$ , the emissivity may be a function of the temperature, and is determined from the **EMISS** property table for material  $N$  (**MP**). The material  $N$  does not need to correlate with the underlying solid thermal elements.

If  $Lab = FSIN$  in a Multi-field solver (single or multiple code coupling) analysis,  $VALUE$  is the surface interface number. If  $Lab = FSIN$  in a unidirectional ANSYS to CFX analysis,  $VALUE$  is not used unless the ANSYS analysis is performed using the Multi-field solver.

#### **VALUE2**

Second surface load value (if any).

If  $Lab = PRES$ ,  $VALUE2$  is the imaginary component of the pressure. Imaginary pressures can only be used by **SURF153**, **SURF154** and **SURF159**, and can only be used for a full harmonic response analysis (**HROPT,FULL**), or by a mode superposition harmonic response analysis (**HROPT,MSUP**) if the mode extraction method is Block Lanczos (**MODOPT,LANB**), PCG Lanczos (**MODOPT,LANPCG**), or Supernode (**MODOPT,SNODE**).

If  $Lab = CONV$ ,  $VALUE2$  is typically the bulk temperature.

If  $Lab = RAD$ ,  $VALUE2$  is the ambient temperature.

If  $Lab = SHLD$ ,  $VALUE2$  is relative permeability and defaults to 1.0.

If  $Lab = IMPD$ ,  $VALUE2$  is reactance in ohms/square.

If  $Lab = RDSF$ ,  $VALUE2$  is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open, radiation will also occur to ambient. If  $VALUE2$  is negative radiation direction is reversed and will occur inside the element for the flagged radiation surfaces. Negative value of enclosure number is applicable for **FLUID141** and **FLUID142** elements, to model radiation occurring between surfaces inside the fluid domain.

If  $Lab = FSIN$  in a unidirectional ANSYS to CFX analysis,  $VALUE2$  is the surface interface number (not available from within the GUI).

If  $Lab = PORT$ ,  $VALUE2$  is not used.

## **Notes**

Individual nodes may not be entered for this command. The node list is to identify a surface and the *Nlist* field must contain a sufficient number of nodes to define an element surface. The loads are internally stored on element faces defined by the specified nodes. All nodes on an element face (including midside nodes, if any) must be specified for the face to be used, and the element must be selected.

If all nodes defining a face are shared by an adjacent face of another selected element, the face is not free and will not have a load applied. If more than one element can share the same nodes (for example, a surface element attached to a solid element), select the desired element type before issuing the **SF** command. The **SF** command applies only to area and volume elements.



For shell elements, if the specified nodes include face one (which is usually the bottom face) along with other faces (such as edges), only face one is used. Where faces cannot be uniquely determined from the nodes, or where the face does not fully describe the load application, use the **SFE** command. A load key of 1 (which is typically the first loading condition on the first face) is used if the face determination is not unique. A uniform load value is applied over the element face.

See the **SFBEAM** command for applying surface loads to beam elements. See the **SFGRAD** command for an alternate tapered load capability. See the **SFFUN** command for applying loads from a node vs. value function. Also see the **SFE** command for applying tapered loads on individual element faces. Use the **SFDELETE** command to delete loads applied with this command. Use the **SFCUM** command to accumulate (add) surface loads applied with **SF**.

Tabular boundary conditions ( $VALUE = \%tabname\%$  and/or  $VALUE2 = \%tabname\%$ ) are available for the following surface load labels (*Lab*) only: PRES (real and/or imaginary components), CONV (film coefficient and/or bulk temperature) or HFLUX, and RAD (surface emissivity and ambient temperature). Use the **\*DIM** command to define a table.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppImped\_E>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppShield>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppInfinite>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppMaxwell>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppMCI>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Node Components**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Nodes**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Surface Rad>On Nodes**  
**Main Menu>Preprocessor>Trefftz Domain>AppInfinite>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppImped\_E>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppShield>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppMCI>On Nodes**



**Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Node Components**  
**Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Nodes**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Surface Rad>On Nodes**

**SFA**, *AREA*, *LKEY*, *Lab*, *VALUE*, *VALUE2*

**Specifies surface loads on the selected areas.**

**SOLUTION:** Solid Surface Loads

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### **AREA**

Area to which surface load applies. If ALL, apply load to all selected areas [ASEL]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component may be substituted for *AREA*.

#### **LKEY**

Load key associated with surface load (defaults to 1). Load keys (1,2,3, etc.) are listed under "Surface Loads" in the input data table for each element type in the *Element Reference*. *LKEY* is ignored if the area is the face of a volume region meshed with volume elements.

#### **Lab**

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each area type in the *Element Reference*.

<b>Discipline</b>	<b>Body Load Label</b>	<b>Label Description</b>
Structural	PRES	pressure
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
	RAD	radiation
	RDSF	surface-to-surface radiation
Fluid	FSI[2]	fluid-structure interaction flag
	IMPD	impedance boundary flag
Magnetic	MXWF	Maxwell force flag
	MCI	magnetic circuit interface
Electric	CHRG	surface charge density
	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for INFIN110 and INFIN111

Discipline	Body Load Label	Label Description
High-frequency electromagnetic	PORT	number 1-50 for a waveguide exterior port
	SHLD	surface shielding properties
	IMPD	surface impedance
Field-surface interface	FSIN[3]	field-surface interface number

1. Thermal labels CONV and HFLUX are mutually exclusive.
2. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID129 or FLUID130 elements.
3. For a fluid-solid interaction analysis, apply the field-surface interface flag (Label = FSIN) twice: once for the fluid side (FLUID141 or FLUID142 elements) and once for the solid side.

**VALUE**

Surface load value or table name reference for specifying tabular boundary conditions.

If  $Lab = CONV$ ,  $VALUE$  is typically the film coefficient and  $VALUE2$  (below) is typically the bulk temperature. If  $Lab = CONV$  and  $VALUE = -N$ , the film coefficient may be a function of temperature and is determined from the HF property table for material  $N$  [MP]. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user-defined for some elements.

If  $Lab = MCI$ ,  $VALUE$  indicates current direction (-1; current flow into the element face (IN), +1; current flow out of the element face (OUT)).

If  $Lab = RAD$ ,  $VALUE$  is the surface emissivity.

If  $Lab = PORT$ ,  $VALUE$  is a port number representing a waveguide exterior port. The port number must be an integer between 1 and 50.

If  $Lab = SHLD$ ,  $VALUE$  is surface conductivity.

If  $Lab = IMPD$ ,  $VALUE$  is resistance in ohms/square.

If  $Lab = RDSF$ ,  $VALUE$  is the emissivity value; the following conditions apply: If  $VALUE$  is between 0 and 1, apply a single value to the surface. If  $VALUE = -N$ , the emissivity may be a function of the temperature, and is determined from the EMISS property table for material  $N$  (MP). The material  $N$  does not need to correlate with the underlying solid thermal elements.

If  $Lab = FSIN$  in a Multi-field solver (single or multiple code coupling) analysis,  $VALUE$  is the surface interface number and  $LKEY$  is ignored. If  $Lab = FSIN$  in a unidirectional ANSYS to CFX analysis,  $VALUE$  is not used unless the ANSYS analysis is performed using the Multi-field solver.

**VALUE2**

Second surface load value (if any).

If  $Lab = CONV$ ,  $VALUE2$  is typically the bulk temperature.

If  $Lab = RADVALUE2$  is ambient temperature.

If  $Lab = SHLD, VALUE2$  is relative permeability (defaults to 1.0).

If  $Lab = IMPD, VALUE2$  is reactance in ohms/square.

If  $Lab = RDSF, VALUE2$  is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open, radiation will also occur to ambient. If  $VALUE2$  is negative radiation direction is reversed and will occur inside the element for the flagged radiation surfaces. Negative value of enclosure number is applicable for **FLUID141** and **FLUID142** elements, to model radiation occurring between surfaces inside the fluid domain.

If  $Lab = FSIN$  in a unidirectional ANSYS to CFX analysis,  $VALUE2$  is the surface interface number (not available from within the GUI).

## Notes

Surface loads may be transferred from areas to elements with the **SFTRAN** or **SBCTRAN** commands. See the **SFGRAD** command for an alternate tapered load capability.

Tabular boundary conditions ( $VALUE = \%tabname\%$  and/or  $VALUE2 = \%tabname\%$ ) are available for the following surface load labels ( $Lab$ ) only: PRES (real and/or imaginary components), CONV (film coefficient and/or bulk temperature) or HFLUX, and RAD (surface emissivity and ambient temperature). Use the **\*DIM** command to define a table.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppImped\_E>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppShield>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppInfinite>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppMaxwell>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppMCI>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Areas**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Surface Rad>On Areas**  
**Main Menu>Preprocessor>Trefftz Domain>AppInfinite>On Areas**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppImped\_E>On Areas**  
**Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppShield>On Areas**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Areas**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas**

Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Areas  
 Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Areas  
 Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Areas  
 Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Areas  
 Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Areas  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Areas  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppMCI>On Areas  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Areas  
 Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Areas  
 Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Areas  
 Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Areas  
 Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Areas  
 Main Menu>Solution>Define Loads>Apply>Thermal>Surface Rad>On Areas

## SFACT, TYPE

Allows safety factor or margin of safety calculations to be made.

POST1: Element Table

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

### TYPE

Type of calculation:

- 0**  
No nodal safety factor or margin of safety calculations.
- 1**  
Calculate and store safety factors in place of nodal stresses.
- 2**  
Calculate and store margins of safety in place of nodal stresses.

## Command Default

No nodal safety factor or margin of safety calculations.

## Notes

Allows safety factor (SF) or margin of safety (MS) calculations to be made for the average nodal stresses according to:

$$SF = SALLOW/|Stress|$$

$$MS = (SALLOW/|Stress|) - 1.0$$

Calculations are done during the display, select, or sort operation (in the active coordinate system [**RSYS**]) with results stored in place of the nodal stresses. Use the **PRNSOL** or **PLNSOL** command to display the results.

## Note

The results are meaningful only for the stress (SIG1, SIGE, etc.) upon which **SALLOW** is based. Nodal temperatures used are those automatically stored for the node. Related commands are **SFCALC**, **SALLOW**, **TALLOW**.

## Menu Paths

Main Menu>General Postproc>Safety Factor>Restore NodeStrs

Main Menu>General Postproc>Safety Factor>SF for Node Strs

## SFADELE, AREA, LKEY, Lab

Deletes surface loads from areas.

**SOLUTION:** Solid Surface Loads

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### AREA

Area to which surface load deletion applies. If ALL, delete load from all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *AREA*.

### LKEY

Load key associated with surface load (defaults to 1). See the **SFA** command for details.

### Lab

Valid surface load label. If ALL, use all appropriate labels. See the **SFA** command for labels.

## Notes

Deletes surface loads (and all corresponding finite element loads) from selected areas.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Surface Ld>On All Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppImped\_E>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppShield>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelExtPort>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppMCI>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Surface Rad>On Areas  
 Main Menu>Solution>Define Loads>Delete>All Load Data>All Surface Ld>On All Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppImped\_E>On Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppShield>On Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelExtPort>On Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Areas  
 Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Areas  
 Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Areas  
 Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Areas  
 Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Areas  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Areas  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppMCI>On Areas  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Areas  
 Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Areas  
 Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Areas  
 Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Areas  
 Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Areas  
 Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Areas  
 Main Menu>Solution>Define Loads>Delete>Thermal>Surface Rad>On Areas

---

## SFALIST, AREA, Lab

Lists the surface loads for the specified area.

SOLUTION: Solid Surface Loads

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### AREA

Area at which surface load is to be listed. If ALL (or blank), list for all selected areas [ASEL]. If AREA = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for AREA.

### Lab

Valid surface load label. If ALL (or blank), use all appropriate labels. See the SFA command for labels.

## Notes

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Areas  
 Utility Menu>List>Loads>Surface Loads>On Picked Areas

**SFBEAM**, *ELEM*, *LKEY*, *Lab*, *VALI*, *VALJ*, *VAL2I*, *VAL2J*, *IOFFST*, *JOFFST*, *LENRAT*

**Specifies surface loads on beam and pipe elements.**

SOLUTION: FE Surface Loads

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

***ELEM***

Element to which surface load is applied. If ALL, apply load to all selected beam elements (**ESEL**). If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted in *ELEM*.

***LKEY***

Load key associated with surface load (defaults to 1). Load keys (1, 2, 3, etc.) are listed under "Surface Loads" in the input table for each element type in the *Element Reference*. For beam and some pipe elements, the load key defines the load orientation.

***Lab***

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *Element Reference*. Structural labels: PRES (pressure).

***VALI*, *VALJ***

Surface load values at nodes I and J. If *VALJ* is blank, it defaults to *VALI*. If *VALJ* is zero, a zero is used.

***VAL2I*, *VAL2J***

Second surface load values at nodes I and J. Currently not used.

***IOFFST*, *JOFFST***

Offset distance from node I (toward node J) where *VALI* is applied, and offset distance from node J (toward node I) where *VALJ* is applied, respectively.

***LENRAT***

Offset distance flag:

**0**

Offset is in terms of length units (default).

**1**

Offset is in terms of a length ratio (0.0 to 1.0).

## Notes

Specifies surface loads on the selected beam elements. Distributed loads are applied on a force-per-length basis (that is, the width of the underlying element is not considered). To list and delete surface loads applied with this command, use the **SFELIST** and **SFEDELE** commands, respectively.

If no offset values (*IOFFSET* and *JOFFSET*) are specified, the load is applied over the full element length. Values may also be input as length fractions, depending on the *LENRAT* setting. For example, assuming a line length of 5.0, an *IOFFST* of 2.0 with *LENRAT* = 0 or an *IOFFST* of 0.4 with *LENRAT* = 1 represent the same point. If *JOFFST* = -1, *VALI* is assumed to be a point load at the location specified via *IOFFST*, and *VALJ* is ignored. (*IOFFSET* cannot be equal to -1.) The offset values are stepped even if you issue a **KBC,0** command.

Offsets are only available for element types **BEAM188** and **PIPE288** if using the cubic shape function (**KEYOPT(3) = 3**) for those element types.

To accumulate (add) surface loads applied with this command, use the **SFCUM,,ADD** command. Use the same offset values used on the previous **SFBEAM** command (for a given element face); otherwise, the loads do not accumulate. If no offsets are specified, the command applies the previous offset values.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Beams**  
**Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Beams**

**SFCALC**, *LabR, LabS, LabT, TYPE*

**Calculates the safety factor or margin of safety.**

POST1: Element Table

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

**LabR**

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

**LabS**

Labeled result item corresponding to the element stress.

**LabT**

Labeled result item corresponding to the element temperature.

**TYPE**

Type of calculation:

**0 or 1**

Use safety factor (SF) calculation.

**2**

Use margin of safety (MS) calculation.

**3**

Use 1/SF calculation.

## Notes

Calculates safety factor (SF) or margin of safety (MS) as described for the **SFACT** command for any labeled result item (see **ETABLE** command) for the selected elements. Use the **PRETAB** or **PLETAB** command to display results. Allowable element stress is determined from the SALLOW-TALLOW table [**SALLOW, TALLOW**].

## Menu Paths

**Main Menu>General Postproc>Safety Factor>SF for ElemTable**



**SFCUM**, *Lab, Oper, FACT, FACT2*

**Specifies that surface loads are to be accumulated.**

SOLUTION: FE Surface Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

**Lab**

Valid surface load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
Substructure	SELV	load vector number
Electric	CHRG	surface charge density
	MXWF	Maxwell force flag
Magnetic	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for <b>IN-FIN110</b> and <b>INFIN111</b>

1. Thermal labels CONV and HFLUX are mutually exclusive.

**Oper**

Accumulation key:

**REPL**

Subsequent values replace the previous values (default).

**ADD**

Subsequent values are added to the previous values.

**IGNO**

Subsequent values are ignored.

**FACT**

Scale factor for the first surface load value. A (blank) or '0' entry defaults to 1.0.

**FACT2**

Scale factor for the second surface load value. A (blank) or '0' entry defaults to 1.0.

**Command Default**

Replace previous values.

**Notes**

Allows repeated surface loads (pressure, convection, etc.) to be replaced, added, or ignored. Surface loads are applied with the **SF**, **SFE**, and **SFBEAM** commands. Issue the **SFELIST** command to list the surface loads. The operations occur when the next surface load specifications are defined. For example, issuing the **SF** command with a pressure value of 25 after a previous **SF** command with a pressure value of 20 causes the

current value of that pressure to be 45 with the add operation, 25 with the replace operation, or 20 with the ignore operation. All new pressures applied with **SF** after the ignore operation will be ignored, even if no current pressure exists on that surface.

Scale factors are also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a pressure of 70. Scale factors are applied even if no previous values exist. Issue **SFCUM,STAT** to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

---

### Note

The FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

**SFCUM** does not work for tabular boundary conditions.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Surface Loads**  
**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Surface Loads**

---

**SFDELE**, *Nlist*, *Lab*

**Deletes surface loads.**

[SOLUTION: FE Surface Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

***Nlist***

Label defining where to find the list of nodes:

**ALL**

Use all selected nodes [**NSEL**]. If P, use graphical picking in GUI. A component label may be substituted for *Nlist*.

***Lab***

Valid surface load label. If ALL, use all appropriate labels. See the **SF** command for labels.

## Notes

Deletes surface loads as applied with the **SF** command. Loads are deleted only for the specified nodes on external faces of selected area and volume elements. For shell elements, if the specified nodes include face one (which is usually the bottom face) along with other faces (such as edges), only the loads on face one will be deleted. The element faces are determined from the list of selected nodes as described for the **SF** command. See the **SFDELE** command for deleting loads explicitly by element faces.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppImped\_E>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppShield>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelExtPort>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppMCI>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Node Components  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Surface Rad>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppImped\_E>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppShield>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelExtPort>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppMCI>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Node Components  
 Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Nodes  
 Main Menu>Solution>Define Loads>Delete>Thermal>Surface Rad>On Nodes

**SFE**, *ELEM*, *LKEY*, *Lab*, *KVAL*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

**Specifies surface loads on elements.**

[SOLUTION: FE Surface Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

***ELEM***

Element to which surface load applies. If ALL, apply load to all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

***LKEY***

Load key or face number associated with surface load (defaults to 1). Load keys (1,2,3, etc.) are listed under "Surface Loads" in the input data table for each element type in the [Element Reference](#).

***Lab***

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the [Element Reference](#).

Discipline	Surface Load Label	Label Description
Structural	PRES	pressure
	FREQ	frequency (harmonic analyses only)
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
	RAD	radiation
	RDSF	surface-to-surface radiation
Fluid	FSI[2]	fluid-structure interaction flag
	VFRC	volume fraction for VOF method
	IMPD	impedance boundary flag
Electric	CHRG	surface charge density
	MXWF	Maxwell force flag
Magnetic	MXWF	Maxwell force flag
High-frequency electromagnetic	PORT	waveguide port number
	SHLD	surface shielding properties
Field-surface interface	FSIN[3]	field-surface interface number
Poromechanics	FFLX	fluid flow flux
Infinite element	INF	exterior surface flag for <a href="#">INFIN110</a> and <a href="#">INFIN111</a>
Substructure	SELV	load vector number

1. Thermal labels CONV and HFLUX are mutually exclusive.

2. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the **FLUID129** or **FLUID130** elements.
3. For a fluid-solid interaction analysis, apply the field-surface interface flag (Label = FSIN) twice: once for the fluid side (**FLUID141** or **FLUID142** elements) and once for the solid side.

**KVAL**

Value key. If *Lab* = PRES:

**0 or 1**

*VAL1* through *VAL4* are used as real components of pressures.

**2**

*VAL1* through *VAL4* are used as imaginary components of pressures.

Value key. If *Lab* = CONV:

**0 or 1**

*VAL1* through *VAL4* are used as the film coefficients.

**2**

*VAL1* through *VAL4* are the bulk temperatures.

**3**

*VAL1* through *VAL4* are used as film effectiveness.

**4**

*VAL1* through *VAL4* are used as free stream temperature.

Value key. If *Lab* = SHLD:

**0 or 1**

*VAL1* through *VAL4* are used as the electrical conductivities.

**2**

*VAL1* through *VAL4* are used as the relative permeabilities.

Value key. If *Lab* = RAD:

**0 or 1**

*VAL1* through *VAL4* are used as the emissivities.

**2**

*VAL1* through *VAL4* are ambient temperatures.

Value key. If *Lab* = RDSF:

**0 or 1**

*VAL1* is the emissivity value between 0 and 1.

**2**

*VAL1* is the enclosure number.

Value key. If *Lab* = SELV:

**0 or 1**

*VAL1* is the multiplier on real load vector *LKEY*.

**2**

*VAL1* is the multiplier on imaginary load vector *LKEY*.

Value key. If  $Lab = VFRC$ :

**0 or 1**

$VAL1$  is the boundary load value.  $VAL1$  defaults to 1.

**2**

$VAL1$  is the wetting status.  $VAL1$  defaults to 0 or non-wetting.

If only one set of data is supplied (either emissivities or temperatures when  $Lab = RAD$ ; or either film coefficients or temperatures when  $Lab = CONV$ ; or either conductivity or relative permeability when  $Lab = SHLD$ ), the other set of data defaults to previously specified values (or zero if not previously specified).

**VAL1**

First surface load value (typically at the first node of the face) or the name of a table for specifying tabular boundary conditions. Face nodes are listed in the order given for "Surface Loads" in the input data table for each element type in the *Element Reference*. For example, for **SOLID185**, the item 1-JILK associates  $LKEY = 1$  (face 1) with nodes J, I, L, and K. Surface load value  $VAL1$  then applies to node J of face 1. To specify a table, enclose the table name in percent signs (%), e.g.,  $\%tablename\%$ . Use the **\*DIM** command to define a table. Only one table can be specified, and it must be specified in the  $VAL1$  position; tables specified in the  $VAL2$ ,  $VAL3$ , or  $VAL4$  positions will be ignored.  $VAL2$  applies to node I, etc.

If  $Lab = PRES$  and  $KVAL = 2$ ,  $VAL1$  is the imaginary pressure, which is used only by **SURF153**, **SURF154**, **SURF156**, or **SURF159** in full harmonic response analyses (**HROPT,FULL**), or by a mode superposition harmonic response analysis (**HROPT,MSUP**) if the mode-extraction method is Block Lanczos (**MODOPT,LANB**), PCG Lanczos (**MODOPT,LANPCG**), or Supernode (**MODOPT,SNODE**).

If  $Lab = CONV$ ,  $KVAL = 0$  or  $1$ , and  $VAL1 = -N$ , the film coefficient is assumed to be a function of temperature and is determined from the HF property table for material  $N$  [**MP**]. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user defined for some elements.

If  $Lab = PORT$ ,  $VAL1$  is a port number representing a waveguide port. The port number must be an integer between 1 and 6.

If  $Lab = RDSF$ ,  $KVAL = 0$  or  $1$ , and  $VAL1 = -N$ , the emissivity is assumed to be a function of the temperature, and is determined from the EMISS property table for material  $N$  (**MP**). The material  $N$  does not need to correlate with the underlying solid thermal elements. If  $Lab = RDSF$ ,  $KVAL = 2$ , and  $VAL1$  is negative, radiation direction is reversed and will occur inside the element for the flagged radiation surfaces. Negative value of enclosure number is applicable for **FLUID141** and **FLUID142** elements, to model radiation occurring between surfaces inside the fluid domain.

If  $Lab = VFRC$ ,  $VAL1$  is the boundary value or the wetting status, as defined by  $KVAL$  above. If  $Lab = VFRC$  and  $KVAL = 2$ , a  $VAL1$  setting of 1 indicates a wetted boundary. At a wetted boundary, the fluid upstream keeps the associated elements full.

If  $Lab = FSIN$  in a Multi-field solver (single or multiple code coupling) analysis,  $VAL1$  is the surface interface number.  $KVAL$  is not used. If  $Lab = FSIN$  in a unidirectional ANSYS to CFX analysis,  $VAL1$  is not used unless the ANSYS analysis is performed using the Multi-field solver.

**VAL2, VAL3, VAL4**

Surface load values at the 2nd, 3rd, and 4th nodes (if any) of the face. If all three values are blank, all three default to  $VAL1$ , giving a constant load. Zero or other blank values are used as zero. If a table is specified in  $VAL1$ , then any tables or numerical values specified in  $VAL2$ ,  $VAL3$ , or  $VAL4$  will be ignored.

If  $L_{ab} = \text{FSIN}$  in a unidirectional ANSYS to CFX analysis,  $VAL2$  is the surface interface number (not available from within the GUI).  $VAL3$  and  $VAL4$  are not used.

## Notes

Specifies surface loads on selected elements.

### Caution

You cannot use the **SFE** command with the **INFIN110** or **INFIN111** elements without prior knowledge of element face orientation, i.e., you must know which face is the exterior in order to flag it. Also, the surface effect elements, **SURF153** and **SURF154**, use **LKEY** to allow the use of tangential and other loads (see **SURF153** and **SURF154** of the *Element Reference* for more details).

Tapered loads may be applied over the faces of most elements.

For beam elements allowing lateral surface loads that may be offset from the nodes, use the **SFBEAM** command to specify the loads and offsets. See the **SF** command for an alternate surface load definition capability based upon node numbers. See the **SFGRAD** command for an alternate tapered load capability. Use the **SFCUM** command to accumulate (add) surface loads applied with **SFE**.

The **SFE** command can also be used to define fluid pressure penetration loads ( $L_{ab} = \text{PRES}$ ) at a contact interface. For this type of load,  $LKEY = 1$  is used to specify the pressure values and  $LKEY = 2$  is used to specify starting points and penetrating points. See [Applying Fluid Penetration Pressure](#) in the *Contact Technology Guide* for details on how to apply this type of load.

Film effectiveness and free stream temperatures specified with  $L_{ab} = \text{CONV}$  are only valid for **SURF151** and **SURF152**. Film effectiveness must be between 0 and 1 and it defaults to 0. If film effectiveness is applied, bulk temperature is ignored. When film effectiveness and free stream temperatures are specified, the commands to specify a surface load gradient (**SFGRAD**) or surface load accumulation (**SFCUM**) are not valid. For more information on film effectiveness, see [Conduction and Convection](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

You can specify a table name only when using structural (**PRES**) and thermal (**CONV** (film coefficient, bulk temperature, film effectiveness, and free stream temperature), **HFLUX**), boundary value and wetting status (**VFRC**), and surface emissivity and ambient temperature (**RAD**) surface load labels. The Volume of Fluid method ( $L_{ab} = \text{VFRC}$ ) is applicable only for **FLUID141**.

When a tabular function load is applied to an element, the load will not vary according to the positioning of the element in space.

This command is also valid in **PREP7**.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For Superelement**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Element Components**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Elements**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Elements>Tapered**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Elements>Uniform**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Elements**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Elements**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Surface Rad>On Elements**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Elements**  
**Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Elements**  
**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Elements**  
**Main Menu>Solution>Define Loads>Apply>Load Vector>For Superelement**  
**Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Element Components**  
**Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Elements**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Elements>Tapered**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Elements>Uniform**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Elements**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Elements**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Surface Rad>On Elements**

## SFEDELE, *ELEM*, *LKEY*, *Lab*

Deletes surface loads from elements.

[SOLUTION: FE Surface Loads](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### ***ELEM***

Element to which surface load deletion applies. If ALL, delete load from all selected elements [[ESEL](#)]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

### ***LKEY***

Load key associated with surface load (defaults to 1). If ALL, delete surface loads for all load keys.

### ***Lab***

Valid surface load label. If ALL, use all appropriate labels. See the [SFE](#) command for labels.

## Notes

Deletes surface loads from selected elements. See the [SFDELE](#) command for an alternate surface load deletion capability based upon selected nodes.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Surface Ld>On All Elems**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppImped\_E>On Elements**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Elements**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Elements**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Elements**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Elements**



Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Volume Fract>Bound Loads>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For Superelement  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppMCI>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Element Components  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Surface Rad>On Elements  
 Main Menu>Solution>Define Loads>Delete>All Load Data>All Surface Ld>On All Elems  
 Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppImped\_E>On Elements  
 Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Elements  
 Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Elements  
 Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Elements  
 Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Elements  
 Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Elements  
 Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Volume Fract>Bound Loads>On Elements  
 Main Menu>Solution>Define Loads>Delete>Load Vector>For Superelement  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Elements  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppMCI>On Elements  
 Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Elements  
 Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Element Components  
 Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Elements  
 Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Elements  
 Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Elements  
 Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Elements  
 Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Elements  
 Main Menu>Solution>Define Loads>Delete>Thermal>Surface Rad>On Elements

---

## SFELIST, *ELEM*, *Lab*

Lists the surface loads for elements.

SOLUTION: FE Surface Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### *ELEM*

Element at which surface load is to be listed. If ALL (or blank), list loads for all selected elements [ESEL]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

### *Lab*

Valid surface load label. If ALL (or blank), use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
	RAD	radiation
	RDSF	surface-to-surface radiation
Fluid	FSI[2]	fluid-structure interaction flag
	VFRC	volume fraction for VOF method
	IMPD	impedance boundary flag
Magnetic	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for <a href="#">INFIN110</a> and <a href="#">INFIN111</a>
Substructure	SELV	load vector number
Electric	CHRG	surface charge density
	MXWF	Maxwell force flag
High-frequency electromagnetic	PORT	waveguide port number
	SHLD	surface shielding properties

1. Thermal labels CONV and HFLUX are mutually exclusive.
2. If  $Lab = FSI$ , only the fluid elements must be selected for the flag to be applied.

## Notes

The surface loads listed correspond to the current database values. The database is not updated for surface loads in POST1. Surface loads specified in tabular form, however, do list their values corresponding to the current results set in POST1.

For [SURF151](#) or [SURF152](#) elements with an extra node for radiation and/or convection calculations (KEYOPT(5) = 1), the bulk temperature listed is the temperature of the extra node. If the thermal solution does not converge, the extra node temperature is not available for listing.

Film effectiveness and free stream temperatures specified by the **SFE** command ( $Lab = CONV$ ) can only be listed by this command. The command lists film coefficients and bulk temperatures first and then film effectiveness and free stream temperatures below those values.

This command is valid in any processor.

## Menu Paths

**Utility Menu>List>Loads>Surface Loads>On All Elements**  
**Utility Menu>List>Loads>Surface Loads>On Picked Elems**

**SFFUN**, *Lab*, *Par*, *Par2*

**Specifies a varying surface load.**

SOLUTION: FE Surface Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

**Lab**

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *Element Reference*. Issue **SFFUN,STATUS** to list current command settings.

Discipline	Body Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
Electric	CHRG	surface charge density

1. Thermal labels CONV and HFLUX are mutually exclusive.

**Par**

Parameter containing list of surface load values. If *Lab* = CONV, values are typically the film coefficients and *Par2* values (below) are typically the bulk temperatures.

**Par2**

Parameter containing list of second surface load values (if any). If *Lab* = CONV, the *Par2* values are typically the bulk temperatures. *Par2* is not used for other surface load labels.

**Notes**

Specifies a surface load "function" to be used when the **SF** or **SFE** command is issued. The function is supplied through an array parameter vector which contains nodal surface load values. Node numbers are implied from the sequential location in the array parameter. For example, a value in location 11 applies to node 11. The element faces are determined from the implied list of nodes when the **SF** or **SFE** command is issued. Zero values should be supplied for nodes that have no load. A tapered load value may be applied over the element face. These loads are in addition to any loads that are also specified with the **SF** or **SFE** commands. Issue **SFFUN** (with blank remaining fields) to remove this specification. Issue **SFFUN,STATUS** to list current settings.

Starting array element numbers must be defined for each array parameter vector. For example, **SFFUN,CONV,A(1,1),A(1,2)** reads the first and second columns of array A (starting with the first array element of each column) and associates the values with the nodes. Operations continue on successive column array elements until the end of the column. Another example to show the order of the commands:

```
*dim,nodepres,array,2
nodepres(1)=11,12
/prep7
et,1,42
n,1
n,2,1
n,3,1,1
n,4,,1
e,1,2,3,4
sfe,1,1,pres,1,3
sfelist ! expected answer: 3 at both nodes 1 and 2
sfedel,all,pres,all
```

```
sffun,pres, nodepres(1)
sfe,1,1,pres,1,5
sfelist ! expected answer: 5+11=16 at node 1, 5+12=17 at node 2
fini
```

**SFFUN** does not work for tabular boundary conditions.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Settings>For Surface Ld>Node Function**  
**Main Menu>Solution>Define Loads>Settings>For Surface Ld>Node Function**

## SFGRAD, Lab, SLKCN, Sldir, SLZER, SLOPE

**Specifies a gradient (slope) for surface loads.**

SOLUTION: FE Surface Loads

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the [Element Reference](#).

Discipline	Body Load Label	Label Description
Structural	PRES	pressure (real pressure only)
Thermal	CONV[1]	convection (bulk temperatures only)
	HFLUX[1]	heat flux
Electric	CHRG	surface charge density

1. Thermal labels CONV and HFLUX are mutually exclusive.

### SLKCN

Reference number of slope coordinate system (used with *Sldir* and *SLZER* to determine COORD). Defaults to 0 (the global Cartesian coordinate system).

### Sldir

Slope direction in coordinate system *SLKCN*:

#### X

Slope is along X direction (default). Interpreted as R direction for non-Cartesian coordinate systems.

#### Y

Slope is along Y direction. Interpreted as  $\theta$  direction for non-Cartesian coordinate systems.

#### Z

Slope is along Z direction. Interpreted as  $\Phi$  direction for spherical or toroidal coordinate systems.

### SLZER

Coordinate location (degrees for angular input) where slope contribution is zero (CVALUE = VALUE). Allows the slope contribution to be shifted along the slope direction. For angular input, *SLZER* should be

between  $\pm 180^\circ$  if the singularity [**CSCIR**] is at  $180^\circ$  and should be between  $0^\circ$  and  $360^\circ$  if the singularity is at  $0^\circ$ .

### **SLOPE**

Slope value (load per unit length or per degree).

## Notes

Specifies a gradient (slope) for surface loads. All surface loads issued with the **SF**, **SFE**, **SFL**, or **SFA** commands while this specification is active will have this gradient applied (for complex pressures, only the real component will be affected; for convections, only the bulk temperature will be affected). The load value, CVALUE, calculated at each node is:

$$CVALUE = VALUE + (SLOPE \times (COORD - SLZER))$$

where VALUE is the load value specified on the subsequent **SF**, **SFE**, **SFL**, or **SFA** commands and COORD is the coordinate value (in the *Sldir* direction of coordinate system *SLKCN*) of the node. Only one **SFGRAD** specification may be active at a time (repeated use of this command replaces the previous specification with the new specification). Issue **SFGRAD** (with blank fields) to remove the specification. Issue **SFGRAD,STAT** to show the current command status. The **SFGRAD** specification (if active) is removed when the **LSREAD** (if any) command is issued.

**SFGRAD** does not work for tabular boundary conditions.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Settings>For Surface Ld>Gradient**  
**Main Menu>Solution>Define Loads>Settings>For Surface Ld>Gradient**

**SFL**, *LINE*, *Lab*, *VALI*, *VALJ*, *VAL2I*, *VAL2J*

**Specifies surface loads on lines of an area.**

**SOLUTION: Solid Surface Loads**

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### **LINE**

Line to which surface load applies. If ALL, apply load to all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *LINE*.

### **Lab**

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *Element Reference*.

Discipline	Body Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[1]	convection

Discipline	Body Load Label	Label Description
	HFLUX[1]	heat flux
	RAD	radiation
	RDSF	surface-to-surface radiation
Fluid	FSI[2]	fluid-structure interaction flag
	VFRC	volume fraction for VOF method
	IMPD	impedance boundary flag
Magnetic	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for <a href="#">INFIN110</a> and <a href="#">INFIN111</a>
Electric	CHRG	surface charge density
	MXWF	Maxwell electrostatic force flag
Field-surface interface	FSIN[3]	field-surface interface number

1. Thermal labels CONV and HFLUX are mutually exclusive.
2. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the [FLUID129](#) or [FLUID130](#) elements.
3. For a fluid-solid interaction analysis, apply the field-surface interface flag (Label = FSIN) twice: once for the fluid side ([FLUID141](#) or [FLUID142](#) elements) and once for the solid side.

#### **VALI, VALJ**

Surface load values at the first keypoint (*VALI*) and at the second keypoint (*VALJ*) of the line, or table name for specifying tabular boundary conditions. If *VALJ* is blank, it defaults to *VALI*. If *VALJ* is zero, a zero is used. If *Lab* = CONV, *VALI* and *VALJ* are the film coefficients and *VAL2I* and *VAL2J* are the bulk temperatures. To specify a table, enclose the table name in percent signs (%), e.g., %*tablename*%. Use the **\*DIM** command to define a table. If *Lab* = CONV and *VALI* = -*N*, the film coefficient may be a function of temperature and is determined from the HF property table for material *N* [**MP**]. If *Lab* = RAD, *VALI* and *VALJ* values are surface emissivities and *VAL2I* and *VAL2J* are ambient temperatures. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user defined for some elements. If *Lab* = RDSF, *VALI* is the emissivity value; the following condition apply: If *VALI* = -*N*, the emissivity may be a function of the temperature and is determined from the EMISS property table for material *N* [**MP**]. If *Lab* = VFRC, *VALI* is the boundary value (defaults to 1). If *Lab* = FSIN in a Multi-field solver (single or multiple code coupling) analysis, *VALI* is the surface interface number. If *Lab* = FSIN in a unidirectional ANSYS to CFX analysis, *VALJ* is the surface interface number (not available from within the GUI) and *VALI* is not used unless the ANSYS analysis is performed using the Multi-field solver.

#### **VAL2I, VAL2J**

Second surface load values (if any). If *Lab* = CONV, *VAL2I* and *VAL2J* are the bulk temperatures. If *Lab* = RAD, *VAL2I* and *VAL2J* are the ambient temperatures. If *Lab* = RDSF, *VAL2I* is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open, radiation will occur to the ambient. Negative value of enclosure number is applicable for [FLUID141](#) and [FLUID142](#) elements, to model radiation occurring between surfaces inside the fluid domain. *VAL2I* and

$VAL2J$  are not used for other surface load labels. If  $VAL2J$  is blank, it defaults to  $VAL2I$ . If  $VAL2J$  is zero, a zero is used. To specify a table ( $Lab = CONV$ ), enclose the table name in percent signs (%), e.g., %*tablename*%. Use the **\*DIM** command to define a table. If  $Lab = VFRC$ ,  $VAL2I$  is the wetting status value (defaults to 0 or non-wetting). If  $Lab = VFRC$ , a  $VAL2I$  setting of 1 indicates a wetted boundary. At a wetted boundary, the fluid upstream keeps the associated elements full.

## Notes

Specifies surface loads on the selected lines of area regions. The lines represent either the edges of area elements or axisymmetric shell elements themselves. Surface loads may be transferred from lines to elements with the **SFTRAN** or **SBCTRAN** commands. See the **SFE** command for a description of surface loads. Loads input on this command may be tapered. See the **SFGRAD** command for an alternate tapered load capability.

You can specify a table name only when using structural (PRES) and thermal (CONV (film coefficient and/or bulk temperature), HFLUX), boundary value and wetting status (VFRC), and surface emissivity and ambient temperature (RAD) surface load labels.  $VALJ$  and  $VAL2J$  are ignored for tabular boundary conditions.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppInfinite>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppMaxwell>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Surface Rad>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Lines**  
**Main Menu>Solution>Define Loads>Apply>Thermal>Surface Rad>On Lines**



**SFLDELE**, *LINE*, *Lab*

Deletes surface loads from lines.

[SOLUTION: Solid Surface Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

***LINE***

Line to which surface load deletion applies. If ALL, delete load from all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *LINE*.

***Lab***

Valid surface load label. If ALL, use all appropriate labels. See the **SFL** command for labels.

**Notes**

Deletes surface loads (and all corresponding finite element loads) from selected lines.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Surface Ld>On All Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Volume Fract>Bound Loads>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Lines**  
**Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Surface Rad>On Lines**  
**Main Menu>Solution>Define Loads>Delete>All Load Data>All Surface Ld>On All Lines**  
**Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Volume Fract>Bound Loads>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Lines**



**Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Lines**  
**Main Menu>Solution>Define Loads>Delete>Thermal>Surface Rad>On Lines**

---

**SFLEX**, *FFAX*, *FFBY*, *FFBZ*, *FFTO*, *FFTSY*, *FFTSZ*

**Sets flexibility factors for the currently defined pipe element section.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***FFAX***

Factor to increase axial flexibility. The default value is 1.0.

***FFBY***

Factor to increase bending flexibility about element y axis (bending in the element x-z plane). The default value is 1.0.

***FFBZ***

Factor to increase bending flexibility about element z axis (bending in the element x-y plane). The default value is *FFBY*.

***FFTO***

Factor to increase torsional flexibility. The default value is 1.0.

***FFTSY***

Factor to increase transverse shear flexibility in the element x-z plane. The default value is 1.0.

***FFTSZ***

Factor to increase transverse shear flexibility in the element x-y plane. The default value is *FFTSY*.

## Notes

The **SFLEX** command sets section-flexibility factors for sections used by [current-technology pipe elements](#).

To increase *stiffness*, use a flexibility factor of less than 1.0.

The *FFBY* and *FFTSY* arguments affect motion in the element x-z plane, and the *FFBZ* and *FFTSZ* arguments affect motion in the element x-y plane. For stout pipe structures with low slenderness ratios, set *both* *FFBY* and *FFTSY*--and/or *both* *FFBZ* and *FFTSZ* (the related bending and transverse shear factors)--to the same value to obtain the expected flexibility effect.

When issued, the **SFLEX** command applies to the pipe section most recently defined via the **SECTYPE** command.

**SFLEX** is valid only for linear material properties and small strain analyses. The command does not support offsets, temperature loading, or initial state loading.

## Menu Paths

**Main Menu>Preprocessor>Sections>Pipe>Flexibility Factors**

---

**SFLIST**, *NODE*, *Lab***Lists surface loads.**

SOLUTION: FE Surface Loads

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH &lt;&gt; PP &lt;&gt; EME MFS

**NODE**

Node at which surface load is to be listed. If ALL (or blank), list for all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NODE*.

**Lab**

Valid surface load label. If ALL (or blank), use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV	convection
	HFLUX	heat flux
	RAD	radiation
	RDSF	surface-to-surface radiation
Fluid	FSI[1]	fluid-structure interaction flag
	IMPD	impedance boundary flag
Magnetic	MXWF	Maxwell force flag
	MCI	magnetic circuit interface
Infinite element	INF	Exterior surface flag for <b>INFIN110</b> and <b>INFIN111</b>
Electric	CHRG	surface charge density
	MXWF	Maxwell force flag
High-frequency electromagnetic	PORT	waveguide port number
	SHLD	surface shielding properties

1. If *Lab* = FSI, only the fluid elements must be selected for the flag to be applied.

**Notes**

Lists the surface loads as applied with the **SF** command. Loads are listed only for the specified nodes on external faces of selected area and volume elements. Use **SFELIST** for line elements. The surface loads listed correspond to the current database values. The database is not updated for surface loads in POST1. Surface loads specified in tabular form, however, do list their values corresponding to the current results set in POST1.

For [SURF151](#) or [SURF152](#) elements with an extra node for radiation and/or convection calculations (KEYOPT(5) = 1), the bulk temperature listed is the temperature of the extra node. If the thermal solution does not converge, the extra node temperature is not available for listing.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Nodes  
Utility Menu>List>Loads>Surface Loads>On Picked Nodes

### SFLLIST, *LINE*, *Lab*

**Lists the surface loads for lines.**

**SOLUTION:** Solid Surface Loads

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

#### *LINE*

Line at which surface load is to be listed. If ALL (or blank), list for all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *LINE*.

#### *Lab*

Valid surface load label. If ALL (or blank), use all appropriate labels. See the **SFL** command for labels.

## Notes

Lists the surface loads for the specified line.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Lines  
Utility Menu>List>Loads>Surface Loads>On Picked Lines

### SFSCALE, *Lab*, *FACT*, *FACT2*

**Scales surface loads on elements.**

**SOLUTION:** FE Surface Loads

MP ME ST PR PRN <> <> FL <> <> <> PP <> EME MFS

#### *Lab*

Valid surface load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	PRES	pressure

Discipline	Body Load Label	Label Description
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
Substructure	SELV	load vector number
Electric	CHRG	surface charge density

1. Thermal labels CONV and HFLUX are mutually exclusive.

**FACT**

Scale factor for the first surface load value. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

**FACT2**

Scale factor for the second surface load value. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

**Notes**

Scales surface loads (pressure, convection, etc.) in the database on the selected elements. Surface loads are applied with the **SF**, **SFE**, or **SFBEAM** commands. Issue the **SFELIST** command to list the surface loads. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

**Note**

Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

**SFSCALE** does not work for tabular boundary conditions.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Surface Loads**

**Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Surface Loads**

**SFTRAN**

**Transfer the solid model surface loads to the finite element model.**

[SOLUTION: Solid Surface Loads](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

**Notes**

Surface loads are transferred only from selected lines and areas to all selected elements. The **SFTRAN** operation is also done if the **SBCTAN** command is issued or automatically done upon initiation of the solution calculations [**SOLVE**].

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Surface Loads**

**Main Menu>Solution>Define Loads>Operate>Transfer to FE>Surface Loads**

**/SHADE**, *WN*, *Type*

**Defines the type of surface shading used with Z-buffering.**

[GRAPHICS:Style](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WN**

Window number (or ALL) to which command applies (defaults to 1).

**Type**

Shading type:

**FACET or 0**

Facet shading (one color per area face) (default).

**GOURAUD or 1**

Gouraud smooth shading (smooth variation of color based on interpolated vertex colors).

**PHONG or 2**

Phong smooth shading (smooth variation of color based on interpolated vertex normals).

## Command Default

Facet shading.

## Notes

Defines the type of surface shading used on area, volume, and PowerGraphics [[/GRAPHICS,POWER](#)] displays when software Z-buffering is enabled [[/TYPE](#)]. This command is only functional for 2-D display devices.

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Style>Hidden-Line Options**

## SHELL, *Loc*

Selects a shell element or shell layer location for results output.

POST1: Controls  
POST26: Controls

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Loc*

Location within shell element (or layer) to obtain stress results:

#### TOP

Top of shell element (or layer) (default).

#### MID

Middle of shell element (or layer). The default method averages the TOP and BOT values to obtain a mid value. Setting KEYOPT(8) = 2 for SHELL181, SHELL208, SHELL209, and ELBOW290 uses MID results obtained directly from the results file.

#### BOT

Bottom of shell element (or layer).

## Command Default

Shell element (or layer) top location.

## Notes

Selects the location within a shell element (or a shell layer) for results output (nodal stresses, strains, etc.). Applies to POST1 selects, sorts, and output [**NSEL**, **NSORT**, **PRNSOL**, **PLNSOL**, **PRPATH**, **PLPATH**, etc.], and is used for storage with the POST26 **ESOL** command. For example, **SHELL, TOP** causes item S of the POST1 **PRNSOL** command or the POST26 **ESOL** command to be the stresses at the top of the shell elements. For layered shell elements, use the **LAYER** (POST1) or **LAYERP26** (POST26) command to select the layer. The **SHELL** command does not apply to the layered thermal shell elements, SHELL131 and SHELL132. For PowerGraphics [**/GRAPHICS**, **POWER**], the **SHELL, MID** command affects both the printed output and the displayed results, while the **SHELL** (TOP or BOT) command prints and displays both the top and bottom layers simultaneously.

In POST26, the **ESOL** data stored is based on the active **SHELL** specification at the time the data is stored. To store data at various specifications (for example, stresses at the top and bottom locations), issue a **STORE** command before each new specification.

## Menu Paths

Main Menu>General Postproc>Options for Outp

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

Utility Menu>List>Results>Options

**/SHOW**, *Fname*, *Ext*, *VECT*, *NCPL*

**Specifies the device and other parameters for graphics displays.**

GRAPHICS: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***Fname***

Device name, filename, or keyword, as listed below:

#### **<device name>**

Any valid graphics display device name (e.g., X11, 3-D etc.). Defaults to X11 for most systems. See [Getting Started with Graphics](#) in the *Basic Analysis Guide* for details. A device name must be defined before activating the Graphical User Interface (GUI). Once the GUI is activated, the device name cannot be changed for that ANSYS session, except for switching between X11 and X11C.

#### **<filename>**

Name of graphics file to which graphics displays are to be diverted (248 characters maximum). Should not be the same as a valid device name or any other *Fname* option. Plots are written to the file *Filename.Ext* (or just *Filename*.*Ext* if *Ext* is left blank) in the working directory. This file can be appended only during the same session; reissuing the same filename in a new session will overwrite existing filenames from previous sessions. Although multiple filenames can be used within one session, only the last file created or accessed will be appended. Issuing /SHOW, CLOSE, or starting a new session will prevent access to any previously created files.

### **TERM**

Graphics displays are switched back to the last-specified device name.

### **CLOSE**

This option purges the graphics file buffer. The CLOSE option should be issued any time you are changing graphics devices or file output types during an ANSYS session. Graphics displays are switched back to the last-specified device name, and any open graphics files are closed. The CLOSE option is similar to the TERM option, however, with the CLOSE option, another process, such as the DISPLAY program, can access the data in the graphics file. The CLOSE option causes graphics file buffers to be flushed to the graphics file.

### **FILE**

Graphics displays are switched back to the last-specified file name.

### **OFF**

Graphics display requests are ignored.

### **(blank)**

If blank in interactive mode, graphics will be displayed on screen as requested by display commands (no file written); if blank in batch mode, graphics data will be written to *Jobname.GRPH*.

### **PSCR**

Creates PostScript graphic files that are named *Jobnamennn.eps*, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, *Jobname000.eps*, *Jobname001.eps*, *Jobname002.eps*, and so on. (See the **PSCR** command for options.) Ignores the *Ext* and *NCPL* fields.

### **HPGL**

Creates Hewlett-Packard Graphics Language files that are named *Jobnamennn.hpgl*, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, *Jobname000.hpgl*, *Jobname001.hpgl*, *Jobname002.hpgl*, and so on. (See the **HPGL** command for options.) Ignores the *Ext* and *NCPL* fields.

**HPGL2**

Creates Hewlett-Packard Graphics Language files that are named `Jobnamennn.hpGL`, where `nn` is a numeric value that is incremented by one as each additional file is created; that is, `Jobname000.hpGL`, `Jobname001.hpGL`, `Jobname02.hpGL`, and so on. The *HPGL2* files have enhanced color. (See the **HPGL** command for options.) Ignores the *Ext* field.

**JPEG**

Creates JPEG files that are named `Jobnamennn.jpg`, where `nnn` is a numeric value that is incremented by one as each additional file is created; that is, `Jobname000.jpg`, `Jobname001.jpg`, `Jobname002.jpg`, and so on. Ignores the *Ext* field.

**TIFF**

Creates tagged image format files that are named `Jobnamennn.tif`, where `nnn` is a numeric value that is incremented by one as each additional file is created; that is, `Jobname000.tif`, `Jobname001.tif`, `Jobname002.tif`, and so on. This value for the *Fname* argument ignores the *Ext* field. (See the **TIFF** command for options.)

**PNG**

Creates PNG (Portable Network Graphics) files that are named `Jobnamennn.png`, where `nnn` is a numeric value that is incremented by one as each additional file is created; that is, `Jobname000.png`, `Jobname001.png`, `Jobname002.png`, and so on. This value for the *Fname* argument ignores the *Ext* field. (See the **PNGR** command for options.)

**VRML**

Creates Virtual Reality Meta Language files named `Jobname000.wrl` that can be displayed on 3-D Internet web browsers. Ignores the *Ext* and *NCPL* fields.

**Ext**

Filename extension (8 character maximum).

**VECT**

Specifies raster or vector display mode. This affects area, volume, and element displays, as well as geometric results displays such as contour plots. See the **/DEVICE** command for an alternate way to toggle between raster and vector mode. Changing *VECT* also resets the **/TYPE** command to its default.

**0**

Raster display (color filled entities; default)

**1**

Vector display (outlined entities; i.e., "wireframe")

**NCPL**

Sets the number of color planes (4 to 8). Default is device-dependent. *NCPL* is not supported by all graphics devices.

## Command Default

For interactive runs, display is shown on the screen; for batch runs, display commands are ignored (graphics file not written).

## Notes

Specifies the device to be used for graphics displays, and specifies other graphics display parameters. Display may be shown at the time of generation (for interactive runs at a graphics display terminal) or diverted to a file for later processing with the **DISPLAY** program. Issue **/PSTATUS** for display status.



Batch runs do not have access to the fonts available on your system. The Courier and Helvetica font files used for JPEG, PNG and TIFF batch output are copyrighted by Adobe Systems Inc. and Digital Equipment Corp. Permission to use these trademarks is hereby granted only in association with the images described above. Batch run JPEG output is produced at the default quality index value of 75, unless specified otherwise.

Interactive displays will default to eight color planes ( $NCPL = 8$ ) for most monitors, while graph file output will default to eight color planes for VRML output, and four color planes for PSCR, HPGL, HPGL2, JPEG, PNG, TIFF and FILE33.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Device Options  
 Utility Menu>PlotCtrls>Redirect Plots>To GRPH File  
 Utility Menu>PlotCtrls>Redirect Plots>To HPGL File  
 Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File  
 Utility Menu>PlotCtrls>Redirect Plots>To PSCR File  
 Utility Menu>PlotCtrls>Redirect Plots>To Screen

---

**/SHOWDISP**, *Dname*, --, --, *NCPL*

Defines the display driver name.

[DISPLAY: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Dname*

Valid driver name (see [Getting Started with Graphics](#) in the *Basic Analysis Guide* for details):

#### <*device name*>

Any linked terminal driver (such as X11, TEKTRONIX, etc.)

#### **HPGL**

Hewlett-Packard Graphics Language

#### **HPGL2**

Hewlett-Packard Graphics Language with enhanced color. (See the **HPGL** command for options.) Ignores the *NCPL* field.

#### **INTERLEAF**

Interleaf ASCII Format, OPS Version 5.0

#### **POSTSCRIPT**

PostScript, Version 1.0 Minimally Conforming

#### **DUMP**

ASCII Text Dump

--, --

Unused fields.

### *NCPL*

Number of color planes (4 to 8). Default is device-dependent.

## Menu Paths

It is part of the **DISPLAY** command.

**SHPP**, *Lab*, *VALUE1*, *VALUE2*

**Controls element shape checking.**

PREP7: Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lab*

Shape checking option. (When *Lab* = WARN, STATUS, SUMMARY, or DEFAULT, the remaining arguments are ignored.)

#### **ON**

Activates element shape checking. New elements, regardless of how they are created, are tested against existing warning and error limits. (The existing limits may be the default limits, or previously modified limits.) Elements that violate error limits produce *error* messages and either (a) cause a meshing failure, or (b) for element creation or storage other than **AMESH** or **VMESH**, *are not stored*. Elements that violate warning limits produce *warning* messages. If shape checking was previously turned off [**SHPP**,OFF] and you turn it on, existing elements are marked as untested; use the **CHECK** command to retest them. With this option, you may also specify a value for *VALUE1* to turn individual shape tests on. If you do not specify a value for *VALUE1*, all shape tests are turned on.

#### **WARN**

Activates element shape checking; however, in contrast to **SHPP**,ON, elements that violate error limits do not cause either a meshing or element storage failure. Instead, they produce *warning* messages to notify you that error limits have been violated. This option does not alter current shape parameter limits. Since the default shape parameter error limits are set to allow almost any usable element, the elements this option allows, which would otherwise be forbidden, are likely to be *very poorly shaped*.

#### **OFF**

Deactivates element shape checking. This setting does not alter current shape parameter limits. Use of this option is risky, since poorly shaped elements can lead to analysis results that are less accurate than would otherwise be expected for a given mesh density. With this option, you may also specify a value for *VALUE1* to turn individual shape tests off. If you do not specify a value for *VALUE1*, all element shape tests are turned off.

#### **SILENT**

Determines whether element shape checking runs in silent mode. In silent mode, ANSYS checks elements without issuing warnings, *with the exception of* the generic warnings that it issues at solution. With this option, you must also specify a value for *VALUE1* (During the execution of certain commands, ANSYS automatically runs element shape checking in silent mode, then internally summarizes the shape test results for all of the new or modified elements. ANSYS does this when it executes any of the following commands: **AGEN**, **AMESH**, **AREFINE**, **ARSYM**, **ATRAN**, **CDREAD**, **EGEN**, **ENGEN**, **ENSYM**, **ERead**, **EREFINE**, **ESYM**, **ET**, **FVMESH**, **KREFINE**, **LREFINE**, **NREFINE**, **TIMP**, **VEXT**, **VGEN**, **VIMP**, **VMESH**, **VOFFST**, **VROTAT**, **VSWEEP**, **VSymm**, and **VTRAN**.)

**STATUS**

Lists the shape parameter limits currently in effect, along with status information about element shape checking (for example, whether any individual shape tests are off, whether any of the shape parameter limits have been modified, and so on).

**SUMMARY**

Lists a summary of element shape test results for all selected elements.

**DEFAULT**

Resets element shape parameter limits to their default values. Also, if any individual tests were turned off, turns them back on. (The **SHPP,DEFAULT** command may be useful if any parameter limits were previously altered by using the **MODIFY** option.)

**OBJECT**

Determines whether element shape test results data is *stored in memory*. When this option is turned on, an "object" is created for storing test results in memory. When this option is turned off, no object is created and no data is stored; thus, any operation that requires shape parameters for an existing element (such as use of the **CHECK** command) causes the shape parameters to be recomputed. (Note the distinction between storing the data in memory and storing it in the database; regardless of whether this option is turned on or off, no element shape test results data will be stored in the database. The element shape parameter object is deleted automatically before any solution.) This setting is independent of shape checking status, with one exception—if shape checking is turned off [**SHPP,OFF**], the object is not created. Keep in mind that recomputing shape parameters is more computationally expensive than retrieving them from the object. With this option, you must also specify a value for the *VALUE1* argument; the *VALUE2* argument is ignored.

**LSTET**

Determines, for Jacobian ratio tests, whether sampling is done at integration points (DesignSpace product method), or at corner nodes. When this option is turned on, sampling is done at integration points, and the default limits for h-element Jacobian ratios are a warning tolerance of 10 and an error tolerance of 40. When this option is turned off, sampling is done at corner nodes, and the corresponding default limits are a warning tolerance of 30 and an error tolerance of 1000. Sampling at the integration points (option on) results in a lower Jacobian ratio, but that ratio is also subjected to a more restrictive error limit. Some elements that have passed the integration point sampling criterion, have failed the corner mode sampling criterion. Because of this, use integration point sampling only for simple linear analyses. For other types of analyses (e.g., nonlinear, electromagnetic), use sampling at corner nodes, which is the more conservative approach. With this option, you must also specify a value for the *VALUE1* argument; the *VALUE2* argument is ignored.

**MODIFY**

Indicates that you want to respecify a shape parameter limit. With this option, you must also specify values for the *VALUE1* and *VALUE2* arguments.

**FLAT**

Determines the warning and error limits used to test elements that may exhibit nonzero/nonconstant Z coordinates. With this option, you must also specify values for the *VALUE1* and/or *VALUE2* arguments.

**VALUE1**

Valid for the **ON**, **OFF**, **SILENT**, **OBJECT**, **LSTET**, **MODIFY**, and **FLAT** options only. When *Lab* = **ON** or **OFF**, use *VALUE1* to individually control (that is, turn off or turn on) specific element shape tests. Thus, *VALUE1* can be **ANGD** (**SHELL28** corner angle deviation tests), **ASPECT** (aspect ratio tests), **PARAL** (deviation from parallelism of opposite edges tests), **MAXANG** (maximum corner angle tests), **JACRAT** (Jacobian ratio tests), **WARP** (warping factor tests), or **ALL** (all tests). When *Lab* = **SILENT**, *VALUE1* can be **ON** (to turn silent mode on) or **OFF** (to turn silent mode off). When *Lab* = **OBJECT**, *VALUE1* can be either 1,

YES, or ON to turn on storage of element shape test data (the default); or it can be 0, NO, or OFF to turn off storage of element shape test data (delete the data and recompute as necessary). When *Lab* = LSTET, *VALUE1* can be either 1, YES, or ON to choose Jacobian sampling at integration points; or it can be 0, NO, or OFF to choose Jacobian sampling at nodes (the default). When *Lab* = MODIFY, *VALUE1* is the numeric location (within the shape parameter limit array) of the shape parameter limit to be modified. Locations are identified in the element shape checking status listing [SHPP,STATUS]. For more information, see the examples in the **Notes** section. When *Lab* = FLAT, *VALUE1* is the warning limit for XY element constant Z sets performed at **CHECK** or **SOLVE**. The default is 1.0e-8.

#### VALUE2

Valid for the MODIFY and FLAT options only. When *Lab* = MODIFY, specifies the new limit for the shape parameter that is in the location indicated by the *VALUE1* argument. See the examples in the **Notes** section. When *Lab* = FLAT, *VALUE2* is the error limit. The default is 1.0e-2.

## Command Default

All shape checking tests are on [SHPP,ON,ALL] with default shape parameter limits. Silent mode is off. Memory object storage of element shape parameters is on.

## Notes

The following examples illustrate how to use the SHPP,MODIFY,VALUE1,VALUE2 command to respecify shape parameter limits. Assume that you issued the SHPP,STATUS command, and you received the output below:

```
ASPECT RATIO (EXCEPT FLOTRAN OR EMAG)
  QUAD OR TRIANGLE ELEMENT OR FACE
    WARNING TOLERANCE ( 1) =   20.00000
    ERROR TOLERANCE   ( 2) =  1000000.
      .
      .
      .
MAXIMUM CORNER ANGLE IN DEGREES (EXCEPT FLOTRAN OR EMAG)
  TRIANGLE ELEMENT OR FACE
    WARNING TOLERANCE (15) =   165.0000
    ERROR TOLERANCE   (16) =   179.9000
```

Notice that in the sample output, the *warning* tolerance for aspect ratios is set to 20. Now assume that you want to "loosen" this shape parameter limit so that it is less restrictive. To allow elements with aspect ratios of up to 500 without causing warning messages, you would issue this command:

```
SHPP,MODIFY,1,500
```

Also notice that each shape parameter's numeric location within the shape parameter limit array appears in the sample output within parentheses. For example, the numeric location of the aspect ratio shape parameter (for warning tolerance) is 1, which is why "1" is specified for the *VALUE1* argument in the example command above.

Now notice that the sample output indicates that any triangle element with an internal angle that is greater than 179.9 degrees will produce an *error* message. Suppose that you want to "tighten" this shape parameter limit, so that it is more restrictive. To cause any triangle or tetrahedron with an internal angle greater than 170 degrees to produce an error message, you would issue this command:

```
SHPP,MODIFY,16,170
```

The existence of badly shaped elements in a model may lead to certain computational errors that can cause your system to abort during ANSYS solution. Therefore, you run the risk of a system abort during solution any time that you turn element shape checking off entirely, run shape checking in warning-only mode, turn off individual shape checks, or loosen shape parameter limits.

Changing any shape parameter limit marks all existing elements as untested; use the **CHECK** command to retest them.

Since the shape parameter limit array was completely reorganized at ANSYS 5.4, you should revise any input files created prior to 5.4 that contain limit changes so that they reflect the reorganized data structure.

For more information about element shape checking, see [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide*.

This command is also valid for [rezoning](#).

## Menu Paths

**Main Menu>Preprocessor>Checking Ctrl>Shape Checking**

**Main Menu>Preprocessor>Checking Ctrl>Toggle Checks**

**Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Element Shape Checking>Shape Checking**

**Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Element Shape Checking>Toggle Checks**

---

## **/SHRINK**, *RATIO*

**Shrinks elements, lines, areas, and volumes for display clarity.**

[GRAPHICS: Scaling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***RATIO***

Shrinkage ratio (input as a decimal (0.0 to 0.5)). Defaults to 0.0 (no shrinkage). Values greater than 0.5 default to 0.1 (10% shrinkage).

## Command Default

Full size entities.

## Notes

Shrinks the elements, lines, areas, and volumes so that adjacent entities are separated for clarity. Portions of this command are not supported by PowerGraphics [[/GRAPHICS,POWER](#)].

If only the common lines of non-coplanar faces are drawn (as per the [/EDGE](#) command), then this command is ignored.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

---

**SLIST**, *SFIRST*, *SLAST*, *SINC*, *Details*, *Type*

**Summarizes the section properties for all defined sections in the current session.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### **SFIRST**

First section ID to be summarized. Defaults to first available section in the database.

### **SLAST**

Last section ID to be summarized. Defaults to last available section in the database.

### **SINC**

Increment of the section ID; defaults to 1.

### **Details**

Determines the content of the summarized information for beams and shells.

#### **BRIEF**

For beams, lists only the section integrated properties (such as *Area*, *Iyy*, and *Iyz*). This option is the default.

#### **FULL**

For beams, lists the section integrated properties, as well as the section nodal coordinates, section cell connectivity information, and section cell integration point coordinates. For shells, the section stiffness (membrane, bending, membrane-bending coupling and transverse shear) are printed.

The shell section stiffness listed considers elastic behavior of materials at reference temperature only. The elements that use the section data may alter the transverse shear stiffness based on slenderness considerations (in addition to the shear correction factors shown).

Section stiffness terms listed via the FULL option do not include section offsets. The ANSYS program considers section offsets during the solution phase of the analysis.

#### **GROUP**

If a section calls other sections, this option lists those sections too.

### **Type**

The section type. Valid arguments are ALL (the default) and the types available on the **SECTYPE** command.

## Notes

For sections integrating **FiberSIM** data (**SECTYPE**, SHELL, FIBERSIM), the **SLIST** command output displays the relevant layer information.

Because the actual section can vary from element to element, the information provided by the *Details* = FULL option is not available when using the **FiberSIM-ANSYS interface**.

By default, the command lists information concerning all sections; however, you can limit the output to only beam or pretension sections via the *Type* key.

## Sample Output

Following is sample output from the **SLIST,,,BRIEF** command for a rectangular beam section subtype (**SECTYPE,,BEAM,RECT**):

```
LIST SECTION ID SETS      1 TO      1 BY      1
```

```
SECTION ID NUMBER:      1
BEAM SECTION TYPE:      Rectangle
BEAM SECTION NAME IS:
BEAM SECTION DATA SUMMARY:
Area                    = 6.0000
Iyy                     = 4.5000
Iyz                     = 0.11281E-15
Izz                     = 2.0000
Warping Constant        = 0.23299
Torsion Constant        = 4.7330
Center of Gravity Y     = -0.30973E-16
Center of Gravity Z     = 0.15376E-15
Shear Center Y          = -0.22957E-13
Shear Center Z          = 0.31281E-13
```

Beam Section is offset to CENTROID of cross section

Following is sample output from the **SLIST** command for a **FiberSIM** subtype (**SECTYPE,,SHELL,FIBERSIM**):

```
LIST SECTION ID SETS      1 TO      4 BY      1
```

```
INPUT SECTION ID NUMBER      4
INPUT SECTION IS SHELL SECTION TYPE USING FIBERSIM DATA
INPUT SHELL SECTION NAME      lamin1
```

Thickness	MatID	Num Intg. Pts	Layer ID
0.2000	1	0	STEEL
1.0000	2	0	ALUM1
0.2000	3	0	ALUM2

This table is a summary of the SECDATA command input and describes the individual plies/layers and does not normally represent the actual layup used. Layup data is available in FiberSIM.

Shell Section is offset to BOTTOM surface of Shell

```
Section Solution Controls
Tolerance in Thickness Direction = 0.15500
Angle Tolerance                   = 15.000
Added Mass Per Unit Area          = 5.00000E-02
Elastic Foundation Stiffness      = 100.00
Parsing Debug                      = 1
Computed Real Constants           = 1
```

## Menu Paths

**Main Menu>Preprocessor>Sections>List Sections**

**SLOAD**, *SECID*, *PLNLAB*, *KINIT*, *KFD*, *FDVALUE*, *LSLOAD*, *LSLOCK***Load a pretension section.**

PREP7: Cross Sections

MP ME ST PR PRN DS DSS &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

**SECID**

Unique section number. The number must already be assigned to a pretension section.

**PLNLAB**

Label representing the pretension load sequence number in the format "PL $nn$ " where  $nn$  is an integer from 1 through 99 (for example, PL01 through PL99).

Specify a value of DELETE to delete all loads on the specified pretension section (*SECID*). In this case, the command ignores any other argument values.

**KINIT**

Initial action key for pretension load PL01. (This field is omitted for PL02 and up.) Three scenarios are possible:

**LOCK**

Constrains (connects) the cutting plane on the pretension section. This value is the default.

**SLID**

Unconstrains (disconnects) the cutting plane on the pretension section.

**TINY**

Applies a very small pretension load (0.1% of *FDVALUE*) before the desired load is established. The small load prevents convergence problems which can occur when the desired load is not established in the first load step. This value is valid only if *KFD* = FORC.

**KFD**

Force/Displacement key. Specifies whether *FDVALUE* is a force or a displacement:

**FORC**

Apply a force on the specified pretension section. This value is the default.

**DISP**

Apply a displacement (adjustment) on the specified pretension section.

**FDVALUE**

Pretension load value. If *KFD* = FORC, this value is a pretension force. If *KFD* = DISP, this value is a pretension displacement (adjustment).

**LSLOAD**

Load step in which to apply the *FDVALUE*.

**LSLOCK**

The load step in which the displacement value resulting from the pretension force is locked. This value is valid only if *KFD* = FORC.

**Command Default**

The default pretension load value *FDVALUE* is zero (no load). A positive value puts the pretension elements in tension.

No default exists for the *LSLOAD* applied load step value. You must specify the load step in which to apply the *FDVALUE*.



No default exists for the *LSLOCK* locked load step value. You must specify the load step in which to lock the *FDVALUE*.

## Notes

The **SLOAD** command applies pretension loads to specified pretension sections (created via the **PSMESH** command). A pretension load is ramp-applied (**KBC** = 0) if it is a force (*KFD* = FORC), and step-applied (**KBC** = 1) if it is a displacement (*KFD* = DISP).

You can “lock” the load value at a specified load step. When locked, the load changes from a force to a displacement, and ANSYS applies the load as a constant displacement in all future load steps. Locking is useful when applying additional loadings. The additional loadings alter the effect of the initial load value, but because locking transforms the load into a displacement, it preserves the initial load's effect.

### Example 1 Applying a Load

The following command shows how to establish loads on a pretension section:

```
SLOAD,1,PL01,TINY,FORC,5000,2,3
```

In this example, the load is applied to pretension section 1, and the sequence begins with the initial action key, *KINIT*, set to TINY. A small stabilization load (5 = 0.10% of 5000) is applied in the first load step, as the actual pretension force is not applied until the second load step. The next four fields set the actual load: the *KFD* value FORC specifies the type of load, *FDVALUE* defines the pretension load value (5000), *LSLOAD* specifies the load step in which the force is applied (2), and the *LSLOCK* field specifies the load step in which the force is locked (3). Additional sets of four fields can be used to define additional loads.

### Example 2 Editing an Existing Load

You can use the **SLOAD** command to edit (overwrite) existing loads on a pretension section. This example changes the load on pretension section 1 (set above) to 6000:

```
SLOAD,1,PL01,,,6000,2,3
```

Unspecified values (blank fields), as shown in this example, remain unchanged from prior settings. If no prior specifications exist, then default values (*KINIT* = LOCK and *KFD* = FORC) apply.

### Example 3 Deleting All Loads

The command can also delete all loads on a specified pretension section, as shown here:

```
SLOAD,1,DELETE
```

### Example 4 Locking a Pretension Element

For a prestressed modal analysis, this command locks the pretension element:

```
SLOAD,1,PL01,LOCK,DISP,0,1,2
```

## Multiple Loadings

The **SLOAD** command allows you to apply multiple loadings. You can add up to 15 loadings (PL01 through PL15), or delete loadings, for any given pretension section(s).

## Example 5 Applying Multiple Loadings

The following **SLOAD** commands, issued in the order shown, establish a pretension load sequence in pretension section 2 with a force of 25 in load step (LS) 2, locked in LS 3-6, a force of 50 in LS 7, locked in LS 8-11, a force of 75 in LS 12, locked in LS 13 and beyond:

```
SLOAD, 2, PL01, LOCK, FORC, 25, 2, 3
```

```
SLOAD, 2, PL02, , FORC, 50, 7, 8
```

```
SLOAD, 2, PL03, , FORC, 75, 12, 13
```

At the same time, you can issue SLOAD commands to apply loads on *other* pretension sections. For example, in addition to the commands listed above, you could issue the following command to apply a load on pretension section 3:

```
SLOAD, 3, PL01, LOCK, FORC, 25, 3, 4
```

## Using the Pretension Section Loads GUI

Any addition or deletion of a loading applies to the selected sections only. ANSYS does not apply or delete a load until you click on the **Apply** or **OK** button.

After you have successfully solved for a specified *LSLOAD* (GUI field **Apply at LS**) and eventually *LSLOCK* (GUI field **Lock at LS**) value, you cannot modify that loading's settings during subsequent steps of the analysis. Similarly, you cannot delete loadings that you have already partially or completely solved.

You can select more than one pretension section at a time in order to specify identical loadings on them. Before you completely solve a given loading, any combination of pretension sections is valid. The following limitations apply:

- After you have completely solved one or more loadings, ANSYS allows multiple selection of *only* those pretension sections having
  - the same number of defined loadings, *and*
  - the identical loading number from the most recent completely solved loading.
- A multiple selection meeting the necessary criteria retains the settings that are identical for all selected pretension sections and leaves all other fields blank.

## Menu Paths

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pretnsn Sectn**

**Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Section Lds**

**Main Menu>Solution>Define Loads>Apply>Structural>Pretnsn Sectn**

**Main Menu>Solution>Define Loads>Delete>All Load Data>All Section Lds**

---

**SMALL**, *IR, IA, IB, IC, Name, --, --, FACTA, FACTB, FACTC*

**Finds the smallest of three variables.**

[POST26: Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

***IA, IB, IC***

Reference numbers of the three variables to be operated on. If only two, leave *IC* blank. If only one, leave *IB* blank also.

***Name***

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

***--, --***

Unused fields.

***FACTA, FACTB, FACTC***

Scaling factors (positive or negative) applied to the corresponding variables (defaults to 1.0).

## Notes

Finds the smallest of three variables according to the operation:

$$IR = \text{smallest of } (FACTA \times IA, FACTB \times IB, FACTC \times IC)$$

## Menu Paths

**Main Menu>TimeHist Postpro>Math Operations>Find Minimum**

**\*SMAT**, *Matrix, Type, Method, Val1, Val2, Val3*

**Creates a sparse matrix.**

[APDL: Matrix Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

***Matrix***

Name used to identify the matrix. Must be specified.

***Type***

Matrix type:

**D --**

Double precision real values (default).

**Z --**

Complex double precision values.

**Method**

Method used to create the matrix:

**COPY --**

Copy an existing matrix.

**IMPORT --**

Import the matrix from a file.

**Val1, Val2, Val3**

Additional input. The meaning of *Val1* through *Val3* will vary depending on the specified *Method*. See details below.

**The following Valx fields are used with Method = COPY.**

**Val1**

Name of the matrix to copy (can be either a dense or a sparse matrix).

**Val2**

To copy only the diagonal, input the label below:

**DIAG --**

Copy only the diagonal of the matrix.

**The following table describes the Valx fields used with Method = IMPORT.**

<b>Method = IMPORT</b>			
<b>Val1</b>	<b>Val2</b>	<b>Val3</b>	<b>Description</b>
FULL	File name	Matrix type: STIFF - Stiffness (or conductivity) matrix MASS - Mass (or specific heat) matrix DAMP - Damping matrix NOD2BCS - Mapping vector relating the full set of nodal DOFs to the subset that the solver uses GMAT - Constraint equation matrix	Import a matrix from an existing FULL file. Currently, only real matrices can be extracted from FULL files.
HB-MAT	File name	File format: ASCII BINARY	Import a matrix from an existing Harwell-Boeing format file.
MMF	File name	(not used)	Import a matrix from an existing Matrix Market format file.
DMIG	File name	Separator character (default = blank)	Import a matrix from an existing NASTRAN DMIG file.
MAT	File name	(not used)	Restore from a previous <b>*EXPORT</b> ( <i>FORMAT = MAT</i> ) command..

## Notes

Use the **\*DMAT** command to create a dense matrix.

Unlike the **\*DMAT** command, the **\*SMAT** command cannot be used to allocate a sparse matrix.

For more information on the NOD2BCS mapping vector, see [Degree of Freedom Ordering](#) in the *ANSYS Parametric Design Language Guide*.

## Menu Paths

**This command cannot be accessed from a menu.**

**SMAX**, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*

**Forms an element table item from the maximum of two other items.**

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *LabR*

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

### *Lab1*

First labeled result item in operation.

### *Lab2*

Second labeled result item in operation (may be blank).

### *FACT1*

Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

### *FACT2*

Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

## Notes

Forms a labeled result item (see **ETABLE** command) for the selected elements by comparing two existing labeled result items according to the operation:

$$LabR = (FACT1 \times Lab1) \text{ cmx } (FACT2 \times Lab2)$$

where "cmx" means "compare and save maximum." If absolute values are requested [**SABS**,1], the absolute values of *Lab1* and *Lab2* are used.

## Menu Paths

**Main Menu>General Postproc>Element Table>Find Maximum**

## /SMBC, Mode

Controls the display of solid model boundary condition symbols and labels.

[DATABASE: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Mode

#### CENT

Solid model boundary condition symbols and labels appear at the centroid of the solid model entity (default).

#### TESS

Solid model boundary condition symbols and labels appear inside each constituent element of the tessellation.

## Notes

*Mode* = CENT is designed to reduce the clutter of boundary condition symbols in solid model plots. For example, if you have assigned normal pressure loads to an area, you may choose to display the pressures as arrows with the **/PSF** command using **/PSF,PRES,NORM,2**. When *Mode* = CENT, the pressure arrow is displayed at the centroid of the area. When *Mode* = TESS, a pressure arrow is displayed at the centroid of each polygon of the area's tessellation.

This command is valid in any processor.

## Menu Paths

This command cannot be accessed from a menu.

---

## SMBODY

Specifies "Body loads on the solid model" as the subsequent status topic.

[SOLUTION: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Solution>Body Loads

---

## SMCONS

Specifies "Constraints on the solid model" as the subsequent status topic.

SOLUTION: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This is a status [STAT] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

### Menu Paths

Utility Menu>List>Status>Solution>DOF Constraints

---

## SMFOR

Specifies "Forces on the solid model" as the subsequent status topic.

SOLUTION: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This is a status [STAT] topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

### Menu Paths

Utility Menu>List>Status>Solution>Forces

---

**SMIN**, LabR, Lab1, Lab2, FACT1, FACT2

Forms an element table item from the minimum of two other items.

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*LabR*

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

**Lab1**

First labeled result item in operation.

**Lab2**

Second labeled result item in operation (may be blank).

**FACT1**

Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

**FACT2**

Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

## Notes

Forms a labeled result item (see **ETABLE** command) for the selected elements by comparing two existing labeled result items according to the operation:

$$LabR = (FACT1 \times Lab1) \text{ cmn } (FACT2 \times Lab2)$$

where "cmn" means "compare and save minimum." If absolute values are requested [**SABS**,1], the absolute values of *Lab1* and *Lab2* are used.

## Menu Paths

**Main Menu>General Postproc>Element Table>Find Minimum**

---

**SMOOTH**, *Vect1*, *Vect2*, *DATAP*, *FITPT*, *Vect3*, *Vect4*, *DISP*

**Allows smoothing of noisy data and provides a graphical representation of the data.**

[POST26: Special Purpose](#)

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

**Vect1**

Name of the first vector that contains the noisy data set (i.e., independent variable). You must create and fill this vector before issuing **SMOOTH**.

**Vect2**

Name of the second vector that contains the dependent set of data. Must be the same length as the first vector. You must create and fill this vector before issuing **SMOOTH**.

**DATAP**

Number of data points to be fitted, starting from the beginning of the vector. If left blank, the entire vector will be fitted. The maximum number of data points is 100,000 (or greater, depending on the memory of the computer).

**FITPT**

Order of the fitting curve that will be used as a smooth representation of the data. This number should be less than or equal to the number of the data points. Default (blank) is one-half the number of data points. Maximum number of smoothed data fitting order is the number of data points up to 50. Depending on this number, the smoothed curve will be one of the following:

**1**

Curve is the absolute average of all of the data points.



**2**

Curve is the least square average of all of the data points.

**3 or more**

Curve is a polynomial of the order (n-1), where n is the number of data fitting order points.

***Vect3***

Name of the vector that contains the smoothed data of the independent variable. This vector should have a length equal to or greater than the number of smoothed data points. In batch (command) mode, you must create this vector before issuing the **SMOOTH** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector smth\_ind.

***Vect4***

Name of the vector that contains the smoothed data of the dependent variable. This vector must be the same length as *Vect3*. In batch (command) mode, you must create this vector before issuing the **SMOOTH** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector smth\_dep.

***DISP***

Specifies how you want to display data. No default; you must specify an option.

**1**

Unsmoothed data only

**2**

Smoothed data only

**3**

Both smoothed and unsmoothed data

**Notes**

You can control the attributes of the graph using standard ANSYS controls (**/GRID**, **/GTHK**, **/COLOR**, etc.). If working interactively, these controls appear in this dialog box for convenience, as well as in their standard dialog boxes. You must always create *Vect1* and *Vect2* (using **\*DIM**) and fill these vectors before smoothing the data. If you're working interactively, ANSYS automatically creates *Vect3* and *Vect4*, but if you're working in batch (command) mode, you must create *Vect3* and *Vect4* (using **\*DIM**) before issuing **SMOOTH**. *Vect3* and *Vect4* are then filled automatically by ANSYS. In addition, ANSYS creates an additional TABLE type array that contains the smoothed array and the unsmoothed data to allow for plotting later with **\*VPLT**. Column 1 in this table corresponds to *Vect1*, column 2 to *Vect2*, and column 3 to *Vect4*. This array is named *Vect3\_SMOOTH*, up to a limit of 32 characters. For example, if the array name is X1, the table name is X1\_SMOOTH.

This command is also valid in PREP7 and SOLUTION.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Smooth Data**

**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Smooth Data**

**Main Menu>Solution>Define Loads>Settings>Replace vs Add>Smooth Data**

**Main Menu>Solution>Loading Options>Smooth Data**

**Main Menu>TimeHist Postpro>Smooth Data**

**SMRTSIZE**, *SIZLVL*, *FAC*, *EXPND*, *TRANS*, *ANGL*, *ANGH*, *GRATIO*, *SMHLC*, *SMANC*, *MXITR*, *SPRX*

**Specifies meshing parameters for automatic (smart) element sizing.**

PREP7: Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **SIZLVL**

Overall element size level for meshing. The level value controls the fineness of the mesh. (Any input in this field causes remaining arguments to be ignored.) Valid inputs are:

***n***

Activate SmartSizing and set the size level to *n*. Must be an integer value from 1 (fine mesh) to 10 (coarse mesh). Remaining arguments are ignored, and argument values are set as shown in [Table 252: SMRTSIZE - Argument Values for h-elements \(p. 1583\)](#).

### **STAT**

List current **SMRTSIZE** settings.

### **DEFA**

Set all **SMRTSIZE** settings to default values (as shown in [Table 252: SMRTSIZE - Argument Values for h-elements \(p. 1583\)](#) for size level 6).

### **OFF**

Deactivate SmartSizing. Current settings of **DESIZE** will be used. To reactivate SmartSizing, issue **SMRTSIZE,*n***.

### **FAC**

Scaling factor applied to the computed default mesh sizing. Defaults to 1 for h-elements (size level 6), which is medium. Values from 0.2 to 5.0 are allowed.

### **EXPND**

Mesh expansion (or contraction) factor. (This factor is the same as **MOPT,EXPND,Value**.) *EXPND* is used to size internal elements in an area based on the size of the elements on the area's boundaries. For example, issuing **SMRTSIZE,,,2** before meshing an area will allow a mesh with elements that are approximately twice as large in the interior of an area as they are on the boundary. If *EXPND* is less than 1, a mesh with smaller elements on the interior of the area will be allowed. *EXPND* should be greater than 0.5 but less than 4. *EXPND* defaults to 1 for h-elements (size level 6), which does not allow expansion or contraction of internal element sizes (except when using **AESIZE** element sizing). (If *EXPND* is set to zero, the default value of 1 will be used.) The actual size of the internal elements will also depend on the *TRANS* option or upon **AESIZE** or **ESIZE** sizing, if used.

### **TRANS**

Mesh transition factor. (This factor is the same as **MOPT,TRANS,Value**.) *TRANS* is used to control how rapidly elements are permitted to change in size from the boundary to the interior of an area. *TRANS* defaults to 2.0 for h-elements (size level 6), which permits elements to approximately double in size from one element to the next as they approach the interior of the area. (If *TRANS* is set to zero, the default value will be used.) *TRANS* must be greater than 1 and, for best results, should be less than 4. The actual size of the internal elements will also depend on the *EXPND* option or upon **AESIZE** or **ESIZE** sizing, if used.

### **ANGL**

Maximum spanned angle per lower-order element for curved lines. Defaults to 22.5 degrees per element (size level 6). This angle limit may be exceeded if the mesher encounters a small feature (hole, fillet, etc.). (This value is *not* the same as that set by **DESIZE,,,ANGL**.)

**ANGH**

Maximum spanned angle per higher-order element for curved lines. Defaults to 30 degrees per element (size level 6). This angle limit may be exceeded if the mesher encounters a small feature (hole, fillet, etc.). (This value is NOT the same as that set by **DESIZE**,,,,,*ANGH*.)

**GRATIO**

Allowable growth ratio used for proximity checking. Defaults to 1.5 for h-elements (size level 6). Values from 1.2 to 5.0 are allowed; however, values from 1.5 to 2.0 are recommended.

**SMHLC**

Small hole coarsening key, can be ON (default for size level 6) or OFF. If ON, this feature suppresses curvature refinement that would result in very small element edges (i.e., refinement around small features).

**SMANC**

Small angle coarsening key, can be ON (default for all levels) or OFF. If ON, this feature restricts proximity refinement in areas where it is ill-advised (that is, in tight corners on areas, especially those that approach 0 degrees).

**MXITR**

Maximum number of sizing iterations (defaults to 4 for all levels).

**SPRX**

Surface proximity refinement key, can be off (*SPRX = 0*, which is the default for all levels) or on via two different values (*SPRX = 1* or *SPRX = 2*). If *SPRX = 1*, surface proximity refinement is performed and any shell elements that need to be modified are modified. If *SPRX=2*, surface proximity refinement is performed but no shell elements are altered.

## Command Default

SmartSizing is off and **DESIZE** is used for automatic element sizing.

## Notes

If a valid level number (1 (fine) to 10 (coarse)) is input on *SIZLV*, inputs for remaining arguments are ignored, and the argument values are set as shown in [Table 252: SMRTSIZE - Argument Values for h-elements](#) (p. 1583).

The settings shown are for h-elements. The first column contains *SIZLV* data, ranging from 10 (coarse) to 1 (fine). The default is 6 (indicated by the shaded row).

**Table 252 SMRTSIZE - Argument Values for h-elements**

	<b>FAC</b>	<b>EX-PND</b>	<b>TRANS</b>	<b>ANGL</b>	<b>ANGH</b>	<b>GRATIO</b>	<b>SMHLC</b>	<b>SMANC</b>	<b>MXITR</b>	<b>SPRX</b>
10	5.0	2.0	2.0*	45.0	45.0*	2.0	on	on	4*	off
9	3.0	1.75	2.0*	36.0	45.0*	1.9	on	on	4*	off
8	1.875	1.5	2.0*	30.0	45.0*	1.8	on	on	4*	off
7	1.5	1.0	2.0*	26.0	36.0*	1.7	on	on	4*	off
6	1.0*	1.0*	2.0*	22.5	30.0*	1.5*	on	on	4*	off
5	0.65	1.0*	2.0*	18.0	27.0	1.5	on	on	4*	off
4	0.4	1.0*	2.0*	15.0	22.0	1.5	off	on	4*	off
3	0.3	1.0*	2.0*	12.0	18.0	1.5	off	on	4*	off

	FAC	EX-PND	TRANS	ANGL	ANGH	GRATIO	SMHLC	SMANC	MXITR	SPRX
2	0.25	1.0*	2.0*	10.0	15.0	1.5	off	on	4*	off
1	0.2	1.0*	2.0*	7.5	15.0	1.4	off	on	4*	off

Where appropriate, SmartSizing will start with **AESIZE** settings. Elsewhere, it will start with any defined **ESIZE**,**SIZE** setting. It will locally override **AESIZE** or **ESIZE** for proximity and curvature. SmartSizing ignores any **ESIZE**,**NDIV** setting.

**LESIZE** line division and spacing specifications will be honored by SmartSizing, unless you give permission for SmartSizing to override them (for proximity or curvature) by setting KYNDIV to 1. Lines not having an **LESIZE** specification are meshed as well as they can be.

This command is also valid for [rezoning](#).

## Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Adv Opts

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Basic

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Status

---

## SMSURF

Specifies "Surface loads on the solid model" as the subsequent status topic.

[SOLUTION: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>Solution>Surface Loads

---

**SMULT**, *LabR, Lab1, Lab2, FACT1, FACT2*

Forms an element table item by multiplying two other items.

[POST1: Element Table](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*LabR*

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

**Lab1**

First labeled result item in operation.

**Lab2**

Second labeled result item in operation (may be blank).

**FACT1**

Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

**FACT2**

Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

## Notes

Forms a labeled result item (see **ETABLE** command) for the selected elements by multiplying two existing labeled result items according to the operation:

$$LabR = (FACT1 \times Lab1) \times (FACT2 \times Lab2)$$

May also be used to scale results for a single labeled result item. If absolute values are requested [**SABS**,1], the absolute values of *Lab1* and *Lab2* are used.

## Menu Paths

**Main Menu>General Postproc>Element Table>Multiply**

**SNOPTION**, *RangeFact*, *BlockSize*, *RobustLev*, *Compute*, --, *Solve\_Info*

**Specifies Supernode (SNODE) eigensolver options.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

**RangeFact**

Factor used to control the range of eigenvalues computed for each supernode. The value of *RangeFact* must be a number between 1.0 and 5.0. By default the *RangeFact* value is set to 2.0, which means that all eigenvalues between 0 and  $2 * FREQE$  are computed for each supernode (where *FREQE* is the upper end of the frequency range of interest as specified on the **MODOPT** command). As the *RangeFact* value increases, the eigensolution for the SNODE solver becomes more accurate and the computational time increases.

**BlockSize**

*BlockSize* to be used when computing the final eigenvectors. The value of *Blocksize* must be either MAX or a number between 1 and *NMODE*, where *NMODE* is the number of modes to be computed as set on the **MODOPT** command. Input a value of MAX to force the algorithm to allocate enough memory to hold all of the final eigenvectors in memory and, therefore, only read through the file containing the supernode eigenvectors once. Note that this setting is ONLY recommended when there is sufficient physical memory on the machine to safely hold all of the final eigenvectors in memory.

**RobustLev**

Parameter used to control the robustness of the SNODE eigensolver. The value of *RobustLev* must be a number between 0 and 10. Lower values of *RobustLev* allow the eigensolver to run in the most efficient manner for optimal performance. Higher values of *RobustLev* often slow down the performance

of the eigensolver, but can increase the robustness; this may be desirable if a problem is detected with the eigensolver or its eigensolution.

**Compute**

Key to control which computations are performed by the Supernode eigensolver:

**EVALUE**

The eigensolver computes only the eigenvalues.

**EVECTOR**

The eigensolver computes only the eigenvectors (must be preceded by a modal analysis where the eigenvalues were computed using the Supernode eigensolver).

**BOTH**

The eigensolver computes both the eigenvalues and eigenvectors in the same pass (default).

--

Unused field

**Solve\_Info**

Solver output option:

**OFF**

Turns off additional output printing from the Supernode eigensolver (default).

**PERFORMANCE**

Turns on additional output printing from the Supernode eigensolver, including a performance summary and a summary of file I/O for the Supernode eigensolver. Information on memory usage during assembly of the global matrices (that is, creation of the `Jobname.FULL` file) is also printed with this option.

## Command Default

*RangeFact* = 2.0. *BlockSize* is set to  $\min(NMODE, 40)$ , where *NMODE* is the number of modes to be computed as set on the **MODOPT** command. *RobustLev* = 0. *Compute* = BOTH. Additional output is not printed (*Solve\_Info* = OFF).

## Notes

This command specifies options for the Supernode (SNODE) eigensolver.

Setting *RangeFact* to a value greater than 2.0 will improve the accuracy of the computed eigenvalues and eigenvectors, but will often increase the computing time of the SNODE eigensolver. Conversely, setting *RangeFact* to a value less than 2.0 will deteriorate the accuracy of the computed eigenvalues and eigenvectors, but will often speedup the computing time of the SNODE eigensolver. The default value of 2.0 has been set as a good blend of accuracy and performance.

The SNODE eigensolver reads the eigenvectors and related information for each supernode from a file and uses that information to compute the final eigenvectors. For each eigenvalue/eigenvector requested by the user, the program must do one pass through the entire file that contains the supernode eigenvectors. By choosing a *BlockSize* value greater than 1, the program can compute *BlockSize* number of final eigenvectors for each pass through the file. Therefore, smaller values of *BlockSize* result in more I/O, and larger values of *BlockSize* result in less I/O. Larger values of *BlockSize* also result in significant additional memory usage, as *BlockSize* number of final eigenvectors must be stored in memory. The default *BlockSize* of  $\min(NMODE, 40)$  is normally a good choice to balance memory and I/O usage.

The *RobustLev* field should only be used when a problem is detected with the accuracy of the final solution or if the Supernode eigensolver fails while computing the eigenvalues/eigenvectors. Setting *RobustLev* to a value greater than 0 will cause the performance of the eigensolver to deteriorate. If the performance deteriorates too much or if the eigensolver continues to fail when setting the *RobustLev* field to higher values, then switching to another eigensolver such as Block Lanczos or PCG Lanczos is recommended.

Setting *Compute* = EVALUE causes the Supernode eigensolver to compute only the requested eigenvalues. During this process a *Jobname*.SNODE file is written; however, a *Jobname*.MODE file is not written. Thus, errors will likely occur in any downstream computations that require the .MODE file (for example, participation factor computations, mode superposition transient/harmonic analysis, PSD analysis). Setting *Compute* = EVECTOR causes the Supernode eigensolver to compute only the corresponding eigenvectors. The *Jobname*.SNODE file and the associated *Jobname*.FULL file are required when requesting these eigenvectors. In other words, the eigenvalues must have already been computed for this model before computing the eigenvectors. This field can be useful in order to separate the two steps (computing eigenvalues and computing eigenvectors).

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>Analysis Options**

---

## SOCEAN, OCID

**Specifies an ocean environment and associates it with an element section.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

### OCID

The ocean environment ID number.

## Notes

The **SOCEAN** command specifies an ocean environment and associates the specified ocean ID (*OCID*) to the element section most recently defined via the **SECTYPE** command. (See [Table 2.55: Ocean \(p. 38\)](#) for the commands used to define your ocean environment.)

Ocean loading applies to the [PIPE288](#) and [PIPE289](#) elements only.

## Menu Paths

**Main Menu>Preprocessor>Sections>Pipe>Add**  
**Main Menu>Preprocessor>Sections>Pipe>Edit**

---



**SOLCONTROL**, *Key1*, *Key2*, *Key3*, *Vtol*

**Specifies whether to use optimized nonlinear solution defaults and some enhanced internal solution algorithms.**

SOLUTION: Analysis Options

MP ME ST PR PRN DS DSS <> EM EH <> PP <> EME MFS

**Key1**

Optimized defaults activation key:

**ON or 1**

Activates optimized defaults for a set of commands applicable to nonlinear solutions. This is the default. The majority of solution command defaults are listed in the "Notes" (p. 1589) section below. See also the description of individual solution commands for default values.

**OFF or 0**

Restores defaults to pre-ANSYS 5.4 values (see the Default States table below). Internal solution algorithms work as for pre-ANSYS 5.4.

**Key2**

Check contact state key. This key is operable only when the optimized defaults are active (*Key1* = ON) and a contact or nonlinear status element is present in the model. When check contact state is active, ANSYS will base the time step size on the specifications of KEYOPT(7) for all contact elements. KEYOPT(7) for contact elements can be used to ensure that the time step interval accounts for changes in the contact status. Also, when *Key2* = ON, ANSYS ensures the time step is small enough to account for changes in nonlinear element status (applies to COMBIN14, COMBIN39, and COMBIN40 elements). Valid arguments for the key are:

**ON or 1**

Activate time step predictions based on specifications of element KEYOPT(7) or the nonlinear status of the element (applies to COMBIN14, COMBIN39, and COMBIN40 elements).

**OFF or 0**

Time step predictions not based on contact status or nonlinear element status (default).

**Key3**

Pressure load stiffness key. In general, use the default setting. Use a non-default setting only if you encounter convergence difficulties. Pressure load stiffness is automatically included when using eigenvalue buckling analyses (ANTYPE, BUCKLE), equivalent to *Key3* = INCP. For all other types of analyses, valid arguments for *Key3* are:

**NOPL**

Do not include pressure load stiffness for any elements.

**no entry (default)**

Include pressure load stiffness for elements SURF153, SURF154, SURF156, SURF159, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, BEAM188, BEAM189, FOLLW201, SHELL208, SHELL209, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290.

Do not include pressure load stiffness for elements SOLID65.

**INCP**

Include pressure load stiffness for elements SOLID65, SURF153, SURF154, SURF154, SURF156, SURF159, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, FOLLW201, SHELL208, SHELL209, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290.



**Vtol**

Tolerance for volumetric compatibility check in [current-technology](#) plane and solid elements (such as [PLANE182](#), [PLANE183](#), [SOLID185](#), [SOLID186](#), [SOLID187](#), [SOLSH190](#), [SOLID272](#), and [SOLID273](#)) when mixed u-P formulation is employed. When displacement convergence is not checked, the default value is  $1.0 \times 10^{-5}$ ; when displacement convergence is checked, the default value is  $1.0 \times 10^{-3}$ . *Vtol* can range from 0.0 to 1.0, but recommended values are between  $1.0 \times 10^{-5}$  and  $1.0 \times 10^{-2}$ . For more details, see [Volumetric Constraint Equations in u-P Formulations](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

**Notes**

The **SOLCONTROL** command is designed to provide reliable and efficient default solution settings for single-field full structural nonlinear or full transient analysis, or single-field thermal analysis. (It is not applicable for reduced transient analysis.) The optimized default settings and advanced internal solution algorithms can be used to solve the majority of structural/thermal, nonlinear/transient problems with the least amount of user intervention. The **SOLCONTROL** command is ON by default. In most cases, to successfully solve a problem the user only needs to:

- Choose **NLGEOM,ON** for large displacement/strain analysis.
- Choose **NROPT,UNSYM** to access unsymmetric pressure load stiffness, material stiffness, friction behavior, etc.
- Provide the initial step size through the **NSUBST** or **DELTIM** command.

To achieve this, the **SOLCONTROL** command chooses better default settings for a number of commands within ANSYS and uses optimized internal solution algorithms.

**THOPT** settings take precedence over **SOLCONTROL** settings.

Commands that ANSYS issues are written to files (such as log and load step files) as **COMMAND,-1** (for example, **KBC,-1**).

**Note**

The state of the **SOLCONTROL** command is not written by the **CDWRITE** or **LSWRITE** commands (so that the **CDWRITE** file does not rigidly define an individual solution command). Also note that switching **SOLCONTROL** ON and OFF between load steps is not recommended.

The **SOLCONTROL** command also serves as a "reset" command; when you issue **SOLCONTROL**, all the control commands set earlier in the interactive or input session are reset to their original default values.

Text database files (.CDB files created by **CDWRITE**) and load step files (.S01, .S02, .Snn, etc. files created by **LSWRITE**) should be handled with care when **SOLCONTROL,ON** (default). Files of these types sometimes contain control commands that you did not issue intentionally. These extra commands can overwrite the default settings specified by **SOLCONTROL**. To avoid overwriting the **SOLCONTROL** settings when you are using .CDB files, use the following procedure:

1. Read the .CDB files into ANSYS.
2. Enter the solution processor **[/SOLU]**.
3. Issue **SOLCONTROL,ON**.

4. Issue the desired control commands to overwrite the **SOLCONTROL** defaults as needed.

To use `.Snn` files properly, you should preview and edit them. Delete the unwanted solution commands before you execute the **LSSOLVE** command.

The following table lists the nonlinear solution parameters and algorithm differences when the command is active and inactive.

**Table 253 SOLCONTROL - Default States Table**

Command	Argument	SOLCONTROL ON	SOLCONTROL OFF
<b>General Options</b>			
<b>NSUBST</b>	NSBSTP	1 load step if contact elements <b>TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177</b> are not present; if these elements are present, 1 or 20 substeps, depending on the physics of the program	Previously specified value. If no specified value, defaults to 1.
	NSBMX	Determined by ANSYS	Previously specified value. If no specified value, defaults to NSBSTP.
	NSBMN	Determined by ANSYS	Previously specified value. If no specified value, defaults to 1.
	Carry	Determined by ANSYS	OFF
<b>DELTIM</b>	DTIME	1 time span of the load step if contact elements <b>TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177</b> are not present; if these elements are present, 1 or 1/20 of the time span of the load step, depending on the physics of the program	Previously specified value, if any.
	DTMIN	Determined by ANSYS	Previously specified value. If no specified value, defaults to DTIME
	DTMAX	Determined by ANSYS	Previously specified value. If no specified value, defaults to time span of load step.
	Carry	Determined by ANSYS	OFF
<b>KBC</b>	KEY	0 (ramped) for static nonlinear, structural and thermal (steady state) analyses, as well as for transient analyses when <b>TIMINT,OFF</b> . 1 (stepped) for transient structural and	0 for all types of transient or nonlinear analysis

Command	Argument	SOLCONTROL ON	SOLCONTROL OFF
		thermal analyses when <b>TIMINT,ON</b> . ( <b>TIMINT,ON</b> is the default for transient analyses.)	
<b>AUTOTS</b>	Key	Chosen by program	OFF
<b>EQSLV</b>		Uses sparse solver. If PCG solver is chosen, sets multiplier to 2.0 for Newton-Raphson iteration.	Uses sparse solver. If PCG solver is chosen, sets multiplier to 1.0 for Newton-Raphson iteration.
<b>CDWRITE</b> and <b>LSWRITE</b>		Does not write default values for most of the relevant solution control commands or options listed in this table.	Write all the default values for solution control commands.
<b>MONITOR</b>		Active	Not available
<b>Nonlinear Options</b>			
<b>CNVTOL</b>	TOLER	Force or moment convergence tolerance = 0.5% Displacement tolerance = 5%	Force or moment convergence tolerance = 0.1% Displacement tolerance not checked.
	MINREF	0.01 for force or moment; for heat flow and others the same as <b>SOLCONTROL,OFF</b>	For force or moment, 1.0 for heat flow, 1.0E-6 otherwise, 0
<b>NEQIT</b>	NEQIT	Between 15 and 26, depending on the physics of the problem.	25
<b>ARCLN</b>		A more aggressive scheme to open-up time step is used. A more stable <b>ARCLN</b> algorithm is used.	Use <b>ARCLN</b> as in Release 5.3.
<b>PRED</b>	Sskey	On, unless <b>SOLID65</b> is present.	OFF
<b>LNSRCH</b>	Key	Automatically turned ON when contact elements present.	OFF
<b>CUTCONTROL</b>	PLSLIMIT	15%	5%
	NPOINT	13	20
<b>OPNCONTROL</b>	TEMP	.01	Not Available
	NUM-STEP	3	Not Available
<b>SSTIF</b>	Key	ON for geometrically nonlinear analysis ( <b>NLGEOM, ON</b> ).	OFF
<b>NROPT</b>	ADPTKY	OFF, except: when frictional contact exists.	Automatically toggled on and off depending on whether plasticity or frictional contact exists or not.
<b>TINTP</b>	THETA	1.0	.5
	TOL	0.0	.2

Command	Argument	SOLCONTROL ON	SOLCONTROL OFF
<b>Algorithm Behavior</b>			
Deformed element shape (Jacobi) check used as criteria for early bisection		Active	Not available
Euler backward theta (for first order equations)		1.0 for thermal analysis.	0.0
Log file		Does not write default values for any of the relevant commands or options listed in this table.	Write all the default values for solution controls commands.
Moment reference values		Automatically corrected when values become too small by using a reaction force times an element characteristic length.	When zero <b>CONVTOL</b> , <b>MINREF</b> value is used.
Automatic time step scheme		Check on nonconvergent patterns. Time step is opened up less aggressively; the increase factor (used in calculating the degree in which the time step is opened) = 1.5 (in most cases). The calculation also takes into account the physics of the problem.	Check on nonconvergent patterns not implemented. Time step is opened up more aggressively; the increase factor = 2.0 (in most cases). No physics dependency involved.
Reset of all solution control defaults in one command.		<b>SOLCONTROL,ON</b> or <b>OFF</b>	Not available
Nonlinear convergence criterion		When force norm is smaller than 1, the calculated force value is still used as the REF value. If the calculated force value is approaching machine zero, the MINREF value is used as REF.	When force norm is smaller than 1, the MINREF value is used as the REF value.
Warning message printed when negative diagonal in matrix is discovered.		Simplified message, not printed in some cases.	Detailed message printed for each iteration.
Stop button		Available in GUI. Jobname . ABT file can also be used.	Use Jobname . ABT file to control normal abort.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Mixed U-P Toler**

**Main Menu>Preprocessor>Loads>Load Step Opts>Solution Ctrl**

**Main Menu>Solution>Load Step Opts>Nonlinear>Mixed U-P Toler**

**Main Menu>Solution>Load Step Opts>Solution Ctrl**

**SOLU**, *NVAR*, *Item*, *Comp*, *Name*

**Specifies solution summary data per substep to be stored.**

POST26: Set Up

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

***NVAR***

Arbitrary reference number assigned to this variable (2 to *NV* [**NUMVAR**]).

***Item***

Label identifying the item. Valid item labels are shown in the table below. Some items may also require a component label.

***Comp***

Component of the item (if required). Valid component labels are shown in the table below. None are currently required.

***Name***

Thirty-two character name identifying the item on printouts and displays. Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

## Notes

See also the **PRITER** command of POST1 to display some of these items directly. Valid for a static or full transient analysis. All other analyses have zeros for the data. Valid item and component labels for solution summary values are:

### Valid Item Labels

Item	Description
ALLF	Total arc-length load factor (ratio of the load at equilibrium to the total applied load)
ALDLF	Arc-length load factor increment (change in ALLF)
ARCL	Normalized arc-length radius
CNVG	Convergence indicator.
CRPRAT	Maximum creep ratio.
CSCV	Current segment convergence value.
CUCV	Current convergence value.
DICV	Displacement convergence value.
DSPRM	Descent parameter.
DTIME	Time step size.
EQIT	Number of equilibrium iterations.
FFCV	Fluid flow convergence value.
FOCV	Force convergence value.
HFCV	Heat flow convergence value.
NCMIT	Cumulative number of iterations.
NCMLS	Cumulative number of load steps
NCMSS	Cumulative number of substeps

### Valid Item Labels

Item	Description
MFCV	Magnetic flux convergence value.
MOCV	Moment convergence value.
MXDVL	Maximum degree of freedom value.
PRCV	Pressure convergence value.
PSINC	Maximum plastic strain increment.
RESFRQ	Response frequency for 2nd order systems.
RESEIG	Response eigenvalue for 1st order systems.
ROCV	Rotation convergence value.
SMCV	Scalar magnetic potential convergence value.
TECV	Temperature convergence value.
VECV	Velocity convergence value.
VOCV	Voltage convergence value.
VMCV	Vector magnetic potential convergence value.

### Menu Paths

**Main Menu>TimeHist Postpro>Define Variables**

**Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables**

---

### /SOLU

**Enters the solution processor.**

SESSION: Processor Entry

SOLUTION: Analysis Options

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This command is valid only at the Begin Level.

### Menu Paths

**Main Menu>Solution**

---

## SOLUOPT

Specifies "Solution options" as the subsequent status topic.

SOLUTION: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

### Menu Paths

Utility Menu>List>Status>Solution>General

---

## SOLVE, *Action*

Starts a solution.

SOLUTION: Analysis Options

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Action*

Action to be performed on solve (used only for linear perturbation analyses).

#### **ELFORM**

Reform all appropriate element matrices in the first phase of a linear perturbation analysis.

### Notes

Starts the solution of one load step of a solution sequence based on the current analysis type and option settings. Use *Action* = ELFORM only in the first phase of a [linear perturbation analysis](#).

### Menu Paths

Main Menu>DesignXplorer>Solution>Solve

Main Menu>Drop Test>Solve

Main Menu>Solution>Run FLOTRAN

Main Menu>Solution>Solve

Main Menu>Solution>Solve>Current LS

---

## SORT

Specifies "Sort settings" as the subsequent status topic.

[POST1: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status [\[STAT\]](#) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a [STAT](#) command, which will report the status for the specified topic.

If entered directly into the program, the [STAT](#) command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>General Postproc>Sort Module**

---

## SOURCE, X, Y, Z

Defines a default location for undefined nodes or keypoints.

[PREP7: Keypoints](#)

[PREP7: Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*x, y, z*

Global Cartesian coordinates for source nodes or keypoints (defaults to the origin).

## Command Default

Global Cartesian origin.

## Notes

Defines a global Cartesian location for undefined nodes or keypoints moved during intersection calculations [[MOVE](#) or [KMOVE](#)].

## Menu Paths

**This command cannot be accessed from a menu.**

---



**SPACE**, *NODE*

Defines a space node for radiation using the Radiation Matrix method.

[AUX12: Radiation Matrix Method](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**NODE**

Node defined to be the space node.

**Command Default**

No space node (no radiation to space).

**Notes**

A space node is required in an open system to account for radiation losses.

If using **SPACE** with the ANSYS Multi-field solver (MFS), you must capture this command in the command file using **MFCMMAND**. This step is necessary because at the end of each field computation, this command is unset.

**Menu Paths**

Main Menu>Radiation Opt>Matrix Method>Other Settings

**SPADP**, *FREQ*, *NUMADP*, *RMSSP*, *FACTB*, *FACTE*, *SLVOPT*, *SLVACC*

Automatically refines a **HF119** tetrahedral element mesh based on S-parameter convergence.

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> <> EH <> <> <> <>

**Freq**

Working frequency (Hz).

**NUMADP**

Maximum number of refinement iterations (0 to 10). Defaults to 0.

**RMSSP**

Root mean square error criteria for |S|. Defaults to 0.01.

**FACTB**

Beginning refinement percentage (0 to 100). Defaults to 10.

**FACTE**

Ending refinement percentage (0 to 100). Defaults to 10.

**SLVOPT**

Solver option:

**0**

SPARSE

**1**

QMR (symmetric matrices)

**2**

ICCG (unsymmetric matrices)

**SLVACC**

Convergence criteria for an iterative solver. Defaults to 1.0E-6.

**Notes**

The refinement factor varies linearly with the number of refinement iterations.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.**Menu Paths****This command cannot be accessed from a menu.**

---

**SPARM**, *PORTI*, *PORTJ*, *PAROPT***Calculates parameters between ports of a network system.**[POST1: Magnetics Calculations](#)

MP &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; EH &lt;&gt; PP &lt;&gt; &lt;&gt;

**PORTI**Port number of the excited port with a excitation mode. (See the description of the **HFPORT** command.)**PORTJ**Port number of the output port. This could be used for a multiport system. All ports but *PORTI* must be matched.**PAROPT**

Output parameter option:

**0**

S-parameters (default).

**1**

Reflection coefficient and voltage standing wave ratio.

**Notes****SPARM** calculates parameters for multiport or single port (*PORTI* = *PORTJ*) systems.If *PAROPT* = 0, the **SPARM** command macro returns the following complex S parameters: S<sub>II</sub>, dB<sub>SII</sub>, S<sub>IIPHR</sub>, S<sub>JJ</sub>, dB<sub>SJJ</sub>, and S<sub>JIPHR</sub>, where "I" is the port number for the excited port, "J" is the output port number and PHR is the phase angle.**Menu Paths****Main Menu>General Postproc>Elec&Mag Calc>Port>S-Parameters**

---

**SPCNOD**, *ENCL*, *NODE*

**Defines a space node for radiation using the Radiosity method.**

SOLUTION: Radiosity  
AUX12: Radiosity Solver

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

**ENCL**

Radiating surface enclosure number. Defaults to 1. If *ENCL* = STAT, the command lists all enclosure space nodes. If *ENCL* = DELE, the command deletes all enclosure space nodes.

**NODE**

Node defined to be the space node.

**Notes**

For open systems, an enclosure may radiate to a space node (*NODE*).

Open systems may be characterized by one or more enclosures (*ENCL*). Each enclosure may radiate to a different space node (*NODE*).

For a space node that is not part of the finite element model, specify the temperature using the **D** command. For the first load step, the space node temperature ramps from the uniform temperature specified by the **TUNIF** command to the temperature specified by the **D** command. For subsequent load steps, it ramps from the previous value of the space node temperature. For intermediate load steps, use the **SPCNOD,DELETE** command and specify the space node temperature again to ramp from the uniform temperature.

For a space node that is part of the finite element model, the temperature is that calculated during the finite element solution.

**Menu Paths**

**Main Menu>Preprocessor>Radiation Opts>Solution Opt**  
**Main Menu>Radiation Opt>Radiosity Meth>Solution Opt**  
**Main Menu>Solution>Radiation Opts>Solution Opt**

**SPCTEMP**, *ENCL*, *TEMP*

**Defines a free-space ambient temperature for radiation using the Radiosity method.**

SOLUTION: Radiosity  
AUX12: Radiosity Solver

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

**ENCL**

Radiating surface enclosure number. Defaults to 1. If *ENCL* = STAT, the command lists all enclosure space temperatures. If *ENCL* = DELE, the command deletes all enclosure space temperatures.

**TEMP**

Temperature of free-space in the reference temperature system. The temperature will be offset by the value specified in the **TOFFST** command for internal calculations.

## Notes

For open systems, an enclosure may radiate to the free-space ambient temperature (*TEMP*).

Open systems may be characterized by one or more enclosures (*ENCL*). Each enclosure may radiate to a different free-space ambient temperature (*TEMP*).

For the first load step, the space temperature ramps from the uniform temperature specified by the **TUNIF** command to the temperature specified by the **SPCTEMP** command. For subsequent load steps, it ramps from the previous value of the space temperature. For intermediate load steps, use the **SPCTEMP,DELETE** command and specify the space temperature again to ramp from the uniform temperature.

If using **SPCTEMP** with the ANSYS Multi-field solver (MFS), you must capture this command in the command file using **MFCMMAND**. This step is necessary because at the end of each field computation, this command is unset.

## Menu Paths

**Main Menu>Preprocessor>Radiation Opts>Solution Opt**  
**Main Menu>Radiation Opt>Radiosity Meth>Solution Opt**  
**Main Menu>Solution>Radiation Opts>Solution Opt**

---

**SPDAMP**, *TBLNO*, *CURVNO*, *DampRatio*

**Defines input spectrum damping in a multi-point response spectrum analysis.**

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *TBLNO*

Input table number. Corresponds to the frequency table number (*TBLNO* on the **SPFREQ** command).

### *CURVNO*

Input curve number. Corresponds to the spectrum values curve number (*CURVNO* on the **SPVAL** command).

### *DampRatio*

Damping ratio for the response spectrum curve. Up to 20 different curves may be defined, each with a different damping ratio. Damping values must be input in ascending order.

## Notes

Defines multi-point response spectrum damping value to be associated with:

- Previously defined frequency points (**SPFREQ**).
- Subsequently defined spectrum points (**SPVAL**).

Damping values are used only to identify input spectrum values for the mode coefficients calculation.

The curve number must be input in ascending order starting with 1.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Spect vs Freq

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Spect vs Freq

## SPEC

Specifies "Miscellaneous specifications" as the subsequent status topic.

POST1: Status

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status **[STAT]** topic command. Status topic commands are generated by the GUI and will appear in the log file (`JOBNAME.LOG`) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

Utility Menu>List>Status>General Postproc>Output Options

**SPFREQ**, *TBLNO*, *FREQ1*, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*

Defines the frequency points for the input spectrum tables **SPVAL** vs. **SPFREQ** for multi-point spectrum analysis.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

*TBLNO*

Input table number. Up to 20 tables may be defined.

*FREQ1*, *FREQ2*, *FREQ3*, ..., *FREQ7*

Frequency points (Hz) for spectrum vs. frequency tables. *FREQ1* should be greater than zero, and values must be in ascending order.

## Notes

The spectrum values are input with the **SPVAL** command. A separate **SPFREQ** command must be used for each table defined. Frequencies must be in ascending order.

Repeat **SPFREQ** command for additional frequency points. Values are added after the last nonzero frequency.

If all fields after **SPFREQ** are blank, all input vs. frequency tables are erased. If *TBLNO* is the only non-blank field, all corresponding **SPVAL** curves are erased.

Use the **SPTOPT** and **STAT** commands to list current frequency points.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

This command cannot be accessed from a menu.

---

**SPFSS**, *FREQB*, *FREQE*, *FREQINC*, *PHIB*, *PHIE*, *PHIINC*, *THETAB*, *THETAE*, *THETAINC*, *SWPOPT*, *EFACC*, *OUTPUT*, *FILEOPT*

**Performs frequency sweep or angle sweep calculations to determine reflection and transmission parameters of a frequency selective surface.**

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

### ***FREQB***

Frequency (Hz) at the beginning of the *FREQB* to *FREQE* range. If *FREQE* is blank, the solution is done only at frequency *FREQB*.

### ***FREQE***

Frequency at end of this range.

### ***FREQINC***

Frequency increment. The number of solutions performed is  $[(FREQE - FREQB) / FREQINC] + 1$ . Solutions are always performed at *FREQB* and *FREQE*.

### ***PHIB***

Angle  $\phi$  (Degree) of spherical coordinates at the beginning of the *PHIB* to *PHIE* range. If *PHIE* is blank, the solution is done only at angle *PHIB*.

### ***PHIE***

Angle  $\phi$  at end of this range.

### ***PHIINC***

$\phi$  angle increment. The number of solutions performed is  $[(PHIE - PHIB) / PHIINC] + 1$ . Solutions are always performed at *PHIB* and *PHIE*.

### ***THETAB***

Angle  $\theta$  (Degree) of spherical coordinates at the beginning of the *THETAB* to *THETAE* range. If *THETAE* is blank, the solution is done only at angle.

### ***THETAE***

Angle  $\theta$  at end of this range.

### ***THETAINC***

$\theta$  angle increment. The number of solutions performed is  $[(THETAE - THETAB) / THETAINC] + 1$ . Solutions are always performed at *THETAB* and *THETAE*.

### ***SWPOPT***

Solution Method (See **HROPT** command):

- 0 Variational Technology (default). See notes below.
- 1 Full method.
- 2 Variational Technology using a perfect absorber. See notes below.

**EFACC**

Electric field accuracy for fast exploration method. Defaults to 0.01.

**OUTPUT**

- 0 Create minimal size results file required to compute S-parameters for the last port excitation case, for all frequencies (default).
- 1 Create complete results file for the last port excitation case, for all frequencies.
- 2 Create complete results file for the last port excitation case, for the last frequency only. Valid only for full method.

**FILEOPT**

- 0 Output S-parameter magnitude and phase angle in Touchstone file.
- 1 Output S-parameter magnitude in dB and phase angle in Touchstone file.

**Notes**

Only one parameter among frequency,  $\phi$  angle, and  $\theta$  angle can be chosen to sweep each time. An output file `filename.fnn` ( $n = 1$  or  $2$ ) is generated after the sweep solution.

To use any Variational Technology method, you must have the separately licensed VT Accelerator. The Variational Technology Perfect Absorber method (Method = VTPA) is about 20% faster but slightly less accurate than the Variational Technology method (Method = VT).

The Variational Technology methods ( $SWPOPT = 0$  or  $2$ ) are only available for frequency sweep calculations.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**This command cannot be accessed from a menu.**

**SPGRAPH**, *TBLNO*, *CURVNO*, *CURVNOBeg*

**Displays input spectrum curves for MPRS analysis.**

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***TBLNO***

Table number to display. Defaults to 1.

***CURVNO***

Curve number to display. Defaults to none.

***CURVNOBeg***

Beginning of the curve number range to display. Defaults to 1.

**Notes**

You can display up to 10 input spectrum curves (**SPVAL** and **SPFREQ** commands) with log X scale.

If the input spectrum curves are not associated with a damping value (**SPDAMP** command), *CURVNO* and *CURVNOBeg* are not applicable and table *TBLNO* is displayed. Otherwise, specify *CURVNO* or *CURVNOBeg*:

- if *CURVNO* is used, one curve is displayed.
- if *CURVNOBeg* is used, up to 10 curves are displayed. *CURVNOBeg* is the beginning of the curve number range of interest.

**Menu Paths**

**This command cannot be accessed from a menu.**

**SPH4**, *XCENTER*, *YCENTER*, *RAD1*, *RAD2*

**Creates a spherical volume anywhere on the working plane.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***XCENTER*, *YCENTER***

Working plane X and Y coordinates of the center of the sphere.

***RAD1*, *RAD2***

Inner and outer radii (either order) of the sphere. A value of zero or blank for either *RAD1* or *RAD2* defines a solid sphere.

**Notes**

Defines either a solid or hollow spherical volume anywhere on the working plane. The sphere must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) A sphere of 360° will be defined with two areas, each consisting of a hemisphere. See the **SPHERE** and **SPHS** commands for other ways to create spheres.



When working with a model imported from an IGES file (DEFAULT import option), you can create only solid spheres. If you enter a value for both *RAD1* and *RAD2* the command is ignored.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Primitives>Solid Sphere**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>Hollow Sphere**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>Solid Sphere**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Sphere>Solid Sphere**

## SPH5, *XEDGE1*, *YEDGE1*, *XEDGE2*, *YEDGE2*

**Creates a spherical volume by diameter end points.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *XEDGE1*, *YEDGE1*

Working plane X and Y coordinates of one edge of the sphere.

### *XEDGE2*, *YEDGE2*

Working plane X and Y coordinates of the other edge of the sphere.

## Notes

Defines a solid spherical volume anywhere on the working plane by specifying diameter end points. The sphere must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) A sphere of 360° will be defined with two areas, each consisting of a hemisphere. See the [SPHERE](#) and [SPH4](#) commands for other ways to create spheres.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>By End Points**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Sphere>By End Points**

## SPHERE, *RAD1*, *RAD2*, *THETA1*, *THETA2*

**Creates a spherical volume centered about the working plane origin.**

[PREP7: Primitives](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *RAD1*, *RAD2*

Inner and outer radii (either order) of the sphere. A value of zero or blank for either *RAD1* or *RAD2* defines a solid sphere.

### *THETA1*, *THETA2*

Starting and ending angles (either order) of the sphere. Used for creating a spherical sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger

angle. The starting angle defaults to 0.0° and the ending angle defaults to 360.0°. See the [Modeling and Meshing Guide](#) for an illustration.

## Notes

Defines either a solid or hollow sphere or spherical sector centered about the working plane origin. The sphere must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) Inaccuracies can develop when the size of the object you create is much smaller than the relative coordinate system values (ratios near to or greater than 1000). If you require an exceptionally small sphere, create a larger object, and scale it down to the appropriate size.

For a solid sphere of 360°, you define it with two areas, each consisting of a hemisphere. See the [SPH4](#) and [SPH5](#) commands for the other ways to create spheres.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>By Dimensions**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Sphere>By Dimensions**

**SPICE**, *Fname*, *Ext*, *RMSERR*, *Z1*, *Z2*, *Z3*, *Z4*, *Z5*, *Z6*, *Z7*, *Z8*, *Z9*, *Z10*, *Z11*, *Z12*, *Z13*, *Z14*, *Z15*, *Z16*

**Generates a SPICE subcircuit model using S-parameters from a Touchstone file.**

POST1: Special Purpose

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path) of the Touchstone input file with s-parameters. An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

### *Ext*

Extension of Touchstone file (.snp where n is the number of ports).

### *RMSERR*

The convergence criteria (root-mean-square error) for the SPICE subcircuit model. Default determined by ANSYS.

### *Z1, Z2, Z3, ..., Z16*

Reference impedance for ports 1, 2, 3, ... 16. Default to 50 ohms.

## Notes

**SPICE** allows a maximum of 16 reference impedances.

The frequency range is determined by the frequency range of the Touchstone file.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

**SPLINE**, *P1, P2, P3, P4, P5, P6, XV1, YV1, ZV1, XV6, YV6, ZV6*

**Generates a segmented spline through a series of keypoints.**

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***P1, P2, P3, . . . , P6***

Keypoints through which the spline is fit. At least two must be defined. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

The following fields are used only if specified end slopes on the line are desired, otherwise zero curvature end slopes will be automatically calculated to produce a line which is "straight" in the active coordinate system. To specify end slopes, use the following fields to define a "slope vector" (one for each end of the line, if desired) that has its tail at the origin and its head at the point *XVn, YVn, ZVn* in the active coordinate system [[CSYS](#)]. The corresponding end slope of the line will then be parallel to this "slope vector."

***XV1, YV1, ZV1***

Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P1* end of the spline. The tail of the vector is at the origin of the coordinate system.

***XV6, YV6, ZV6***

Location of the head of the "slope vector" corresponding to the slope at the *P6* (or the last keypoint if fewer than six specified) end of the spline.

## Notes

The output from this command is a series of connected lines (one line between each pair of keypoints) that together form a spline. Note that solid modeling in a toroidal coordinate system is not recommended.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Lines>Splines>Segmented Spline**

**Main Menu>Preprocessor>Modeling>Create>Lines>Splines>With Options>Segmented Spline**

**SPLIT**, *NA1, NA2, NINC, MESH*

**Displays the selected areas and a faceted view of their underlying surfaces**

[PREP7:Areas](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NA1***

Starting area for display of areas and underlying surfaces. If *NA1* = ALL (default), *NA2* and *NINC* are ignored and all selected areas are displayed ([ASEL](#) command).

***NA2***

Last area to be displayed.

***NINC***

Numeric value setting steps between *NA1* and *NA2* for display. Default value is (1).

***MESH***

Specifies a rectangular mesh density used to display the underlying surface (default 4, i.e. 4 x 4).

## Notes

This command is valid in any processor. The plot output displays the external and internal trim curves and underlying surface. You cannot obtain a faceted view of your surface areas when you are using the **/EXPAND** command to create larger graphics displays.

Use **APLOT** for trimmed surface display.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**SPMWRITE**, *Method*, *NMODE*, *Inputs*, *InputLabels*, *Outputs*, *OutputLabels*, *NIC*, *VelAccKey*, *FileFormat*

**Calculates the state-space matrices and writes them to the SPM file.**

POST1: Special Purpose

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *Method*

Reduction method for the calculation of the state-space matrices.

#### **MODAL**

Method based on modal analysis results from LANB, LANPCG, or SNODE eigensolver (default).

### *NMODE*

Number of modes to be used. Defaults to all modes.

### *Inputs*

Definition of the inputs. Defaults to all load vectors on the MODE file.

If an integer is entered, it specifies the number of load vectors from the MODE file used for the definition of the inputs. The first *Inputs* load vectors are used.

If *Inputs* is an array parameter, the first column is the node number and the second column is the structural degree of freedom (1=UX, 2=UY, 3=UZ, 4=ROTX, 5=ROTY, 6=ROTZ) indicating input points. The number of rows in the array parameter is equal to the number of inputs.

### *InputLabels*

Definition of the input labels. Defaults to the load vector numbers or input definition (node and degree of freedom array parameter), depending on the *Inputs* specification.

If a character array parameter is entered (Type=CHAR in the **\*DIM** command), each 8 character string represents an input label. Only valid when *Inputs* is an array parameter

### *Outputs*

Definition of the outputs. Defaults to the inputs.

If an array parameter is entered, the first column is the node number and the second column is the structural degree of freedom (1=UX, 2=UY, 3=UZ, 4=ROTX, 5=ROTY, 6=ROTZ) of the output points. The number of rows in the array parameter is equal to the number of outputs.

### *OutputLabels*

Definition of the output labels. Defaults to the output definition (node and degree of freedom) if used, else defaults to the *InputLabels*.

If a character array parameter is entered (Type=CHAR in the \*DIM command), each 8 character string represents an output label.

**NIC**

Load vector on the *MODE* file used for the calculation of the initial conditions. Defaults to no initial condition.

**VelAccKey**

Output velocities and accelerations key.

**OFF**

Output displacements only (default).

**ON**

Output displacements, velocities and accelerations.

**FileFormat**

The format of the SPM file.

**0**

Dense format.

**1**

Matrix Market Exchange format (non-zero terms only).

**2**

Simplorer SML format without reference (default).

**3**

Simplorer SML format with common reference.

**4**

Simplorer SML format with independent references.

**Notes**

The SPMWRITE generates the file Jobname.SPM containing the state-space matrices and other information.

The following applies to the SML formats (FileFormat = 2, 3, and 4):

- For conservative systems where the outputs are equal to the inputs (*Outputs* is left blank):
  - The labels for the inputs (*InputLabels*) are required.
  - The *Inputs* must use the array parameter option so that the input degrees of freedom (DOFs) are known.
- For non-conservative systems where the outputs are not equal to the inputs:
  - The labels for the outputs (*OutputLabels*) are required.
  - The file formats with references (*FileFormat* = 3 and 4) do not apply.
- Velocity and acceleration results are not included in the state-space matrices calculation (VelAccKey = OFF)
- File format with common reference (FileFormat = 3) does not apply if the inputs are based on DOFs of a different nature. All input DOFs must be either all rotational or all translational and not a mix of the two.
- A graphics file (Jobname\_SPM.PNG) is generated. It contains an element plot of the model.

For more details about the reduction method and the generation of the state-space matrices, see [Reduced-Order Modeling for State-Space Matrices Export](#) in the Theory Reference.

For examples of the command usage, see "[State-Space Matrices Export](#)".

## Menu Paths

**This command cannot be accessed from a menu.**

**SPOINT**, *NODE*, *X*, *Y*, *Z*

**Defines a point for moment summations.**

POST1: Special Purpose

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

*NODE*

Node number of the desired point. If zero, use *X,Y,Z* to describe point.

*X*, *Y*, *Z*

Global Cartesian coordinates of the desired summation point. Used if *NODE* is 0. Defaults to (0,0,0).

## Command Default

No point is defined by default; you must either specify a node or coordinates.

## Notes

Defines a point (any point other than the origin) about which the tabular moment summations are computed [[NFORCE](#), [FSUM](#)]. If force summations are desired in other than the global Cartesian directions, a node number must be specified on the *NODE* field, *and* the desired coordinate system must be activated with [RSYS](#).

## Menu Paths

Main Menu>General Postproc>Nodal Calcs>Summation Pt>At Node

Main Menu>General Postproc>Nodal Calcs>Summation Pt>At XYZ Loc

**SPOINT**, *Sptype*, *NMODE*, *Elcalc*, *modReuseKey*

**Selects the spectrum type and other spectrum options.**

SOLUTION: Spectrum Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

Product Restrictions

*Sptype*

Spectrum type:

**SPRS**

Single point excitation response spectrum (default). See also the [SVTYP](#) command.

**MPRS**

Multiple point excitation response spectrum.

**DDAM**

Dynamic design analysis method.

**PSD**

Power spectral density.

**NMODE**

Use the first *NMODE* modes from the modal analysis. Defaults to all extracted modes, as specified by the **MODOPT** and **BUCOPT** commands. *NMODE* cannot be larger than 10000.

**Elcalc**

Element results calculation key (for *Sptype* = PSD only):

**NO**

Do not calculate element results and reaction forces (default).

**YES**

Calculate element results and reaction forces, as well as the nodal degree of freedom solution.

**modReuseKey**

Key for existing *MODE* file reuse when running multiple spectrum analyses:

**NO**

No spectrum analysis has been performed yet (default).

**YES**

This is not the first spectrum analysis. The *MODE* file will be reused and the necessary files will be cleaned up for the new spectrum analysis.

## Notes

Valid only for a spectrum analysis (**ANTYPE,SPECTR**). This operation must be preceded by a modal solution (**ANTYPE,MODAL**) with the appropriate files available. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

## Product Restrictions

Only *Sptype* = SPRS is allowed in ANSYS Professional.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Solution>Analysis Type>Analysis Options**

**SPREAD**, *VALUE*

Turns on a dashed tolerance curve for the subsequent curve plots.

[POST26: Display](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**VALUE**

Amount of tolerance. For example, 0.1 is  $\pm 10\%$ .

**Command Default**

No tolerance curve.

**Menu Paths**

Main Menu>TimeHist Postpro>Settings>Graph

**SPSCAN**, *FREQ, LOCAL, PHIB, PHIE, PHIINC, THETAB, THETA E, THETA INC, FILEOPT, SLVOPT, SLVACC, SYMOPT*

Performs a harmonic analysis of a unit cell over a range of angles and extracts the S-parameter.

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

**FREQ**

Working frequency in Hz.

**LOCAL**

Local coordinate system number (defaults to 0).

**PHIB**

Beginning angle from x-axis towards y-axis,  $\phi$ , in degrees (defaults to 0).

**PHIE**

Ending angle from x-axis towards y-axis,  $\phi$ , in degrees (defaults to 0).

**PHIINC**

Increment of  $\phi$  in degrees (defaults to 0).

**THETAB**

Beginning angle from +z-axis towards -z-axis,  $\theta$ , in degrees (defaults to 0).

**THETA E**

Ending angle from +z-axis towards -z-axis,  $\theta$ , in degrees (defaults to 0).

**THETA INC**

Increment of  $\theta$  in degrees (defaults to 0).

**FILEOPT**

**0**  
Output S-parameter magnitude and phase angle to Jobname . scan.

**1**  
Output S-parameter magnitude in dB and phase angle to Jobname . scan.



**SLVOPT**

Solver option:

**0**

SPARSE

**1**

QMR (symmetric matrices)

**2**

ICCG (unsymmetric matrices)

**SLVACC**

Convergence criteria for an iterative solver. Defaults to 1.0E-6.

**SYMOPT**

Key for S-parameter output format:

**0**

The S-parameter matrix is symmetrical (default). The following are examples.

For a 2 port system that has only port 1 excited, the assumption is that  $S_{12} = S_{21}$  and  $S_{22} = S_{11}$ .

$$[S] = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix}$$

For a 3 port system that has only port 1 excited, the assumption is that  $S_{12} = S_{21}$  and  $S_{13} = S_{31}$ .

$$[S] = \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & 0 & 0 \\ S_{31} & 0 & 0 \end{bmatrix}$$

**1**

The S-parameter matrix is unsymmetrical. The following are examples.

For a 2 port system that has only port 1 excited, the assumption is that  $S_{12}$  and  $S_{22}$  are zero.

$$[S] = \begin{bmatrix} S_{11} & 0 \\ S_{21} & 0 \end{bmatrix}$$

For a 3 port system that has only port 1 excited, the assumption is that  $S_{12}$ ,  $S_{22}$ ,  $S_{32}$ ,  $S_{13}$ ,  $S_{23}$ , and  $S_{33}$  are all zero.

$$[S] = \begin{bmatrix} S_{11} & 0 & 0 \\ S_{21} & 0 & 0 \\ S_{31} & 0 & 0 \end{bmatrix}$$

**Notes**

See the figure [Figure 4.19: "Spherical Coordinates"](#) in the *High-Frequency Electromagnetic Analysis Guide* for an illustration of the coordinate system.

## Menu Paths

Main Menu>Solution>Solve>S-Par Angle Sweep

---

**SPSWP**, *FREQB*, *FREQE*, *FREQINC*, *SWPOPT*, *EFACC*, *OUTPUT*, *FILEOPT*, *SLVOPT*, *SLVACC*, *SYMOPT*

Computes S-parameters over a frequency range and writes them to a file.

[SOLUTION: Analysis Options](#)

MP <> <> <> <> <> <> <> <> EH <> PP <> <> <>

### **FREQB**

Frequency (Hz) at the beginning of the *FREQB* to *FREQE* range. If *FREQE* is blank, the solution is done only at frequency *FREQB*.

### **FREQE**

Frequency at end of this range.

### **FREQINC**

Frequency increment. The number of solutions performed is  $[(FREQE - FREQB)/FREQINC] + 1$ . Solutions are always performed at *FREQB* and *FREQE*.

### **SWPOPT**

Solution Method (See [HROPT](#) command):

- 0** Variational Technology (default).
- 1** Full method.
- 2** Variational Technology using a perfect absorber.

---

## Note

To use any Variational Technology method, you must have the separately licensed VT Accelerator. The Variational Technology Perfect Absorber method (*Method* = VTPA) is about 20% faster but slightly less accurate than the Variational Technology method (*Method* = VT).

### **EFACC**

Electric field accuracy for fast exploration method. Defaults to 0.01.

### **OUTPUT**

- 0** Create minimal size results file required to compute S-parameters for the last port excitation case (see "[Notes](#)" (p. 1616)), for all frequencies. (default).
- 1** Create complete results file for the last port excitation case (see "[Notes](#)" (p. 1616)), for all frequencies.
- 2** Create complete results file for the last port excitation case (see "[Notes](#)" (p. 1616)), for the last frequency only, valid only for full method.

**FILEOPT**

- 0** Output S-parameter magnitude and phase angle in Touchstone file.
- 1** Output S-parameter magnitude in dB and phase angle in Touchstone file.
- 2** Output S-parameter real and imaginary parts in Touchstone file.

**SLVOPT**

- 0** SPARSE
- 1** QMR (symmetric matrices).
- 2** ICCG (unsymmetric matrices).

**SLVACC**

Convergence criteria for an iterative solver. Defaults to 1.0E-6.

**SYMOPT**

Key for S-parameter output format:

- 0** The S-parameter matrix is symmetrical (default). The following are examples.

For a 2 port system that has only port 1 excited, the assumption is that  $S_{12} = S_{21}$  and  $S_{22} = S_{11}$ .

$$[S] = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix}$$

For a 3 port system that has only port 1 excited, the assumption is that  $S_{12} = S_{21}$  and  $S_{13} = S_{31}$ .

$$[S] = \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & 0 & 0 \\ S_{31} & 0 & 0 \end{bmatrix}$$

- 1** The S-parameter matrix is unsymmetrical. The following are examples.

For a 2 port system that has only port 1 excited, the assumption is that  $S_{12}$  and  $S_{22}$  are zero.

$$[S] = \begin{bmatrix} S_{11} & 0 \\ S_{21} & 0 \end{bmatrix}$$

For a 3 port system that has only port 1 excited, the assumption is that  $S_{12}$ ,  $S_{22}$ ,  $S_{32}$ ,  $S_{13}$ ,  $S_{23}$ , and  $S_{33}$  are all zero.

$$[S] = \begin{bmatrix} S_{11} & 0 & 0 \\ S_{21} & 0 & 0 \\ S_{31} & 0 & 0 \end{bmatrix}$$

## Notes

This command computes S-parameters by sequencing through a series of harmonic solutions with different port excitations over the desired frequency range. To use this function, port flag boundary conditions must be previously set (see **SF**, **SFA** for exterior waveguide ports, or **BFA**, **BFL** or **BF** for interior waveguide or transmission line ports). Ports should be numbered sequentially from "1" with no gaps in the numbering. Resulting S-parameters are written to a file `jobname.snp`, where *n* is the number of ports. The file is written in Touchstone format.

S-parameters require a sequence of solutions whereby for each solution, one port is "excited" and the other ports are "matched". Each solution represents one column of a S-parameter matrix (i.e., if Port 1 is excited for a three-port system, the resulting column represents the S11, S21 and S31 S-parameters). A full S-parameter matrix for a *n*-port system at one frequency requires *n* solutions alternating "excited" and "matched" port boundary conditions at each port. The **SPSWP** command will solve a column of the S-parameter matrix for each port that has a defined excitation. If all ports have a defined excitation, then the full S-parameter matrix will be computed.

To prepare the ports for **SPSWP**, each port must be defined (flagged) as a port and it must have a port excitation defined. For waveguide ports, excitation is defined using the **HFPORT** command. For transmission line ports, excitation is defined using the **BFA**, **BFL**, or **BF** commands using the JS, H, or EF load options. Transmission line excitation must be defined on the port surface itself (nodes, lines, or areas).

## Menu Paths

**Main Menu>Solution>Solve>S-Par Sweep**

---

## SPTOPT

Specifies "Spectrum analysis options" as the subsequent status topic.

SOLUTION:Status

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

## Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Show Status**

**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Show Status**

**Utility Menu>List>Status>Solution>Spectrum Options**

---

**SPUNIT**, *TBLNO*, *Type*, *GVALUE*, *KeyInterp*

**Defines the type of multi-point response spectrum.**

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

***TBLNO***

Input table number.

***Type***

Label identifying the type of spectrum:

**DISP**

Displacement spectrum (**SPVAL** values interpreted as displacements with units of length).

**VELO**

Velocity spectrum (**SPVAL** values interpreted as velocities with units of length/time).

**ACEL**

Acceleration spectrum (**SPVAL** values interpreted as accelerations with units of length/time<sup>2</sup>).

**ACCG**

Acceleration spectrum (**SPVAL** values interpreted as accelerations with units of g/time<sup>2</sup>).

**FORC**

Force spectrum.

**PRES**

Pressure spectrum.

***GVALUE***

Value of acceleration due to gravity in any arbitrary units for Type=ACCG table. Default is 386.4 in/sec<sup>2</sup>.

***KeyInterp***

Key to activate or deactivate the linear interpolation between input response spectrum points and input response spectrum curves:

**0 (OFF or NO)**

Deactivate linear and use logarithmic interpolation. This value is the default.

**1 (ON or YES)**

Activate linear interpolation.

## Command Default

Acceleration (ACEL) spectrum (acceleration<sup>2</sup>/Hz).

## Notes

Defines the type of multi-point response spectrum defined by the **SPFREQ** and **SPVAL** commands.

Force (**FORC**) and pressure (**PRES**) type spectra can be used only as a nodal excitation.

*GVALUE* is valid only when type=ACCG is specified. A zero or negative value cannot be used. A parameter substitution can also be performed.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Settings**  
**Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Settings**

---

**SPVAL**, *TBLNO*, *CURVNO*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

**Defines multi-point response spectrum values.**

[SOLUTION: Spectrum Options](#)

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

### *TBLNO*

Input table number. It corresponds to *TBLNO* on the **SPFREQ** command.

### *CURVNO*

Input curve number. It corresponds to *CURVNO* on the **SPDAMP** command (optional).

### *SV1*, *SV2*, *SV3*, . . . . ., *SV7*

Spectral values corresponding to the frequency points (**SPFREQ**) and damping ratio (**SPDAMP**). Values are interpreted as defined with the **SPUNIT** command.

## Notes

Defines multi-point response spectrum values to be associated with the previously defined frequency points (**SPFREQ**). It can also be associated with the previously defined damping value (**SPDAMP**). If *CURVNO* is not specified, the input spectrum is not associated with a damping value.

Repeat **SPVAL** command for additional values, up to the number of frequency points (**SPFREQ**). Values are added after the last nonzero value.

The interpolation method between response spectrum points and curves is specified using *KeyInterp* on the **SPUNIT** command. It is logarithmic by default.

Use the **SPTOPT** and **STAT** commands to list current spectrum curve values.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Spect vs Freq**  
**Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Spect vs Freq**

---

**SQRT**, *IR*, *IA*, *--*, *--*, *Name*, *--*, *--*, *FACTA*

**Forms the square root of a variable.**

[POST26: Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***IR***

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

***IA***

Reference number of the variable to be operated on.

*--*, *--*

Unused fields.

***Name***

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

*--*, *--*

Unused fields.

***FACTA***

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

## Notes

Forms the square root of a variable according to the operation:

$$IR = \sqrt{FACTA \times IA}$$

## Menu Paths

**Main Menu>TimeHist Postpro>Math Operations>Square Root**

**\*SREAD**, *StrArray*, *Fname*, *Ext*, *--*, *nChar*, *nSkip*, *nRead*

**Reads a file into a string array parameter.**

[APDL: Array Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

***StrArray***

Name of the "string array" parameter which will hold the read file. String array parameters are similar to character arrays, but each array element can be as long as 128 characters. If the string parameter does not exist, it will be created. The array will be created as: \*DIM,StrArray,STRING,nChar,nRead

**Fname**

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

**Ext**

Filename extension (8 character maximum).

--

Unused field.

**nChar**

Number of characters per line to read (default is length of the longest line in the file).

**nSkip**

Number of lines to skip at the start of the file (default is 0).

**nRead**

Number of lines to read from the file (default is the entire file).

## Notes

The **\*SREAD** command reads from a file into a string array parameter. The file must be an ASCII text file.

## Menu Paths

**This command cannot be accessed from a menu.**

---

### **SRSS**, *SIGNIF*, *Label*, *AbsSumKey*

**Specifies the square root of sum of squares mode combination method.**

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***SIGNIF***

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multipoint, or DDAM response (**SPOPT**, *SPRS*, *MPRS* or *DDAM*), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**, *PSD*.)

***Label***

Label identifying the combined mode solution output.

**DISP**

Displacement solution (default). Displacements, stresses, forces, etc., are available.

**VELO**

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

**ACEL**

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.



**AbsSumKey**

Absolute Sum combination key (for **SPOPT**,MPRS only):

**NO**

Do not use the Absolute Sum method (default).

**YES**

Combine the modes per excitation direction using the Absolute Sum method, then combine the resulting quantities using the square root of sum of squares method.

When using Absolute Sum combination, the excitation direction must be specified using the **SED** command.

**Notes**

This command is also valid for PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine**  
**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine**  
**Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine**  
**Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine**  
**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine**

**SSBT**,  $B_{11}^T$ ,  $B_{22}^T$ ,  $B_{12}^T$ ,  $T$

**Specifies preintegrated bending thermal effects for shell sections.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

$B_{11}^T$ ,  $B_{22}^T$ ,  $B_{12}^T$

Bending thermal effects component [ $B^T$ ].

$T$

Temperature.

**Notes**

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \epsilon \\ \kappa \end{Bmatrix} - (T - T^I) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$

$$[S] = [E]\{\gamma\}$$

The **SSBT** command, one of several [preintegrated shell section commands](#), specifies the bending thermal effects quantity (submatrix  $[B^T]$  data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

The  $[B^T]$  quantity represents bending stress resultants caused by a unit raise in temperature on a fully constrained model. For a layered composite shell, it is usually necessary to specify both the  $[B^T]$  and  $[M^T]$  quantities (by issuing the **SSBT** and **SSMT** commands, respectively).

Unspecified values default to zero.

Related commands are **SSPA**, **SSPB**, **SSPD**, **SSPE**, **SSMT**, and **SSPM**.

If you are using the **SHELL181** or **SHELL281** element's Membrane option (KEYOPT(1) = 1), it is not necessary to issue this command.

For complete information, see [Using Preintegrated General Shell Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Shell>Pre-integrated**

---

**/SSCALE**, *WN*, *SMULT*

**Sets the contour multiplier for topographic displays.**

[GRAPHICS:Scaling](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *WN*

Window number (or ALL) to which command applies (defaults to 1).

### *SMULT*

Contour multiplier that factors in results based on the product of the multiplier and the result being plotted. Defaults to 0.0 (no topographic effects).

## Command Default

No topographic contour effects.

## Notes

Use this command to scale values to the geometry when the contours are shown elevated. For section displays **[/TYPE]**, the elevation is performed perpendicular to the section face.

Nonzero contour multipliers factoring in large results (stresses or displacements) can produce very large distortion, causing images to disappear. To bring a distorted image back into view, reduce the contour multiplier value.

Portions of this command are not supported by PowerGraphics **[/GRAPHICS,POWER]**.

## Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Contour Style

**SSLN**, *FACT*, *SIZE*

Selects and displays small lines in the model.

[PREP7:Lines](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**FACT**

Factor used to determine small lines. *FACT* times the average line length in the model is used as the line length limit below which lines will be selected.

**SIZE**

Line length limit for line selection. Lines that have a length less than or equal to *SIZE* will be selected. Used only if *FACT* is blank.

## Notes

**SSLN** invokes a predefined ANSYS macro for selecting small lines in a model. Lines that are smaller than or equal to the specified limit (*FACT* or *SIZE*) are selected and line numbers are displayed. This command macro is useful for detecting very small lines in a model that may cause problems (i.e., poorly shaped elements or a meshing failure) during meshing. All lines that are not "small" will be unselected and can be reselected with the **LSEL** command.

## Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Sel Small Lines

**SSMT**,  $M^T_{11}$ ,  $M^T_{22}$ ,  $M^T_{12}$ ,  $T$

Specifies preintegrated membrane thermal effects for shell sections.

[PREP7:Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

$M^T_{11}$ ,  $M^T_{22}$ ,  $M^T_{12}$

Membrane thermal effects component [ $M^T$ ].

$T$

Temperature.

## Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} \mathbf{N} \\ \mathbf{M} \end{Bmatrix} = \begin{bmatrix} \mathbf{A} & \mathbf{B} \\ \mathbf{B}^T & \mathbf{D} \end{bmatrix} \begin{Bmatrix} \boldsymbol{\varepsilon} \\ \boldsymbol{\kappa} \end{Bmatrix} - (\mathbf{T} - \mathbf{T}^I) \begin{Bmatrix} \mathbf{M}^T \\ \mathbf{B}^T \end{Bmatrix}$$

$$[\mathbf{S}] = [\mathbf{E}]\{\boldsymbol{\gamma}\}$$

The **SSMT** command, one of several [preintegrated shell section commands](#), specifies the membrane thermal effects quantity (submatrix  $[\mathbf{M}^T]$  data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

The  $[\mathbf{M}^T]$  quantity represents membrane stress resultants caused by a unit raise in temperature on a fully constrained model. For a layered composite shell, it is usually necessary to specify both the  $[\mathbf{M}^T]$  and  $[\mathbf{B}^T]$  quantities (by issuing the **SSMT** and **SSBT** commands, respectively).

Unspecified values default to zero.

Related commands are **SSPA**, **SSPB**, **SSPD**, **SSPE**, **SSBT**, and **SSPM**.

For complete information, see [Using Preintegrated General Shell Sections](#).

## Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

**SSPA**,  $A_{11}$ ,  $A_{21}$ ,  $A_{31}$ ,  $A_{22}$ ,  $A_{32}$ ,  $A_{33}$ ,  $T$

Specifies a preintegrated membrane stiffness for shell sections.

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

$A_{11}$ ,  $A_{21}$ ,  $A_{31}$ ,  $A_{22}$ ,  $A_{32}$ ,  $A_{33}$

Membrane stiffness component (symmetric lower part of submatrix  $[\mathbf{A}]$ ).

$T$

Temperature.

## Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} \mathbf{N} \\ \mathbf{M} \end{Bmatrix} = \begin{bmatrix} \mathbf{A} & \mathbf{B} \\ \mathbf{B}^T & \mathbf{D} \end{bmatrix} \begin{Bmatrix} \boldsymbol{\varepsilon} \\ \boldsymbol{\kappa} \end{Bmatrix} - (\mathbf{T} - \mathbf{T}^I) \begin{Bmatrix} \mathbf{M}^T \\ \mathbf{B}^T \end{Bmatrix}$$

$$[\mathbf{S}] = [\mathbf{E}]\{\boldsymbol{\gamma}\}$$

The **SSPA** command, one of several [preintegrated shell section commands](#), specifies the membrane stiffness quantity (submatrix  $[\mathbf{A}]$ ) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified values default to zero.

Related commands are **SSPB**, **SSPD**, **SSPE**, **SSMT**, **SSBT**, and **SSPM**.

For complete information, see [Using Preintegrated General Shell Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Shell>Pre-integrated**

**SSPB**,  $B_{11}$ ,  $B_{21}$ ,  $B_{31}$ ,  $B_{22}$ ,  $B_{32}$ ,  $B_{33}$ ,  $T$ ,  $B_{12}$ ,  $B_{13}$ ,  $B_{23}$

**Specifies a preintegrated coupling stiffness for shell sections.**

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

$B_{11}$ ,  $B_{21}$ ,  $B_{31}$ ,  $B_{22}$ ,  $B_{32}$ ,  $B_{33}$

Coupling stiffness component (symmetric lower part of submatrix **[B]**).

$T$

Temperature.

$B_{12}$ ,  $B_{13}$ ,  $B_{23}$

Upper part of submatrix **[B]**

## Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \end{Bmatrix} - (T - T^T) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$

$$[S] = [E]\{\gamma\}$$

If the coefficients  $B_{12}$ ,  $B_{13}$ ,  $B_{23}$  are undefined, ANSYS uses a symmetric form of submatrix **[B]**. If any one of the coefficients  $B_{12}$ ,  $B_{13}$ ,  $B_{23}$  is nonzero, ANSYS considers submatrix **[B]** to be unsymmetric.

The **SSPB** command, one of several [preintegrated shell section commands](#), specifies the coupling stiffness quantity (submatrix **[B]** data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified values default to zero.

Related commands are **SSPA**, **SSPD**, **SSPE**, **SSMT**, **SSBT**, and **SSPM**.

If you are using the **SHELL181** or **SHELL281** element's Membrane option (KEYOPT(1) = 1), it is not necessary to issue this command.

For complete information, see [Using Preintegrated General Shell Sections](#).

## Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

---

**SSPD**,  $D_{11}$ ,  $D_{21}$ ,  $D_{31}$ ,  $D_{22}$ ,  $D_{32}$ ,  $D_{33}$ ,  $T$

Specifies a preintegrated bending stiffness for shell sections.

[PREP7: Cross Sections](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

$D_{11}$ ,  $D_{21}$ ,  $D_{31}$ ,  $D_{22}$ ,  $D_{32}$ ,  $D_{33}$

Bending stiffness component (symmetric lower part of submatrix  $[D]$ ).

$T$

Temperature.

## Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \epsilon \\ \kappa \end{Bmatrix} - (T - T^I) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$

$$[S] = [E]\{\gamma\}$$

The **SSPD** command, one of several [preintegrated shell section commands](#), specifies the bending stiffness quantity (submatrix  $[D]$  data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified commands default to zero.

Related commands are **SSPA**, **SSPB**, **SSPE**, **SSMT**, **SSBT**, and **SSPM**.

If you are using the **SHELL181** or **SHELL281** element's Membrane option (**KEYOPT**(1) = 1), it is not necessary to issue this command.

For complete information, see [Using Preintegrated General Shell Sections](#).

## Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

---

**SSPE**,  $E_{11}$ ,  $E_{21}$ ,  $E_{22}$ ,  $T$ **Specifies a preintegrated transverse shear stiffness for shell sections.**[PREP7: Cross Sections](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

 $E_{11}$ ,  $E_{21}$ ,  $E_{22}$ Transverse shear stiffness component (symmetric lower part of submatrix  $[E]$ ). $T$ 

Temperature.

## Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \end{Bmatrix} - (T - T^I) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$

$$[S] = [E]\{\gamma\}$$

The **SSPE** command, one of several [preintegrated shell section commands](#), specifies the transverse shear stiffness quantity (submatrix  $[E]$  data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE** command).

Unspecified values default to zero.

Related commands are **SSPA**, **SSPB**, **SSPD**, **SSMT**, **SSBT**, and **SSPM**.

If you are using the **SHELL181** or **SHELL281** element's Membrane option (**KEYOPT**(1) = 1), it is not necessary to issue this command.

For complete information, see [Using Preintegrated General Shell Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Shell>Pre-integrated**

**SSPM**,  $DENS$ ,  $T$ **Specifies mass density for a preintegrated shell section.**[PREP7: Cross Sections](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

 $DENS$ 

Mass density.

 $T$ 

Temperature.

## Notes

The **SSPM** command, one of several [preintegrated shell section commands](#), specifies the mass density (assuming a unit thickness) for a preintegrated shell section. The value specified is associated with the section most recently defined (via the **SECTYPE** command).

Related commands are **SSPA**, **SSPB**, **SSPD**, **SSPE**, **SSMT**, and **SSBT**.

For complete information, see [Using Preintegrated General Shell Sections](#).

## Menu Paths

**Main Menu>Preprocessor>Sections>Shell>Pre-integrated**

---

### SSTIF, *Key*

**Activates stress stiffness effects in a nonlinear analysis.**

[SOLUTION: Nonlinear Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### Key

Stress stiffening key:

#### OFF

No stress stiffening is included (default unless **NLGEOM,ON**).

#### ON

Stress stiffening is included (default if **NLGEOM,ON**).

## Command Default

**SSTIF** will be turned on if **NLGEOM,ON**; otherwise it will be turned off.

## Notes

Activates stress stiffness effects in a nonlinear analysis (**ANTYPE,STATIC** or **TRANS**). (The **PSTRES** command also controls the generation of the stress stiffness matrix and therefore should not be used in conjunction with **SSTIF**.) If used within the solution processor, this command is valid only within the first load step.

When **SOLCONTROL** and **NLGEOM** are ON, **SSTIF** defaults to ON. This normally forms all of the consistent tangent matrix. However, for some special nonlinear cases, this can lead to divergence caused by some elements which do not provide a complete consistent tangent (notably, elements outside the 18x family). In such a case, ANSYS recommends issuing an **SSTIF,OFF** command to achieve convergence. For [current-technology elements](#), setting **SSTIF,OFF** when **NLGEOM** is ON has no effect (because stress stiffness effects are always included).

The default values given for this command assume **SOLCONTROL,ON** (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL,ON** and **SOLCONTROL,OFF**.

This command is also valid in PREP7.



## Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options  
Main Menu>Solution>Analysis Type>Analysis Options

## SSUM

Calculates and prints the sum of element table items.

POST1: Element Table

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

Calculates and prints the tabular sum of each existing labeled result item [ETABLE] for the selected elements. If absolute values are requested [SABS,1], absolute values are used.

## Menu Paths

Main Menu>General Postproc>Element Table>Sum of Each Item

**STABILIZE**, *Key*, *Method*, *VALUE*, *SubStpOpt*, *FORCELIMIT*

Activates stabilization for all elements that support nonlinear stabilization.

SOLUTION: Analysis Options

MP ME ST PR PRN <> <> FL EM EH <> PP <> EME MFS

### Key

Key for controlling nonlinear stabilization:

#### OFF

Deactivate stabilization. This value is the default.

#### CONSTANT

Activate stabilization. The energy-dissipation ratio or damping factor remains constant during the load step.

#### REDUCE

Activate stabilization. The energy-dissipation ratio or damping factor is reduced linearly to zero at the end of the load step from the specified or calculated value.

### Method

The stabilization-control method:

#### ENERGY

Use the energy-dissipation ratio as the control. This value is the default when *Key* ≠ OFF.

#### DAMPING

Use the damping factor as the control.

**VALUE**

The energy-dissipation ratio (*Method* = ENERGY) or damping factor (*Method* = DAMPING). This value must be greater than 0 when *Method* = ENERGY or *Method* = DAMPING. When *Method* = ENERGY, this value is usually a number between 0 and 1.

**SubStpOpt**

Option for the first substep of the load step:

**NO**

Stabilization is not activated for the first substep even when it does not converge after the minimal allowed time increment is reached. This value is the default when *Key* ≠ OFF.

**MINTIME**

Stabilization is activated for the first substep if it still does not converge after the minimal allowed time increment is reached.

**ANYTIME**

Stabilization is activated for the first substep. Use this option if stabilization was active for the previous load step via *Key* = CONSTANT.

**FORCELIMIT**

The stabilization force limit coefficient, such that  $0 < \text{FORCELIMIT} < 1$ . The default value is 0.2. To omit a stabilization force check, set this value to 0.

## Command Default

Once issued, the stabilization effects of the command remain until you issue either a **STABILIZE** command (with no arguments) or a **STABILIZE,OFF** command. If you issue the command with no arguments, the effect is to deactivate stabilization.

## Notes

Once issued, a **STABILIZE** command remains in effect until you reissue the command.

For the energy dissipation ratio, specify *VALUE* = 1.0e-4 if you have no prior experience with the current model; if convergence problems are still an issue, increase the value gradually. The damping factor is mesh-, material-, and time-step-dependent; an initial reference value from the previous run (such as a run with the energy-dissipation ratio as input) should suggest itself.

Exercise caution when specifying *SubStpOpt* = MINTIME or ANYTIME for the first load step; ANSYS recommends this option only for experienced users. If stabilization was active for the previous load step via *Key* = CONSTANT and convergence is an issue for the first substep, specify *SubStpOpt* = ANYTIME.

When the L2-norm of the stabilization force (CSRSS value) exceeds the L2-norm of the internal force multiplied by the stabilization force coefficient, ANSYS issues a message displaying both the stabilization force norm and the internal force norm. The *FORCELIMIT* argument allows you to change the default stabilization force coefficient (normally 20 percent).

This command stabilizes the degrees of freedom for [current-technology elements](#) only. Other elements can be included in the FE model, but their degrees of freedom are not stabilized.

For more information about nonlinear stabilization, see [Unstable Structures](#) in the *Structural Analysis Guide*. For additional tips that can help you to achieve a stable final model, see [Keep It Simple](#) in the *Structural Analysis Guide*.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL**

---

### STAOPT, *Method*

**Specifies static analysis options.**

[SOLUTION: Analysis Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### *Method*

Solution method for the static analysis:

#### **DEFA**

Regular ANSYS solve (default).

#### **VT**

Solve with Variational Technology.

## Notes

Specifies the method of solution for a static analysis (**ANTYPE**,STATIC). If used in SOLUTION, this command is valid only within the first load step.

The VT option is valid for either thermal or structural nonlinear analyses, where it attempts to reduce the total number of iterations. You can also use the VT option for ANSYS DesignXplorer design parameter studies.

After a Design Exploration solution (**SOLVE** command), the Variational Technology model is "frozen", i.e., you cannot add or delete input variables.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>DesignXplorer>Solution>Solve**  
**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>Analysis Options**

---

## STAT

Displays the status of database settings.

DATABASE: Set Up  
DISPLAY: Action

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

In the DISPLAY program, **STAT** will show the current status of the program settings.

In the ANSYS program, **STAT** is a command generated by the GUI and will appear in the log file (Job-name.LOG) if status is requested for some items under **Utility Menu > List > Status**. Generally, **STAT** will be preceded by one of the commands listed below, which specifies the particular topic that status was requested for.

If entered directly into the program, the **STAT** command should be immediately preceded by the desired topic command listed below. In processors other than those listed below (e.g., AUX12), no topic command should precede **STAT**.

This command is valid in any processor.

PREP7 topic commands (and their corresponding topics) are:

Topic Command	Topic
<b>ETYPE</b>	Element types
<b>RCON</b>	Real constants
<b>MATER</b>	Material properties
<b>TBLE</b>	Data table properties
<b>PRIM</b>	Solid model primitives
<b>KEYPTS</b>	Keypoints
<b>LINE</b>	Lines
<b>AREAS</b>	Areas
<b>VOLUMES</b>	Volumes
<b>GEOMETRY</b>	Solid model information
<b>MESHING</b>	Meshing
<b>BOOL</b>	Booleans
<b>NODES</b>	Nodes
<b>ELEM</b>	Elements
<b>SELM</b>	Superelements
<b>DIGIT</b>	Node digitizing
<b>COUPLE</b>	Node coupling
<b>CEQN</b>	Constraint equations
<b>REORDER</b>	Model reordering

SOLUTION topic commands (and their corresponding topics) are:

<b>Topic Command</b>	<b>Topic</b>
<b>ATYPE</b>	Analysis types
<b>MASTER</b>	Master DOF
<b>GAP</b>	Reduced transient gap conditions
<b>DEACT</b>	Element birth and death (deactivation)
<b>LSOPER</b>	Load step operations
<b>FECNS</b>	Constraints on nodes
<b>FEFOR</b>	Forces on nodes
<b>FESURF</b>	Surface loads on elements
<b>FEBODY</b>	Body loads on elements
<b>SMCONS</b>	Constraints on the solid model
<b>SMFOR</b>	Forces on the solid model
<b>SMSURF</b>	Surface loads on the solid model
<b>SMBODY</b>	Body loads on the solid model
<b>INRTIA</b>	Inertial loads
<b>GENOPT</b>	General options
<b>DYNOPT</b>	Dynamic analysis options
<b>NLOPT</b>	Nonlinear analysis options
<b>OUTOPT</b>	Output options
<b>BIOOPT</b>	Biot-Savart options
<b>SPTOPT</b>	Spectrum analysis options
<b>SOLUOPT</b>	Solution options
<b>FLOTRAN</b>	FLOTRAN data settings

POST1 topic commands (and their corresponding topics) are:

<b>Topic Command</b>	<b>Topic</b>
<b>DEFINE</b>	Data definition settings
<b>SORT</b>	Sort settings
<b>PRINT</b>	Print settings
<b>DISPLAY</b>	Display settings
<b>CALC</b>	Calculation settings
<b>PATH</b>	Path data settings
<b>LCCALC</b>	Load case settings
<b>DATADEF</b>	Directly defined data status
<b>FATIGUE</b>	Fatigue data status
<b>POINT</b>	Point flow tracing settings
<b>SPEC</b>	Miscellaneous specifications

POST26 topic commands (and their corresponding topics) are:

Topic Command	Topic
<b>DEFINE</b>	Data definition settings
<b>OPERATE</b>	Operation data
<b>PRINT</b>	Print settings
<b>PLOTTING</b>	Plotting settings

## Menu Paths

Main Menu>Prob Design>Prob Database>Status

Main Menu>Solution>Solve>Current LS

**\*STATUS**, *Par*, *IMIN*, *IMAX*, *JMIN*, *JMAX*, *KMIN*, *KMAX*, *LMIN*, *LMAX*, *MMIN*, *MMAX*, *KPRI*

Lists the current parameters and abbreviations.

[APDL: Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Par*

Specifies the parameter or sets of parameters listed. For array parameters, use *IMIN*, *IMAX*, etc. to specify ranges. Use **\*DIM** to define array parameters. Use **\*VEDIT** to review array parameters interactively. Use **\*VWRITE** to print array values in a formatted output. If *Par* is blank, list all scalar parameter values, array parameter dimensions, and abbreviations. If *ARGX*, list the active set of local macro parameters (ARG1 to ARG9 and AR10 to AR99) [**\*USE**].

The following are possible values for *Par*

#### **ALL or blank --**

Lists all parameters (except local macro parameters and those with names beginning or ending with an underbar) and toolbar abbreviations.

#### **\_PRM --**

Lists only parameters with names beginning with an underbar (\_). These are ANSYS internal parameters.

#### **PRM\_ --**

Lists only parameters with names ending with an underbar (\_). A good APDL programming convention is to ensure that all parameters created by your system programmer are named with a trailing underbar.

#### **ABBR --**

Lists all toolbar abbreviations.

#### **PARM --**

Lists all parameters (except local macro parameters and those with names beginning or ending with an underbar).

#### **PARNAME --**

Lists only the parameter specified. *PARNAME* cannot be a local macro parameter name.

**ARGX --**

Lists all local macro parameter values (ARG1- AR99) that are non-zero or non-blank.

**IMIN, IMAX, JMIN, JMAX, KMIN, KMAX, LMIN, LMAX, MMIN, MMAX**

Range of array elements to display (in terms of the dimensions (row, column, plane, book, and shelf). Minimum values default to 1. Maximum values default to the maximum dimension values. Zero may be input for *IMIN*, *JMIN*, and *KMIN* to display the index numbers. See **\*TAXIS** command to list index numbers of 4- and 5-D tables.

**KPRI**

Use this field to list your primary variable labels (X, Y, Z, TIME, etc.).

**1**

List the labels (default). *YES*, *Y*, or *ON* are also valid entries.

**0**

Do not list the labels. *NO*, *N*, or *OFF* are also valid entries.

## Notes

You cannot obtain the value for a single local parameter (e.g., **\*STATUS**,ARG2). You can only request all local parameters simultaneously using **\*STATUS**,ARGX.

This command is valid in any processor.

## Menu Paths

Utility Menu>List>Other>Named Parameter

Utility Menu>List>Other>Parameters

Utility Menu>List>Status>Parameters>All Parameters

Utility Menu>List>Status>Parameters>Named Parameters

## /STATUS, Lab

Lists the status of items for the run.

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Lab**

Items to list status for:

**ALL**

List all below (default).

**TITLE**

List only titles, Jobname, and revision number.

**UNITS**

List only units.

**MEM**

List only memory data statistics.

**DB**

List only database statistics

**CONFIG**

List only configuration parameters.

**GLOBAL**

Provides a global status summary.

**SOLU**

Provides a solution status summary.

**PROD**

Provides a product summary.

**Notes**

Displays various items active for the run (such as the ANSYS revision number, Jobname, titles, units, configuration parameters, database statistics, etc.).

This command is valid in any processor.

**Menu Paths**

Utility Menu>List>Status>Global Status

**STEF, VALUE**

Specifies Stefan-Boltzmann radiation constant.

AUX12: General Radiation  
SOLUTION: Radiosity

MP ME ST PR <> <> <> <> <> <> PP <> EME MFS

**VALUE**

Stefan-Boltzmann constant (defaults to 0.119E-10 Btu/hr/in<sup>2</sup>/ °R<sup>4</sup>).

**Command Default**

0.119E-10 Btu/hr/in<sup>2</sup>/ °R<sup>4</sup>.

**Notes**

You can use this command in the general preprocessor (PREP7) and in the Solution processor to specify the Stefan-Boltzmann constant for a FLOTRAN analysis using radiation surface boundary conditions.

**Menu Paths**

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Displacement>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Displacement>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Displacement>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Displacement>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Forces>Body Forces>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Turbulence>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Turbulence>On Lines



Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Turbulence>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Velocity>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Velocity>On Keypoints  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Velocity>On Lines  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Velocity>On Nodes  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Bound Loads>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Bound Loads>On Lines  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Areas  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Elements  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Lines  
 Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Nodes  
 Main Menu>Preprocessor>Radiation Opts>Solution Opt  
 Main Menu>Radiation Opt>Matrix Method>Other Settings  
 Main Menu>Radiation Opt>Radiosity Meth>Solution Opt  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Displacement>On Areas  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Displacement>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Displacement>On Lines  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Displacement>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Forces>Body Forces>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Turbulence>On Areas  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Turbulence>On Lines  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Turbulence>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Velocity>On Areas  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Velocity>On Keypoints  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Velocity>On Lines  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Velocity>On Nodes  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Bound Loads>On Elements  
 Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Bound Loads>On Lines  
 Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Areas  
 Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Elements  
 Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Lines  
 Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Nodes  
 Main Menu>Solution>Radiation Opts>Solution Opt

---

**/STITLE**, *NLINE*, *Title*

**Defines subtitles.**

[DATABASE: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NLINE***

Subtitle line number (1 to 4). Defaults to 1.

***Title***

Input up to 70 alphanumeric characters. Parameter substitution may be forced within the title by enclosing the parameter name or parametric expression within percent (%) signs. If *Title* is blank, this subtitle is deleted.

## Notes

Subtitles (4 maximum) are displayed in the output along with the main title `[/TITLE]`. Subtitles do not appear in GUI windows or in ANSYS plot displays. The first subtitle is also written to various ANSYS files along with the main title. Previous subtitles may be overwritten or deleted. Issue `/STATUS` to display titles.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

### **STORE**, *Lab*, *NPTS*

**Stores data in the database for the defined variables.**

[POST26: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *Lab*

Valid labels:

##### **MERGE**

Merge data from results file for the time points in memory with the existing data using current specifications (default).

##### **NEW**

Store a new set of data, replacing any previously stored data with current result file specifications and deleting any previously-calculated (OPER) variables. Variables defined using the **ANSOL** command are also deleted.

##### **APPEN**

Append data from results file to the existing data.

##### **ALLOC**

Allocate (and zero) space for *NPTS* data points.

##### **PSD**

Create a new set of frequency points for PSD calculations (replacing any previously stored data and erasing any previously calculated data).

#### *NPTS*

The number of time points (or frequency points) for storage (used only with *Lab* = ALLOC or PSD). The value may be input when using POST26 with data supplied from other than a results file. This value is automatically determined from the results file data with the NEW, APPEN, and MERGE options. For the PSD option, *NPTS* determines the resolution of the frequency vector (valid numbers are between 1 and 10, defaults to 5).

## Command Default

Merge newly-defined variables with previously stored variables for the time points stored in memory using the current specifications. If **STORE** is preceded by **TIMERANGE** or **NSTORE**, the default is **STORE,NEW**.

## Notes

This command stores data from the results file in the database for the defined variables [**NSOL**, **ESOL**, **SOLU**, **JSOL**] per specification [**FORCE**, **LAYERP26**, **SHELL**]. See the *Basic Analysis Guide* for more information.

The **STORE,PSD** command will create a new frequency vector (variable 1) for response PSD calculations [**RPSD**]. This command should first be issued before defining variables [**NSOL**, **ESOL**, **RFORCE**] for which response PSD's are to be calculated.

## Menu Paths

Main Menu>TimeHist Postpro>Store Data

**SUBSET**, *Lstep*, *SBSTEP*, *FACT*, *KIMG*, *TIME*, *ANGLE*, *NSET*

Reads results for the selected portions of the model.

POST1: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lstep*

Load step number of the data set to be read (defaults to 1):

*N*

Read load step *N*.

### **FIRST**

Read the first data set (*SBSTEP* and *TIME* are ignored).

### **LAST**

Read the last data set (*SBSTEP* and *TIME* are ignored).

### **NEXT**

Read the next data set (*SBSTEP* and *TIME* are ignored). If at the last data set, the first data set will be read as the next.

### **NEAR**

Read the data set nearest to *TIME* (*SBSTEP* is ignored). If *TIME* is blank, read the first data set.

### **LIST**

Scan the results file and list a summary of each load step. (*FACT*, *KIMG*, *TIME* and *ANGLE* are ignored.)

### *SBSTEP*

Substep number (within *Lstep*). For the buckling (**ANTYPE**,BUCKLE) analysis or the modal (**ANTYPE**,MODAL) analysis, the substep corresponds to the mode number. Defaults to last substep of load step (except for **ANTYPE**,BUCKLE or MODAL). If *Lstep* = LIST, *SBSTEP* = 0 or 1 lists the basic step information, whereas *SBSTEP* = 2 also lists the load step title, and labels imaginary data sets if they exist.

### *FACT*

Scale factor applied to data read from the file. If zero (or blank), a value of 1.0 is used. Harmonic velocities or accelerations may be calculated from the displacement results from a modal (**ANTYPE**,MODAL) or harmonic response (**ANTYPE**,HARMIC) analyses. If *FACT* = VELO, the harmonic velocities (*v*) are calculated from the displacements (*d*) at a particular frequency (*f*) according to the relationship  $v = 2 \pi f d$ . Similarly, if *FACT* = ACEL, the harmonic accelerations (*a*) are calculated as  $a = (2 \pi f)^2 d$ .

**KIMG**

Used only with results from complex analyses:

**0**

Store real part of complex solution

**1**

Store imaginary part.

**TIME**

Time-point identifying the data set to be read. For the harmonic responses analyses, time corresponds to the frequency. For the buckling analysis, time corresponds to the load factor. Used only in the following cases: If *Lstep* is NEAR, read the data set nearest to *TIME*. If both *Lstep* and *SBSTEP* are zero (or blank), read data set at time = *TIME*. If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. Solution items not written to the results file [OUTRES] for either data set will result in a null item after data set interpolation. If *TIME* is beyond the last time point on the file, use the last time point.

**ANGLE**

Circumferential location (0.0 to 360°). Defines the circumferential location for the harmonic calculations used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the harmonic elements (PLANE25, PLANE75, PLANE78, FLUID81, PLANE83, and SHELL61) of the load case. See the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for details. Note that factored values of applied constraints and loads will overwrite any values existing in the database.

**NSET**

Data set number of the data set to be read. If a positive value for *NSET* is entered, *Lstep*, *SBSTEP*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by \*SET,LIST.

## Notes

Reads a data set from the results file into the database for the selected portions of the model only. Data that has not been specified for retrieval from the results file by the INRES command will be listed as having a zero value. Each time that the SUBSET command is issued, the data currently in the database will be overwritten with a new set of data. Various operations may also be performed during the read operation. The database must have the model geometry available (or used the RESUME command before the SUBSET command to restore the geometry from File.DB).

## Menu Paths

**Main Menu>General Postproc>Read Results>By Load Step**

**Main Menu>General Postproc>Read Results>By Set Number**

**Main Menu>General Postproc>Read Results>By Time/Freq**

---

**SUCALC**, *RSetName*, *lab1*, *Oper*, *lab2*, *fact1*, *fact2*, *const*

Create new result data by operating on two existing result data sets on a given surface.

POST1: Surface Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**RSetName**

Eight character name for new result data.

**lab1**

First result data upon which to operate.

**Oper**

Mathematical operation to perform.

**ADD**

$(lab1 + lab2 + const)$

**SUB**

$(lab1 - lab2 + const)$

**MULT**

$(lab1 * lab2 + const)$

**DIV**

$(lab1 / lab2 + const)$

**EXP**

$(lab1 ^ fact1 + lab2 ^ fact2 + const)$

**COS**

$(\cos (lab1) + const)$

**SIN**

$(\sin (lab1) + const)$

**ACOS**

$(\arccos (lab1) + const)$

**ASIN**

$(\arcsin (lab1) + const)$

**ATAN**

$(\arctan (lab1) + const)$

**ATA2**

$(\arctan2 (lab1 / lab2) + const)$

**LOG**

$(\log (lab1) + const)$

**ABS**

$(\text{abs} (lab1) + const)$

**ZERO**

$(0 + const)$

**lab2**

Second result data upon which to operate.

**fact1**

First scaling factor (for EXP option only).

**fact2**

Second scaling factor (for EXP option only).

**const**

Constant added to the values in the resulting data.

## Menu Paths

**Main Menu>General Postproc>Surface Operations>Math Operations>Absolute**  
**Main Menu>General Postproc>Surface Operations>Math Operations>Add**  
**Main Menu>General Postproc>Surface Operations>Math Operations>ArcCosine**  
**Main Menu>General Postproc>Surface Operations>Math Operations>ArcSine**  
**Main Menu>General Postproc>Surface Operations>Math Operations>ArcTangent**  
**Main Menu>General Postproc>Surface Operations>Math Operations>ArcTangent2**  
**Main Menu>General Postproc>Surface Operations>Math Operations>Cosine**  
**Main Menu>General Postproc>Surface Operations>Math Operations>Divide**  
**Main Menu>General Postproc>Surface Operations>Math Operations>Exponentiate**  
**Main Menu>General Postproc>Surface Operations>Math Operations>Initialize**  
**Main Menu>General Postproc>Surface Operations>Math Operations>Multiply**  
**Main Menu>General Postproc>Surface Operations>Math Operations>Natural Log**  
**Main Menu>General Postproc>Surface Operations>Math Operations>Sine**  
**Main Menu>General Postproc>Surface Operations>Math Operations>Subtract**

---

**SUCR**, *SurfName*, *SurfType*, *nRefine*, *Radius*, *blank*, *blank*, *TolOut*

**Create a surface.**

[POST1: Surface Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**SurfName**

Eight character surface name.

**SurfType**

Surface type.

**CPLANE**

Surface is defined by the cutting plane in window one (controlled by the working plane (/CPLANE,1), NOT the view settings (/CPLANE,0)).

**SPHERE**

Surface is defined by a spherical surface centered about the working plane origin.

**INFC**

Surface is defined by a cylindrical surface centered about the working plane origin and extending indefinitely in the positive and negative Z directions.

**nRefine**

Refinement level.

**For SurfType = CPLANE**

The refinement level of the surface "mesh". This will be an integer between 0 and 3 (default = 0). See Notes below.

**For SurfType = SPHERE**

The number of divisions along a 90° arc (minimum = 9). The default is 9.

**For SurfType = INFC**

The number of divisions along a 90° arc (minimum = 9). The default is 9.

**Radius**

Appropriate radius value (for INFC or SPHERE).

**TolOut**

Tolerance value for inclusion of element facets within a prescribed volume. (for INFC)

**Notes**

This command creates a new surface and stores the following data for that surface:

GCX, GCY, GCZ - global Cartesian coordinates at each point on the surface.  
 NORMX, NORMY, NORMZ - components of the unit normal at each point on the surface.  
 DA - the contributory area of each point.

For *SurfType* = CPLANE, *nRefine* refers to the number of points that define the surface. An *nRefine* value of zero is used for points where the element face intersects the cutting plane.

If *SurfType* = CPLANE and *nRefine* = 0, the points reside at the section cuts where the element intersects the cutting plane. Increasing *nRefine* from 0 to 1 will subdivide each surface facet into 4 subfacets, and increase the number of points at which results can be interpolated.

For *SurfType* = CPLANE, the setting from the **/EFACET** command will affect the creation of surface facets and the quality of the fit of the surface in the model. **SUCR** employs geometry data from PowerGraphics to aid in determining where the surface intersects the model. If **/EFACET,1** is in effect when the **SUCR** command is issued, then the curvature of high order elements (that is, elements with midside nodes) will be ignored. If your model contains high order elements, you can see a better fit for your surface if **/EFACET,2** is in effect when the **SUCR** command is issued. Currently, the **SUCR** command interprets **/EFACET,4** to mean **/EFACET,2**.

For *SurfType* = INFC, a default tolerance of 0.01 will be applied to include the vertices of any facets that fall out of the cylinder definition. this tolerance increases the facet size by 1% to check for inclusion. Excluding facets under such a small tolerance may yield unacceptable (aesthetically) results. Increasing the tolerance by a larger amount (0.1 or 10%) will sometimes yield smother edges along the surface you create.

**Menu Paths**

**Main Menu>General Postproc>Surface Operations>Create Surface>Inf. Cylinder**  
**Main Menu>General Postproc>Surface Operations>Create Surface>On Cuttng Plane**  
**Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node**  
**Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>By Dimensions**

**SUDEL**, *SurfName*

Delete geometry information as well as any mapped results for specified surface.

[POST1: Surface Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***SurfName***

Eight character surface name.

*SurfName* = ALL will delete all surface geometry and result information.

**Menu Paths**

**Main Menu>General Postproc>Surface Operations>Delete Surfaces**

---

**SUEVAL**, *Parm, lab1, Oper*

Perform operations on a mapped item and store result in a scalar parameter.

[POST1: Surface Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Parm***

APDL parameter name.

***lab1***

Eight character set name for the first set used in calculation.

***Oper***

Operation to perform:

**SUM**

Sum of *lab1* result values.

**INTG**

Integral of *lab1* over surface.

**AVG**

Area-weighted average of a result item  $[\sum(lab1*DA) / \sum(DA)]$

**Notes**

The result of this operation is a scalar APDL parameter value. If multiple surfaces are selected when this command is issued, then the operation is carried out on each surface individually and the parameter represents the cumulative value of the operation on all selected surfaces.

**Menu Paths**

**Main Menu>General Postproc>Surface Operations>Math Operations>Average Result**

**Main Menu>General Postproc>Surface Operations>Math Operations>Integrate Results**

**Main Menu>General Postproc>Surface Operations>Math Operations>Sum of Results**

---



**SUGET**, *SurfName*, *RSetName*, *Parm*, *Geom*

**Moves surface geometry and mapped results to an array parameter.**

[POST1: Surface Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***SurfName***

Eight character surface name.

***RSetName***

Eight character result name.

***Parm***

APDL array parameter name (up to 32 characters).

***Geom***

Switch controlling how data is written.

**ON (or 1 or YES)**

Writes geometry data and interpolated results information to the parameter.

**OFF (or 0 or NO)**

Writes only interpolated results information to the parameter. (Default)

## Notes

For *Geom* = OFF (or 0 or NO), only results information is written to this parameter.

For *Geom* = ON (or 1 or YES), both geometry data and results information are written to this parameter. Geometry data includes 7 data items: (GCX, GCY, GCZ, NORMX, NORMY, NORMZ, and DA). Results information is then written to the 8th column of the parameter. SetNames of GCX, GCY, GCZ, NORMX, NORMY, NORMZ, and DA are predefined and computed when **SUCR** is issued.

## Menu Paths

**Main Menu>General Postproc>Surface Operations>Results to Array**

**SUMAP**, *RSetName*, *Item*, *Comp*

**Map results onto selected surface(s).**

[POST1: Surface Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***RSetName***

Eight character name for the result being mapped.

***Item***

Label identifying the item.

Valid item labels are defined in **PLNSOL**. Some items also require a component label.

If *Item* = CLEAR, the specified result set will be deleted from all selected surfaces

**Comp**

Component label of item (if required).

**Notes**

Results are mapped in current **RSYS**. This command interpolates and stores results data on to each of the selected surfaces. **SUMAP**, **ALL**, **CLEAR** will delete all results sets from all selected surfaces.

**Menu Paths**

**Main Menu>General Postproc>Surface Operations>Clear Results**

---

**SUMTYPE**, *Label*

Sets the type of summation to be used in the following load case operations.

[POST1:Results](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Label**

Summation type

**COMP**

Combine element component stresses only. Stresses such as average nodal stresses, principal stresses, equivalent stresses, and stress intensities are derived from the combined element component stresses. Default.

**PRIN**

Combine principal stress, equivalent stress, and stress intensity directly as stored on the results file. Component stresses are not available with this option.

**Notes**

Issue **SUMTYPE**,**PRIN** when you want to have a load case operation (**LCOPER**) act on the principal / equivalent stresses instead of the component stresses. Also issue **SUMTYPE**,**PRIN** when you want to read in load cases (**LCASE**). Note that the **SUMTYPE** setting is not maintained between /POST1 sessions.

**SUMTYPE**,**PRIN** also causes principal nodal values to be the average of the contributing principal element nodal values (see **AVPRIN**,1).

**BEAM188** and **BEAM189** elements compute principal stress, equivalent stress, and stress intensity values on request instead of storing them on the results file; **SUMTYPE**,**PRIN** does not apply for these elements.

**Menu Paths**

**Main Menu>General Postproc>Load Case>Calc Options>Stress Options**

---

**SUPL**, *SurfName*, *RSetName*, *KWIRE*

**Plot result data on all selected surfaces or on a specified surface.**

POST1: Surface Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***SurfName***

Eight character surface name. *ALL* will plot all selected surfaces.

***RSetName***

Eight character result name.

***KWIRE***

Plot in context of model.

**0**

Plot results without the outline of selected elements.

**1**

Plot results with the outline of selected elements.

## Notes

If *RSetName* is left blank, then the surface geometry will be plotted. If the Setname portion of the argument is a vector prefix (i.e. if result sets of name SetNameX, SetNameY and SetNameZ exist), ANSYS will plot these vectors on the surface as arrows. For example, **SUPL**, *ALL*, *NORM* will plot the surface normals as vectors on all selected surfaces, since *NORMX*, *NORMY* and *NORMZ* are pre-defined geometry items.

## Menu Paths

Main Menu>General Postproc>Surface Operations>Plot Results

Main Menu>General Postproc>Surface Operations>Plot Vectors

**SUPR**, *SurfName*, *RSetName*

**Print global status, geometry information and/or result information.**

POST1: Surface Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***SurfName***

Eight character surface name. If *SurfName* = *ALL*, repeat printout for all selected surfaces.

***RSetName***

Eight character result set name.

## Notes

When no arguments are specified, **SUPR** generates a global status summary of all defined surfaces. If only *SurfName* is specified, the geometry information for that surface is printed. If both *SurfName* and *RSetName* are specified, the value of the results set at each point, in addition to the geometry information, is printed.

## Menu Paths

Main Menu>General Postproc>Surface Operations>Print Results  
Main Menu>General Postproc>Surface Operations>Status>Global

---

**SURESU**, --, *Fname*, *Fext*, *Fdir*

Read a set of surface definitions and result items from a file and make them the current set.

[POST1: Surface Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

--

Unused field.

***Fname***

Eight character name.

***Fext***

Extension name.

***Fdir***

Optional path specification.

## Notes

Reading (and therefore resuming) surface and result definitions from a file overwrite any existing surface definitions.

Reading surfaces back into the postprocessor (**/POST1**) does *not* insure that the surfaces (and their results) are appropriate for the model currently residing in **/POST1**.

## Menu Paths

Main Menu>General Postproc>Surface Operations>Resume Surfaces

---

**SUSAVE**, *Lab*, *Fname*, *Fext*, *Fdir*

Saves surface definitions to a file.

[POST1: Surface Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Lab***

Eight-character surface name.

If *Lab* = ALL (default), then all surfaces are saved to the file.

If *Lab* = S, only currently selected surfaces are saved to the file.

***Fname***

File name and directory path (248 character maximum, including directory). If you do not specify a directory path, the default is your working directory and you can use all 248 characters for the file name. The file name defaults to the jobname.

***Fext***

File name extension (eight-character maximum). The extension defaults to "surf".

***Fdir***

Optional path specification.

## Notes

The **SUSAVE** command saves surface definitions (geometry information)--and any result items mapped onto the surfaces--to a file.

Issuing the **SUSAVE** command has no effect on the database. The database remains unchanged.

Subsequent executions of the **SUSAVE** command overwrite previous data in the file.

To read the contents of the file created via the **SUSAVE** command, issue the **SURESU** command.

## Menu Paths

**Main Menu>General Postproc>Surface Operations>Save Surfaces**

**SUSEL**, *Type*, *Name1*, *Name2*, *Name3*, *Name4*, *Name5*, *Name6*, *Name7*, *Name8*

### Selects a subset of surfaces

[POST1: Surface Operations](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Type***

Label identifying the type of select:

**S**

Selects a new set (default).

**R**

Reselects a set from the current set.

**A**

Additionally selects a set and extends the current set.

**U**

Unselects a set from the current set.

**ALL**

Also selects all surfaces.

**NONE**

Unselects all surfaces.

***Name1*, *Name2*, *Name3*, . . . , *Name8***

Eight character surface names

## Notes

The selected set of surfaces is used in the following operations: **SUMAP**, **SUDEL**, **SUCALC**, **SUEVAL**, and **SUECT**.

## Menu Paths

**Main Menu>General Postproc>Surface Operations>Select Surfaces**

---

**SUECT**, *RSetName*, *lab1*, *Oper*, *lab2*, *Offset*

**Create new result data by operating on two existing result vectors on a given surface.**

POST1: Surface Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *RSetName*

Eight character name of the result data output. There will be one or three *RSetName* values depending on the operation specified in *Oper*.

### *lab1*

Eight character name of the mapped data that forms vector 1. Specified sets must exist on all selected surfaces for this operation to take place. The names NORM and GC will be reserved for normals and for global (x, y, z).

### *Oper*

#### **DOT**

Computes dot product between *lab1* and *lab2* vectors. The result is a scalar parameter (*RSetName*) and each value within the set can be modified (incremented) via *Offset*.

#### **CROSS**

Computes cross product between *lab1* and *lab2* vectors. Each X, Y, Z value in the result can be modified (incremented) via *Offset*.

#### **SMULT**

Scales (*lab1x*, *lab1y*, *lab1z*) vector by scalar *lab2*. Each X, Y, Z value in the result can be modified (incremented) via *Offset*.

### *lab2*

Eight character name of the mapped data that forms vector 2. Sets with names Lab2X, Lab2Y, and Lab2Z must exist on all selected surfaces for operation to take place. For *Oper* = SMULT a scalar value or another predefined scalar item (e.g., DA) can be supplied.

### **Offset**

An offset value to be applied to the resultant *RSetName*. One value is specified for *Oper* = DOT, and three values are specified for *Oper* = SMULT.

## Menu Paths

**Main Menu>General Postproc>Surface Operations>Math Operations>Cross Product**

**Main Menu>General Postproc>Surface Operations>Math Operations>Dot Product**

**Main Menu>General Postproc>Surface Operations>Math Operations>Scale Result**

---

**SV**, *DAMP*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*, *SV8*, *SV9*

**Defines spectrum values to be associated with frequency points.**

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### **DAMP**

Damping ratio for this response spectrum curve. If the same as a previously defined curve, the *SV* values are added to the previous curve. Up to four different curves may be defined, each with a different damping ratio. Damping values must be input in ascending order.

**SV1**, **SV2**, **SV3**, . . . , **SV9**

Spectrum values corresponding to the frequency points [**FREQ**]. Values are interpreted as defined with the **SVTYP** command. *SV* values should not be zero. Values required outside the frequency range use the extreme input values.

## Notes

Defines the spectrum values to be associated with the previously defined frequency points [**FREQ**]. Applies only to the single-point response spectrum. Damping has no effect on the frequency solution. Damping values are used only to identify *SV* curves for the mode combinations calculation. Only the curve with the lowest damping value is used in the initial mode coefficient calculation. Use **STAT** command to list current spectrum curve values.

Repeat **SV** command for additional *SV* points (100 maximum per *DAMP* curve). *SV* values are added to the *DAMP* curve after the last nonzero *SV* value.

The interpolation method between response spectrum points and curves is specified using *KeyInterp* in the **SVTYP** command. It is logarithmic by default.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Spectr Values**  
**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Spectr Values**

**SVPLOT**, *OptionScale*, *damp1*, *damp2*, *damp3*, *damp4*

**Displays input spectrum curves.**

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### **OptionScale**

Flag to activate or deactivate input spectrum value scaling:

#### **OFF**

Do not scale the input spectrum values with scale factor **FACT** (**SVTYP** command). This is the default value.

#### **ON**

Scale the input spectrum values with scale factor **FACT** (**SVTYP** command)

**damp1**

Damping ratio corresponding to DAMP (**SV** command) defining the first spectrum curve.

**damp2**

Damping ratio corresponding to DAMP (**SV** command) defining the second spectrum curve.

**damp3**

Damping ratio corresponding to DAMP (**SV** command) defining the third spectrum curve.

**damp4**

Damping ratio corresponding to DAMP (**SV** command) defining the fourth spectrum curve.

## Notes

You can display up to four input spectrum tables (**SV** and **FREQ** commands) with log X scale. If no damping ratio is specified, all spectrum tables are displayed.

This command is valid in any processor.

## Menu Paths

To be supplied.

---

### **SVTYP**, *KSV*, *FACT*, *KeyInterp*

**Defines the type of single-point response spectrum.**

[SOLUTION: Spectrum Options](#)

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

**KSV**

Response spectrum type:

**0**

Seismic velocity response spectrum loading (SV values interpreted as velocities with units of length/time).

**1**

Force response spectrum loading (SV values interpreted as force amplitude multipliers).

**2**

Seismic acceleration response spectrum loading (SV values interpreted as accelerations with units of length/time<sup>2</sup>).

**3**

Seismic displacement response spectrum loading (SV values interpreted as displacements with units of length).

**4**

PSD loading (SV values interpreted as acceleration<sup>2</sup>/(cycles/time), such as (in/sec<sup>2</sup>)<sup>2</sup>/Hz (not g<sup>2</sup>/Hz)). (Not recommended)

**FACT**

Scale factor applied to spectrum values (defaults to 1.0). Values are scaled when the solution is initiated [**SOLVE**]. Database values remain the same.



**KeyInterp**

Key to activate or deactivate the linear interpolation between input response spectrum points and input response spectrum curves:

**0 (OFF or NO)**

Deactivate linear and use logarithmic interpolation. This value is the default.

**1 (ON or YES)**

Activate linear interpolation.

**Command Default**

Seismic velocity response spectrum.

**Notes**

Defines the type of single-point response spectrum [**SPOPT**]. The seismic excitation direction is defined with the **SED** command.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Settings**

**Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Settings**

**SWADD**, *Ecomp*, *SHRD*, *NCM1*, *NCM2*, *NCM3*, *NCM4*, *NCM5*, *NCM6*, *NCM7*, *NCM8*, *NCM9*

**Adds more surfaces to an existing spot weld set.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***Ecomp***

Name of an existing spot weld set that was previously defined using **SWGEN**.

***SHRD***

Search radius. Defaults to 4 times the spot weld radius defined for the spot weld set (*SWRD* on **SWGEN**).

***NCM1*, *NCM2*, *NCM3*, . . . , *NCM9***

Surfaces to be added to the spot weld set. Each surface can be input as a predefined node component or a meshed area number.

**Notes**

This command adds surfaces to an existing spot weld set defined by the **SWGEN** command. You can add additional surfaces by repeating the **SWADD** command. However, the maximum number of allowable surfaces (including the 2 surfaces used for the original set defined by **SWGEN**) for each spot weld set is 11. See [Adding Surfaces to a Basic Set](#) for more information.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Add More Surfaces>By Areas**

**SWDEL**, *Ecomp***Deletes spot weld sets.**[PREP7: Elements](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

***Ecomp***

Name of an existing spot weld set that was previously defined using **SWGEN**. If *Ecomp* = ALL (default) all spot welds are deleted.

**Notes**

This command deletes spot weld sets previously defined by the **SWGEN** command.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Delete**

**SWGEN**, *Ecomp*, *SWRD*, *NCM1*, *NCM2*, *SND1*, *SND2*, *SHRD*, *DIRX*, *DIRY*, *DIRZ*, *ITTY*, *ICTY***Creates a new spot weld set.**[PREP7: Elements](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

***Ecomp***

Name to identify the new spot weld. This name will be used for the element component containing the new contact, target, and beam elements generated for the spot weld set.

***SWRD***

Spot weld radius.

***NCM1***

Name of a component containing nodes on the first spot weld surface, or a meshed area number for the surface.

***NCM2***

Name of a component containing nodes on the second spot weld surface, or a meshed area number for the surface.

***SND1***

Node number of the first spot weld node corresponding to the first surface (*NCM1*). This node can be on or close to the first surface.

***SND2***

Node number of the second spot weld node corresponding to the second surface (*NCM2*). This node can be on or close to the second surface. ANSYS will create the node if it is not specified.

***SHRD***

Search radius. Defaults to 4 times the spot weld radius *SWRD*.

***DIRX*, *DIRY*, *DIRZ***

Spot weld projection direction in terms of normal X, Y, and Z components.

**ITTY**

Target element type ID.

**ICTY**

Contact element type ID.

## Notes

This command creates a new spot weld set. You can add more surfaces to the set using **SWADD** after the initial **SWGEN** command. However, the maximum number of allowable surfaces (including the 2 surfaces used for the original set) for each spot weld set is 11.

*Ecomp*, *SWRD*, *NCM1*, *NCM2*, and *SND1* must be specified. *SND2*, *SHRD*, *DIRX*, *DIRY*, *DIRZ*, *ITTY*, *ICTY* are optional inputs. If the second spot weld node (*SND2*) is specified, that node is used to determine the spot weld projection direction, and *DIRX*, *DIRY* and *DIRZ* are ignored.

If *ITTY* is specified, the following corresponding target element key option must be set: KEYOPT(5) = 4. If *ICTY* is specified, the following corresponding contact element key options must be set: KEYOPT(2) = 2, KEYOPT(12) = 5.

Use the **SWLIST** and **SWDEL** commands to list or delete spot welds. See [Creating a Basic Spot Weld Set with SWGEN](#) for detailed information on defining spot welds.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Add More Surfaces>By Node Component**

**Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Create New Set>By Area**

**Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Create New Set>By Node Component**

---

## SWLIST, *Ecomp*

**Lists spot weld sets.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***Ecomp***

Name of an existing spot weld set that was previously defined using **SWGEN**. If *Ecomp* = ALL (default), all spot weld sets are listed.

## Notes

This command lists spot weld node, beam, and contact pair information for all defined spot weld sets, or for the specified set. To ensure that all defined spotwelds are listed, issue **CMSEL,ALL** (to select all components) before issuing the **SWLIST** command.

When **SWLIST** is issued in POST1, the beam forces and moments are output. For the case of a deformable spot weld, the stresses are also output in the beam local coordinate system.

## Menu Paths

Main Menu>General Postproc>List Results>SpotWeld Solution  
 Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>List  
 Utility Menu>List>Results>SpotWeld Solution

## SYNCHRO, *RATIO*, *Cname*

Specifies whether the excitation frequency is synchronous or asynchronous with the rotational velocity of a structure.

SOLUTION: Inertia

MP ME ST <> <> <> <> EM <> <> PP <> EME MFS

### *RATIO*

The ratio between the frequency of excitation and the frequency of the rotational velocity of the structure. This value must be greater than 0. The default is an unbalance excitation ( $RATIO = 1.0$ ).

### *Cname*

The name of the rotating component on which to apply the harmonic excitation.

## Notes

The **SYNCHRO** command specifies whether the excitation frequency is synchronous or asynchronous with the rotational velocity of a structure in a harmonic analysis. Use the command to take into account rotating harmonic forces on rotating structures.

ANSYS calculates the rotational velocity  $\Omega$  of the structure from the excitation frequency  $f$ , defined (via the **HARFRQ** command) as  $\Omega = 2\pi f / RATIO$ . The rotational velocity is applied along the direction cosines of the rotation axis (specified via an **OMEGA** or **CMOMEGA** command).

Specifying any value for *RATIO* causes a general rotational force excitation and not an unbalance force. To define an unbalance excitation force ( $F = \Omega^2 * Unb$ ), *RATIO* should be left blank (the nodal unbalance *Unb* is specified via the **F** command).

The **SYNCHRO** command is valid only for a full-solution harmonic analysis (**HROPT,Method** = FULL) and the Variational Technology method (**HROPT,Method** = VT) involving a rotating structure (**OMEGA** or **CMOMEGA**) with Coriolis enabled in a *stationary* reference frame (**CORIOLIS,,,,RefFrame** = ON).

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>Synchronous  
 Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>Synchronous

**/SYP**, *String*, ARG1, ARG2, ARG3, ARG4, ARG5, ARG6, ARG7, ARG8

**Passes a command string and arguments to the operating system.**

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *String*

Command string (cannot include commas). See also the **/SYS** command.

### **ARG1, ARG2, ARG3, . . . , ARG8**

Arguments to be appended to the command string, separated by blanks, commas, or other delimiter characters (see the *Operations Guide*). The arguments may be numbers, parameters, or parametric expressions.

## Notes

Passes a command string to the operating system for execution, along with arguments to be appended to the command string. See the *Operations Guide* for details. ANSYS may not be aware of your specific user environment. In particular, this command may not recognize UNIX aliases, depending on the hardware platform and user environment.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

**/SYS**, *String*

**Passes a command string to the operating system.**

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *String*

Command string, up to 639 characters (including blanks, commas, etc.). The specified string is passed verbatim to the operating system, i.e., no parameter substitution is performed.

## Notes

Passes a command string to the operating system for execution (see the *Operations Guide*). Typical strings are system commands such as list, copy, rename, etc. Control returns to the ANSYS program after the system procedure is completed. ANSYS may not be aware of your specific user environment. In particular, this command may not recognize UNIX aliases, depending on the hardware platform and user environment.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**



---

## T Commands

**TALLOW**, *TEMP1*, *TEMP2*, *TEMP3*, *TEMP4*, *TEMP5*, *TEMP6*

Defines the temperature table for safety factor calculations.

POST1: Element Table

MP ME ST PR PRN <> <> <> <> <> DY PP <> EME MFS

*TEMP1*, *TEMP2*, *TEMP3*, . . . , *TEMP6*

Input up to six temperatures covering the range of nodal temperatures. Temperatures must be input in ascending order.

### Notes

Defines the temperature table for safety factor calculations [**SFACT**, **SALLOW**]. Use **STAT** command to list current temperature table. Repeat **TALLOW** command to zero table and redefine points (6 maximum).

Safety factor calculations are not supported by PowerGraphics. Both the **SALLOW** and **TALLOW** commands must be used with the Full Model Graphics display method active.

### Menu Paths

Main Menu>General Postproc>Safety Factor>Allowable Strs>Reset Temps  
Main Menu>General Postproc>Safety Factor>Allowable Strs>Temp-depend

---

**\*TAXIS**, *ParmLoc*, *nAxis*, *Val1*, *Val2*, *Val3*, *Val4*, *Val5*, *Val6*, *Val7*, *Val8*, *Val9*, *Val10*

Defines table index numbers.

APDL: Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Argument Descriptions

*ParmLoc*

Name and starting location in the table array parameter for indexing. Indexing occurs along the axis defined with *nAxis*.

*nAxis*

Axis along which indexing occurs. Valid labels are:

**1** --

Corresponds to Row. Default.

**2** --

Corresponds to Column.

**3** --  
Corresponds to Plane.

**4** --  
Corresponds to Book.

**5** --  
Corresponds to Shelf.

**ALL** --  
Lists all index numbers. Valid only if *Val1* = LIST.

***Val1, Val2, Val3, ..., Val10***

Values of the index numbers for the axis *nAxis*, starting from the table array parameter location *ParmLoc*. You can define up to ten values.

To list the index values specified with *nAxis*, issue *Val1* = LIST. If *Val1* = LIST, *Val2* - *Val10* are ignored.

## Notes

**\*TAXIS** is a convenient method to define table index values. These values reside in the zero column, row, etc. Instead of filling values in these zero location spots, use the **\*TAXIS** command. For example,

```
*TAXIS, longtable(1,4,1,1),2,1.0,2.2,3.5,4.7,5.9
```

would fill index values 1.0, 2.2, 3.5, 4.7, and 5.9 in *nAxis* 2 (column location), starting at location 4.

To list index numbers, issue **\*TAXIS**, *ParmLoc*, *nAxis*, LIST, where *nAxis* = 1 through 5 or ALL.

## Menu Paths

**This command cannot be accessed from a menu.**

**TB**, *Lab*, *MAT*, *NTEMP*, *NPTS*, *TBOPT*, *EOSOPT*, *FuncName*

**Activates a data table for nonlinear material properties or special element input.**

PREP7: Data Tables

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS  
Product Restrictions

**Lab**

Type of data table:

### AHYPER

Anisotropic Hyperelasticity model (SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SHELL208, SHELL209, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290. See *AHYPER Specifications* (p. 1680) for more information.

### ANEL

Anisotropic elastic matrix (SOLID5, PLANE13, SOLID98, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290, and explicit dynamic elements SOLID164 and SOLID168). Also PLANE223, SOLID226, SOLID227. See *ANEL Specifications* (p. 1677) for more information.



**ANISO**

Anisotropic plasticity (SOLID62, SOLID65, and PLANE183). See *ANISO Specifications* (p. 1671) for more information.

**BB**

Bergstrom-Boyce (PLANE182, PLANE183, SHELL181, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290). See *Bergstrom-Boyce Specifications* (p. 1681) for more information.

**BH**

Magnetic field data (SOLID5, PLANE13, PLANE53, SOLID62, SOLID96, SOLID97, SOLID98, SOLID117, PLANE233, SOLID236, SOLID237). See *BH Specifications* (p. 1685) for more information.

**BISO**

Bilinear isotropic hardening using von Mises or Hill plasticity.

For von Mises plasticity: SOLID62, SOLID65, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290, and explicit dynamic elements PLANE162, SHELL163, SOLID164, and SOLID168.

For Hill plasticity: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290.

See *BISO Specifications* (p. 1669) for more information.

**BKIN**

Bilinear kinematic hardening using von Mises or Hill plasticity

For von Mises plasticity: SOLID62, SOLID65, LINK180, SHELL181, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SHELL281, PIPE288, PIPE289, ELBOW290, PLANE182, and PLANE183, and explicit dynamic elements LINK160, BEAM161, PLANE162, SHELL163, SOLID164, and SOLID168.

For Hill plasticity: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290. See *BKIN Specifications* (p. 1668) for more information.

**CAST**

Cast iron material model (PLANE182 (not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185, SOLID186, SOLID187, SOLSH190, PLANE223, SOLID226, SOLID227, SOLID272, SOLID273, SOLID285, PIPE288, and PIPE289). See *CAST Specifications* (p. 1672) for more information.

**CDM**

Mullins effect for Isotropic Hyperelasticity models (SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290). See *Mullins Effect Specifications* (p. 1684) for more information.

**CHABOCHE**

Chaboche nonlinear kinematic hardening using von Mises or Hill plasticity (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290). See *CHABOCHE Specifications* (p. 1668) for more information.

**CNDE**

Anisotropic electric current conductivity (HF119, HF120). See *CNDE Specifications* (p. 1686) for more information.

**CNDM**

Anisotropic magnetic current conductivity (HF119, HF120). See *CNDM Specifications* (p. 1686) for more information.

**COMP**

Composite material models (explicit dynamic elements PLANE162, SHELL163, SOLID164, and SOLID168). See *COMP Specifications* (p. 1689) for more information.

**CONCR**

Concrete element data (SOLID65) or concrete damage model (explicit dynamic elements SOLID164 and SOLID168). See *CONCR Specifications* (p. 1672) for more information.

**CREEP**

Viscoplasticity/creep. You can model pure creep, creep with isotropic hardening plasticity, or creep with kinematic hardening plasticity using both von Mises or Hill potentials. See *Material Model Combinations* in the *Element Reference* for further information on combining models.

For explicit creep with von Mises potential: SOLID62 and SOLID65.

For implicit creep with von Mises or Hill potential: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SOLID285, SHELL281, PIPE288, PIPE289, and ELBOW290.

See *CREEP Specifications* (p. 1679) for more information.

**CZM**

Cohesive zone material model (contact elements CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177, and interface elements INTER202, INTER203, INTER204, INTER205). See *CZM Specifications* (p. 1693) for more information.

**DISCRETE**

Explicit spring-damper (discrete) material models (COMBI165). See *DISCRETE Specifications* (p. 1689) for more information.

**DP**

Drucker-Prager plasticity (SOLID62, SOLID65, and PLANE83). See *DP Specifications* (p. 1672) for more information.

**DPER**

Anisotropic electric permittivity ( HF118, HF119, HF120, PLANE223, SOLID226, SOLID227). See *DPER Specifications* (p. 1686) for more information.

**EDP**

Extended Drucker-Prager model for granular materials such as rock, concrete, soil, ceramics and other pressure dependent materials. (PLANE182(not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185, SOLID186, SOLID187, SOLSH190, PLANE223, SOLID226, SOLID227, SOLID272, SOLID273, SOLID285, PIPE288, and PIPE289). See *EDP Specifications* (p. 1673) for more information.

**ELASTIC**

Elastic material property (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285,

PIPE288, PIPE289, and ELBOW290). Elastic properties can be defined as frequency or temperature dependent properties for use in full harmonic analyses (use **TBFIELD**). See *ELASTIC Specifications* (p. 1677) for more information.

**EOS**

Equation of state (explicit dynamic elements only). See *EOS Specifications* (p. 1690) for more information.

**EVISC**

Viscoelastic element data (via explicit dynamic elements BEAM161, PLANE162, SOLID164, and SOLID168). See *EVISC Specifications* (p. 1690) for more information.

**EXPE**

Experimental data. See *EXPE Specifications* (p. 1693) for more information.

**FCON**

Fluid conductance data (FLUID116). See *FCON Specifications* (p. 1691) for more information.

**FCLI**

Material strength limits for calculating failure criteria. See *FCLI Specifications* (p. 1694) for more information.

**FLUID**

Fluid material models (valid for hydrostatic fluid elements HSFLD241, and HSFLD242). See *FLUID Specifications* (p. 1694) for more information.

**FOAM**

Foam material models (explicit dynamic elements PLANE162, SOLID164, and SOLID168). See *FOAM Specifications* (p. 1691) for more information.

**FRIC**

Coefficient of friction based on Coulomb's Law (CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177, and CONTA178) or user defined friction properties (CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA177, and CONTA178). See *FRIC Specifications* (p. 1695) for more information.

**GASKET**

Gasket material model (INTER192, INTER193, INTER194, and INTER195). See *GASKET Specifications* (p. 1695) for more information.

**GCAP**

Geological Cap material model (explicit dynamic elements SOLID164 and SOLID168). See *GCAP Specifications* (p. 1691) for more information.

**GURSON**

Gurson pressure-dependent plasticity model for porous metals (PLANE182 (not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185, SOLID186, SOLID187, SOLSH190, PLANE223, SOLID226, SOLID227, SOLID272, SOLID273, and SOLID285). See *GURSON Specifications* (p. 1673) for more information. Also see Gurson-Chaboche Material Model in the *Structural Analysis Guide*.

**HFFDL**

Frequency-dependent lossy dielectric (HF119 and HF120). See *HFFDL Specifications* (p. 1686) for more information.

**HFLM**

Film coefficient data (FLUID116). See *HFLM Specifications* (p. 1687) for more information.

**HILL**

Hill anisotropy when combined with other material options simulates plasticity, viscoplasticity, and creep -- all with the Hill potential. See [Material Model Combinations](#) in the *Element Reference* for further information on combining models (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290). See *HILL Specifications* (p. 1671) for more information.

**HONEY**

Honeycomb material models (explicit dynamic elements PLANE162, SOLID164, and SOLID168). See *HONEY Specifications* (p. 1692) for more information.

**HYPER**

Hyperelasticity models (Arruda-Boyce, Blatz-Ko, Extended Tube, Gent, Mooney-Rivlin (default), Neo-Hookean, Ogden, Ogden Foam, Polynomial Form, Response Function, Yeoh, and user-defined SHELL181, PIPE288, PIPE289, ELBOW290, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, SOLID272, SOLID273, SHELL281, and SOLID285]). See *HYPER Specifications* (p. 1681) for more information.

**JOIN**

Linear and nonlinear elastic stiffness, linear and nonlinear damping, and frictional behavior (MPC184). See *JOIN Specifications* (p. 1674) for more information.

**KINH**

Multilinear kinematic hardening using von Mises or Hill plasticity.

For von Mises plasticity: PLANE13, SOLID62, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SHELL281, PIPE288, PIPE289, and ELBOW290.

For Hill plasticity: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, and ELBOW290. KINH is the same as MKIN with *TBOPT* = 2, but with fewer restrictions on the number of points per curve and the number of temperatures. See *KINH Specifications* (p. 1669) for more information.

**LSEM**

Anisotropic electric and magnetic loss tangents (HF119 and HF120). See *LSEM Specifications* (p. 1687) for more information.

**MELAS**

Multilinear elasticity (SOLID62 and SOLID65). See *MELAS Specifications* (p. 1678) for more information.

**MISO**

Multilinear isotropic hardening using von Mises or Hill plasticity.

For von Mises plasticity: SOLID62, SOLID65, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290.

For Hill plasticity: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, and ELBOW290.

See *MISO Specifications* (p. 1670) for more information.

**MKIN**

Multilinear kinematic hardening using von Mises or Hill plasticity.

For von Mises plasticity: SOLID62, SOLID65, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290.

For Hill plasticity: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, and ELBOW290.

See *MKIN Specifications* (p. 1669) for more information.

**MOONEY**

Mooney-Rivlin hyperelastic element data (explicit dynamic elements PLANE162, SHELL163, SOLID164, and SOLID168). See *MOONEY Specifications* (p. 1683) for more information.

**MUR**

Anisotropic relative permeability ( HF118, HF119, HF120). See *MUR Specifications* (p. 1687) for more information.

**NLISO**

Voce isotropic hardening law or power law for modeling nonlinear isotropic hardening using von Mises or Hill plasticity (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290). See *NLISO Specifications* (p. 1670) for more information.

**PIEZ**

Piezoelectric matrix (SOLID5, PLANE13, SOLID98). Also PLANE223, SOLID226, SOLID227. See *PIEZ Specifications* (p. 1688) for more information.

**PLASTIC**

Nonlinear plasticity with stress-vs.-plastic strain data (LINK180, SHELL181, PIPE288, PIPE289, ELBOW290, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, and SOLID285). See *PLASTIC Specifications* (p. 1670) for more information.

**PLAW**

Plasticity laws (explicit dynamic elements LINK160, BEAM161, PLANE162, SHELL163, SOLID164, and SOLID168). See *PLAW Specifications* (p. 1692) for more information.

**PM**

Porous media material model (coupled elements CPT212, CPT213, CPT215, CPT216, and CPT217). See *PM Specifications* (p. 1688) for more information.

**PRONY**

Prony series constants for viscoelastic materials (LINK180, SHELL181, PIPE288, PIPE289, ELBOW290, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, and SOLID285). See *PRONY Specifications* (p. 1684) for more information.

**PZRS**

Piezoresistive materials (PLANE223, SOLID226, SOLID227). See *PZRS Specifications* (p. 1688) for more information.

**RATE**

Rate-dependent plasticity (viscoplasticity) when combined with the BISO, MISO, NLISO or PLASTIC material options, or rate-dependent anisotropic plasticity (anisotropic viscoplasticity) when combined with the HILL and BISO, MISO, NLISO or PLASTIC material options (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290). For more information about combining material models, see *Material Model Combinations* in the *Element Reference*.

The exponential visco-hardening option includes an explicit function for directly defining static yield stresses of materials.

An Anand unified plasticity model is also available, requiring no combination with other material models and supporting the following elements: PLANE182 and PLANE183 (neither applicable for plane stress), SOLID185, SOLID186, SOLID187, SOLSH190, SOLID272, SOLID273, SOLID285, PIPE288, and PIPE289.

See *RATE Specifications* (p. 1679) for more information.

**SDAMP**

Material structural damping coefficient (SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, SOLID272, SOLID273, SHELL281, SOLID285, and ELBOW290). Damping coefficients can be defined as frequency or temperature dependent properties for use in full harmonic analyses (use **TBFIELD**). SDAMP specifies damping in terms of the loss factor, which is equal to 2x the damping ratio. See *SDAMP Specifications* (p. 1678) and *Damping Matrices* for more information.

For the relationship between SDAMP, **DMPRAT**, and **MP,DAMP**, see "Notes" (p. 1696)

**Note**

When specifying frequency dependent damping using **TB,SDAMP** you must specify the material property using **TB,ELAS**.

**SHIFT**

Shift function for viscoelastic materials (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE223, SOLID226, SOLID227, PIPE288, PIPE289, ELBOW290, REINF264, REINF265, SOLID272, SOLID273, SHELL281, and SOLID285). See *SHIFT Specifications* (p. 1685) for more information.

**SMA**

Shape memory alloy for simulation of hysteresis superelastic behavior with no performance deformation. Plane stress is not supported. See *Shape Memory Alloys* in the *Element Reference*, and *Shape Memory Alloy* in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for further information. The **TB,SMA** option is valid for PLANE182, PLANE183, and PLANE223 (with plane strain or axisymmetric stress states), and for SOLID185, SOLID186, SOLID187, SOLSH190, SOLID226, SOLID227, SOLID272, SOLID273, and SOLID285. See *SMA Specifications* (p. 1676) for more information.

**STATE**

User-defined state variables, used with **TB,USER** or **TB,CREEP** [when the USER CREEP option is chosen (*TB OPT* = 100)]. When used with **TB,USER**, **TB,STATE** is only used with the USERMAT subroutine (*not* USERPL). When used with **TB,CREEP**, **TB,STATE** is only used with the USERCREEP subroutine. See *STATE Specifications* (p. 1696) for more information.



**SWELL**

Swelling constants ([SOLID62](#) and [SOLID65](#)). See [SWELL Specifications](#) (p. 1680) for more information.

**UNIAXIAL**

Uniaxial stress-strain relation associated with the Cast Iron material model ([PLANE182](#), [PLANE183](#), and [PLANE223](#) (not applicable for plane stress), [SOLID185](#), [SOLID186](#), [SOLID187](#), [SOLSH190](#), [SOLID226](#), [SOLID227](#), [SOLID272](#), [SOLID273](#), and [SOLID285](#)). See [UNIAXIAL Specifications](#) (p. 1676) for more information.

**USER**

User-defined material model; general purpose except for incompressible material models ([LINK180](#), [SHELL181](#), [PLANE182](#), [PLANE183](#), [SOLID185](#), [SOLID186](#), [SOLID187](#), [BEAM188](#), [BEAM189](#), [SOLSH190](#), [SHELL208](#), [SHELL209](#), [REINF264](#), [REINF265](#), [SOLID272](#), [SOLID273](#), [SHELL281](#), [SOLID285](#), [PIPE288](#), [PIPE289](#), and [ELBOW290](#)).

Also, user-defined plasticity or viscoplasticity ([SOLID62](#), [SOLID65](#), and [PLANE183](#)).

See [USER Specifications](#) (p. 1696) for more information.

**MAT**

Material reference number. The default value is 1.

**NTEMP**

The number of temperatures for which data will be provided (if applicable). Specify temperatures via the [TBTEMP](#) command.

**NPTS**

For most labels where *NPTS* is defined, the number of data points to be specified for a given temperature. Define data points via the [TBDATA](#) or [TBPT](#) commands.

**EOSOPT**

Indicates which equation of state model will be used. Used only for explicit dynamics, and only when *Lab* = EOS.

- 1** Linear polynomial equation of state
- 2** Gruneisen equation of state
- 3** Tabulated equation of state

**FuncName**

The name of the function to be used (entered as *%tablename%*, where *tablename* is the name of the table created by the Function Tool). Valid only when *Lab* = JOIN (joint element material) and nonlinear stiffness or damping are specified on the *TBOPT* field (see [JOIN Specifications](#) (p. 1674)). The function must be previously defined using the Function Tool. To learn more about how to create a function, see "Using the Function Tool" in the [Basic Analysis Guide](#).

---

## Data Table Specifications

Following is a listing of all valid **TB** command labels (*Lab* values). For each material type, the data table includes requirements for the *NTEMP*, *NPTS*, and *TBOPT* options, along with links to more detailed documentation if needed.

Kinematic Hardening Tables  
Isotropic Hardening Tables  
Anisotropic Plasticity Tables  
Nonmetal and Other Plasticity Tables  
Elasticity Tables  
Rate-Dependent Plasticity Tables  
Hyperelasticity Tables  
Viscoelasticity Tables  
Multiphysics Tables  
Porous Media Tables  
ANSYS LS-DYNA Tables  
Special Material Tables  
User Tables

---

## ***Kinematic Hardening Tables***

### **BKIN Specifications**

***NTEMP:***

Number of temperatures for which data will be provided. Default = 6, Maximum = 6

***NPTS:***

Not used.

***TBOPT:***

Stress-strain options (not used in an explicit dynamics analysis).

**0** --

No stress relaxation with temperature increase (this is not recommended for nonisothermal problems).

**1** --

Rice's hardening rule, which takes into account stress relaxation with increasing temperature. This value is the default..

**References:**

Bilinear Kinematic Hardening Constants (TB,BKIN) in the *Element Reference*.

Plastic Material Options in the *Structural Analysis Guide*.

Nonlinear Inelastic Models in the *ANSYS LS-DYNA User's Guide*.

### **CHABOCHE Specifications**

***NTEMP:***

Number of temperatures for which data will be provided. Default = 1, Maximum value of *NTEMP* is such that  $NTEMP \times (1 + 2NPTS) = 1000$

***NPTS:***

Number of kinematic models to be superposed. Default = 1, Maximum value of *NPTS* is such that  $NTEMP \times (1 + 2NPTS) = 1000$

***TBOPT:***

Not used.

**References:**

Nonlinear Kinematic Hardening Constants (TB,CHABOCHE) in the *Element Reference*.



Plastic Material Options in the *Structural Analysis Guide*.

## KINH Specifications

### **NTEMP:**

Number of temperatures for which data will be provided. Default = 1, Maximum = 40

### **NPTS:**

Number of data points to be specified for a given temperature. Default = 20, Maximum = 20

### **TBOPT:**

Use 0 or leave blank to define stress -vs- total strain curve.

Use 4 or enter "PLASTIC" to define stress -vs- plastic strain curve. This option supports only elements LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, REINF264, REINF265, SOLID272, SOLID273, SOLID285, SHELL281, PIPE288, PIPE289, and ELBOW290, .

### **References:**

Multilinear Kinematic Hardening Constants (TB,KINH or TB,MKIN) in the *Element Reference*.

Plastic Material Options in the *Structural Analysis Guide*.

## MKIN Specifications

### **NTEMP:**

Number of temperatures for which data will be provided. Default = 5, Maximum = 5

### **NPTS:**

Not used.

### **TBOPT:**

Stress-strain options.

#### **0 --**

No stress relaxation with temperature increase (this is not recommended for nonisothermal problems); also produces thermal ratcheting. This value is the default.

#### **1 --**

Recalculate total plastic strain using new weight factors of the subvolume.

#### **2 --**

Scale layer plastic strains to keep total plastic strain constant; agrees with Rice's model (TB, BKIN with TBOPT = 1). Produces stable stress-strain cycles.

### **References:**

Multilinear Kinematic Hardening Constants (TB,KINH or TB,MKIN) in the *Element Reference*.

Plastic Material Options in the *Structural Analysis Guide*.

---

## **Isotropic Hardening Tables**

### BISO Specifications

#### **NTEMP:**

Number of temperatures for which data will be provided. Default = 6, Maximum = 6

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

Bilinear Isotropic Hardening Constants (TB,BISO) in the *Element Reference*.

Plastic Material Options in the *Structural Analysis Guide*.

Nonlinear Inelastic Models in the *ANSYS LS-DYNA User's Guide*.

## MISO Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1, Maximum = 20

**NPTS:**

Number of data points to be specified for a given temperature. Default = 20, Maximum = 100

**TBOPT:**

Not used.

**References:**

Multilinear Isotropic Hardening Constants (TB,MISO) in the *Element Reference*.

Plastic Material Options in the *Structural Analysis Guide*.

## NLISO Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1, Maximum = 20

**NPTS:**

Number of data points to be specified for a given temperature. Default = 4, Maximum = 4

**TBOPT:**

Isotropic hardening options.

**VOCE --**

Voce hardening law. This value is the default.

**POWER --**

Power hardening law..

**References:**

Nonlinear Isotropic Hardening Constants (TB,NLISO) in the *Element Reference*.

Plastic Material Options in the *Structural Analysis Guide*.

## PLASTIC Specifications

**NTEMP:**

The number of temperature points (default = 1). You can specify up to 20.

**NPTS:**

The number of stress versus plastic strain data points (default = 20). You can specify up to 100.

**TBOPT:**

Type of plastic hardening:

**MISO --**

Multilinear isotropic hardening plasticity.

**KINH --**

Multilinear kinematic hardening plasticity.

**References:**

Plasticity in the *Structural Analysis Guide*.

Multilinear Kinematic Hardening in the *Element Reference*.

Multilinear Isotropic Hardening in the *Element Reference*.

---

**Anisotropic Plasticity Tables****ANISO Specifications****NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

Anisotropic Constants (TB,ANISO) in the *Element Reference*.

Plastic Material Options in the *Structural Analysis Guide*.

**HILL Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1, Maximum = 40

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

Hill's Anisotropy Constants (TB,HILL) in the *Element Reference*.

Plastic Material Options in the *Structural Analysis Guide*.

---

## ***Nonmetal and Other Plasticity Tables***

### **CAST Specifications**

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1; Max = 10.

**NPTS:**

Not used.

**TBOPT:**

Defines hardening type.

**ISOTROPIC --**

Specifies cast iron plasticity with isotropic hardening.

**References:**

Cast Iron Plasticity Material Constants (TB,CAST) in the *Element Reference*.

### **CONCR Specifications**

**NTEMP:**

Number of temperatures for which data will be provided (used only if TBOPT = 0 or 1). Default = 6, Maximum = 6

**NPTS:**

Not used.

**TBOPT:**

Concrete material options.

**0 or 1 --**

General concrete option for element [SOLID65](#).

**2 --**

Concrete damage model for explicit dynamic elements [SOLID164](#) and [SOLID168](#).

**References:**

[SOLID65](#) in the *Element Reference*.

Concrete Damage Model in the *ANSYS LS-DYNA User's Guide*.

### **DP Specifications**

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

Drucker-Prager Constants (TB,DP) in the *Element Reference*.

Plastic Material Options in the *Structural Analysis Guide*.

## EDP Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1.

**NPTS:**

Number of data points to be specified for a given temperature.

**TBOPT:**

EDP material options.

**LYFUN --**

Linear yield function.

**PYFUN --**

Power law yield function.

**HYFUN --**

Hyperbolic yield function.

**LFPOT --**

Linear flow potential function.

**PFPOT --**

Power law flow potential function.

**HPOT --**

Hyperbolic flow potential function.

**CYFUN --**

Cap yield function.

**CFPOT --**

Cap flow potential function.

**References:**

See [Extended Drucker-Prager](#) in the *Element Reference*.

Also see [Plastic Material Options](#) in the *Structural Analysis Guide*.

## GURSON Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1.

**NPTS:**

Number of data points to be specified for a given temperature.

**TBOPT:**

GURSON material options.

**BASE --**

Basic model without nucleation or coalescence. This value is the default.

**SNU --**

Strain controlled nucleation.

**SSNU --**

Stress controlled nucleation.

**COAL --**

Coalescence

**References:**

See [Gurson's Model](#) in the *Element Reference*.

Also see [Plastic Material Options](#) in the *Structural Analysis Guide*.

**JOIN Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1.

**NPTS:**

Number of data points to be specified for a given temperature. *NPTS* is ignored if *TBOPT* = STIF or DAMP.

If Coulomb friction is specified, *NPTS* is used only for *TBOPT* = MUS1, MUS4, and MUS6.

**TBOPT:**

Joint element material options.

**Linear stiffness behavior:****STIF --**

Linear stiffness.

**Nonlinear stiffness behavior:****JNSA --**

Nonlinear stiffness behavior in all available components of relative motion for the joint element.

**JNS1 --**

Nonlinear stiffness behavior in local UX direction only.

**JNS2 --**

Nonlinear stiffness behavior in local UY direction only.

**JNS3 --**

Nonlinear stiffness behavior in local UZ direction only.

**JNS4 --**

Nonlinear stiffness behavior in local ROTX direction only.

**JNS5 --**

Nonlinear stiffness behavior in local ROTY direction only.

**JNS6 --**

Nonlinear stiffness behavior in local ROTZ direction only.

**Linear damping behavior:****DAMP --**

Linear damping.

**Nonlinear damping behavior:****JNDA --**

Nonlinear damping behavior in all available components of relative motion for the joint element.

**JND1 --**

Nonlinear damping behavior in local UX direction only.

**JND2 --**

Nonlinear damping behavior in local UY direction only.

**JND3 --**

Nonlinear damping behavior in local UZ direction only.

**JND4 --**

Nonlinear damping behavior in local ROTX direction only.

**JND5 --**

Nonlinear damping behavior in local ROTY direction only.

**JND6 --**

Nonlinear damping behavior in local ROTZ direction only.

**Friction Behavior:****Coulomb friction coefficient -**

The values can be specified using either **TBDATA** ( $NPTS = 0$ ) or **TBPT** ( $NPTS$  is nonzero).

**MUS1 --**

Coulomb friction coefficient (stiction) in local UX direction only.

**MUS4 --**

Coulomb friction coefficient (stiction) in local ROTX direction only.

**MUS6 --**

Coulomb friction coefficient (stiction) in local ROTZ direction only.

**Coulomb friction coefficient - Exponential Law -**

Use **TBDATA** to specify  $\mu_s$ ,  $\mu_d$ , and  $c$  for the exponential law.

**EXP1 --**

Exponential law for friction in local UX direction only.

**EXP4 --**

Exponential law for friction in local ROTX direction only.

**EXP6 --**

Exponential law for friction in local ROTZ direction only.

**Elastic slip:****SL1 --**

Elastic slip in local UX direction only.

**SL4 --**

Elastic slip in local ROTX direction only.

**SL6 --**

Elastic slip in local ROTZ direction only.

**TMX1 --**

Critical force in local UX direction only.

**TMX4 --**

Critical moment in local ROTX direction only.

**TMX6 --**

Critical moment in local ROTZ direction only.

**Stick-stiffness:****SK1 --**

Stick-stiffness in local UX direction only.

**SK4 --**

Stick-stiffness in local ROTX direction only.

**SK6 --**

Stick-stiffness in local ROTZ direction only.

**Interference fit force/moment:****FI1 --**

Interference fit force in local UX direction only.

**FI4 --**

Interference fit moment in local ROTX direction only.

**FI6 --**

Interference fit moment in local ROTZ direction only.

**References:**

[MPC184 Joint Material Models \(TB,JOIN\)](#) in the *Element Reference*.

**SMA Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1, maximum = 40.

**NPTS:**

Not used.

**TBOPT:**

Not used

**References:**

[Shape Memory Alloy Material Model \(TB,SMA\)](#) in the *Element Reference*.

[Shape Memory Alloys](#) in the *Structural Analysis Guide*, and [Shape Memory Alloy](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

**UNIAXIAL Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1; Max = 10.

**NPTS:**

Number of data points to be specified for a given temperature. Default = 20; Max = 20.

**TBOPT:**

Defines stress-strain relationship for cast iron plasticity.

**TENSION --**

Defines stress-strain relation in tension

**COMPRESSION --**

Defines stress-strain relation in compression.



**References:**

Cast Iron Plasticity Material Constants (TB,CAST) in the *Element Reference*.

---

## Elasticity Tables

### ANEL Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 6, Maximum = 6. *NTEMP* is not used for explicit dynamic elements.

**NPTS:**

Not used.

**TBOPT:**

Anisotropic elastic matrix options.

**0** --

Elasticity matrix used as supplied (input in stiffness form).

**1** --

Elasticity matrix inverted before use (input in flexibility form). This option is not valid for explicit dynamic elements.

**References:**

Anisotropic Elastic Material Model (TB,ANEL) in the *Element Reference*.

Linear Elastic Models in the *ANSYS LS-DYNA User's Guide*.

### ELASTIC Specifications

**NTEMP:**

Not used.

**NPTS:**

Number of properties to be defined for the material option. This value is set automatically based on the elasticity option (*TBOPT*) selected. If *TBOPT* is not specified, it is set to ISOT by default and *NPTS* is set to 2.

**TBOPT:**

Elasticity options:

**ISOT** --

Isotropic property (EX, NUXY). This is the default value. *NPTS* = 2. Setting *NPTS* = 2 selects this option automatically.

**OELN** --

Orthotropic option with minor Poisson's ratio (EX, EY, EZ, GXY, GYZ, GXZ, NUXY, NUYZ, NUXZ). *NPTS* = 9. Setting *NPTS* = 9 selects this option automatically. All nine parameters must be set, even for the 2-D case.

**OELM** --

Orthotropic option with major Poisson's ratio (EX, EY, EZ, GXY, GYZ, GXZ, PRXY, PRYZ, PRXZ). *NPTS* = 9. All nine parameters must be set, even for the 2-D case.

**AELS --**

Anisotropic option in stiffness form (D11, D21, D31, D41, D51, D61, D22, D32, D42, D52, D62, D33, D43, ..... D66). *NPTS* = 21. Setting *NPTS* = 21 selects this option automatically.

**AELF --**

Anisotropic option in compliance form (C11, C21, C31, C41, C51, C61, C22, C32, C42, C52, C62, C33, C43, ..... C66). *NPTS* = 21.

**USER --**

User-defined linear elastic properties. For more information, see the documentation for the `user_tbelastic` subroutine in the [Guide to ANSYS User Programmable Features](#).

**References:**

See the **TBFIELD** command for more information on defining temperature and/or frequency dependent properties.

Full Harmonic Response Analysis in the [Structural Analysis Guide](#).

**MELAS Specifications*****NTEMP:***

Number of temperatures for which data will be provided. Default = 1, Maximum = 20

***NPTS:***

Number of data points to be specified for a given temperature. Default = 20, Maximum = 100

***TBOPT:***

Not used.

**References:**

Multilinear Elastic Constants (TB,MELAS) in the [Element Reference](#).

Multilinear Elasticity in the [Structural Analysis Guide](#).

**SDAMP Specifications*****NTEMP:***

Not used.

***NPTS:***

Number of properties to be defined for the material option.

**1 --**

Structural damping coefficient. This is the default.

***TBOPT:***

Not Used.

**References:**

See the **TBFIELD** command for more information on defining temperature and/or frequency dependent properties.

Full Harmonic Response Analysis in the [Structural Analysis Guide](#).

## Rate-Dependent Plasticity Tables

### CREEP Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1, Maximum value of *NTEMP* is such that  $NTEMP \times NPTS = 1000$  for implicit creep and 250 for explicit creep.

**NPTS:**

Number of data points to be specified for a given temperature. Default = 12 for implicit creep and 72 for explicit creep, Maximum value of *NPTS* is such that  $NTEMP \times NPTS = 1000$  for implicit creep and 250 for explicit creep.

**TBOPT:**

Creep model options.

**0 --**

(or Blank) Explicit creep option. Creep model is defined by constants  $C_6$ ,  $C_{12}$ , and  $C_{66}$ , through **TBDATA**. See [Primary Explicit Creep Equation for  \$C\_6 = 0\$](#)  through [Irradiation Induced Explicit Creep Equation for  \$C\_{66} = 5\$](#)  in the *Element Reference* for the associated equations. (Applicable to **SOLID62** and **SOLID65**.)  $C_6 = 100$  defines the USER CREEP option for explicit creep. You must define the creep law using the subroutine `USERCR.F`. See the [Guide to ANSYS User Programmable Features](#) for more information.

**1 through 13 --**

Implicit creep option. See [Table 2.4: "Implicit Creep Equations"](#) in the *Element Reference* for a list of available equations. Use **TBTEMP** and **TBDATA** to define temperature-dependent constants. (Applicable to **LINK180**, **SHELL181**, **PLANE182**, **PLANE183**, **SOLID185**, **SOLID186**, **SOLID187**, **BEAM188**, **BEAM189**, **SOLSH190**, **SHELL208**, **SHELL209**, **REINF264**, **REINF265**, **SOLID272**, **SOLID273**, **SHELL281**, **SOLID285**, **PIPE288**, **PIPE289**, and **ELBOW290**).

**100 --**

USER CREEP option (applicable to **LINK180**, **SHELL181**, **PLANE182**, **PLANE183**, **SOLID185**, **SOLID186**, **SOLID187**, **BEAM188**, **BEAM189**, **SOLSH190**, **SHELL208**, **SHELL209**, **REINF264**, **REINF265**, **SOLID272**, **SOLID273**, **SHELL281**, **SOLID285**, **PIPE288**, **PIPE289**, and **ELBOW290**). You must define the creep law using the subroutine `USERCREEP.F`. See the [Guide to ANSYS User Programmable Features](#) for more information. Use **TBTEMP** and **TBDATA** to define temperature-dependent constants. For implicit creep, use with **TB,STATE** for defining the number of state variables.

**References:**

[Creep Equations \(TB,CREEP\)](#) in the *Element Reference*.

[Creep](#) in the *Structural Analysis Guide*.

### RATE Specifications

**NTEMP:**

The number of temperatures for which data will be provided. The default is 1. The maximum *NTEMP* value is such that  $NTEMP \times NPTS = 1000$ .

**NPTS:**

The number of data points to be specified for a given temperature. The default is 2. The maximum *NPTS* value is such that  $NTEMP \times NPTS = 1000$ .

**TBOPT:**

Rate-dependent viscoplasticity options.

**PERZYNA --**

Perzyna option. This value is the default.

**PEIRCE --**

Peirce option

**EVH --**

Exponential visco-hardening option

**ANAND --**

Anand option

**References:**

Rate-Dependent Plastic (Viscoplastic) Material Models (TB,RATE) in the *Element Reference*.

Viscoplasticity in the *Structural Analysis Guide*.

Rate-Dependent Plasticity in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

**SWELL Specifications****NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

Swelling Equation Constants (TB,SWELL) in the *Element Reference*.

Swelling in the *Structural Analysis Guide*.

---



---

**Hyperelasticity Tables****AHYPER Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1.

**NPTS:**

Number of data points to be specified for a given temperature.

**TBOPT:**

Anisotropic hyperelastic material options.

**POLY --**

Anisotropic potential.

**AVEC --**

Define the A vector.

**BVEC --**

Define the B vector.

**PVOL --**  
Volumetric potential.

**References:**

Anisotropic Hyperelasticity in the *Element Reference*.

Anisotropic Hyperelasticity in the *Theory Reference for the Mechanical APDL and Mechanical Applications*

## Bergstrom-Boyce Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1. Maximum must be a value such that  $(NTEMP \times NPTS) \leq 1000$ .

**NPTS:**

Number of material constants. If  $TBOPT = ISO$ , then  $NPTS = 7$ . If  $TBOPT = 1$ , then  $NPTS = 1$ .

**TBOPT:**

Isochoric or volumetric strain-energy function:

**ISO --**

Define material constants for isochoric strain energy.

**PVOL --**

Define material constants for volumetric strain energy.

**References:**

Bergstrom-Boyce in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

Bergstrom-Boyce Material Constants (TB,BB) in the *Element Reference*.

Bergstrom-Boyce Hyperviscoelastic Material Model in the *Structural Analysis Guide*.

## HYPER Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1. Maximum value of  $NTEMP$  is such that  $NTEMP \times NPTS = 1000$ .

**NPTS:**

Number of data points to be specified for a given temperature, except for  $TBOPT = MOONEY$ , where  $NPTS$  is the number of parameters in the Mooney-Rivlin model (2 [default], 3, 5, or 9), and  $TBOPT = RESPONSE$ , where  $NPTS$  is the number of terms in the volumetric strain energy polynomial.

**TBOPT:**

Hyperelastic material options. (The default option is  $TBOPT = MOONEY$ .)

**BOYCE--**

Arruda-Boyce model. For  $NPTS$ , the default = 3 and the maximum = 3.

**References:**

Arruda-Boyce Hyperelastic Material Constants (TB,HYPER,,,,BOYCE) in the *Element Reference*.

Arruda-Boyce Hyperelastic Option in the *Structural Analysis Guide*.

**BLATZ --**

Blatz-Ko model. For  $NPTS$ , the default = 1 and the maximum = 1.

**References:**

Blatz-Ko Foam Hyperelastic Material Constants (TB,HYPER,,,,BLATZ) in the *Element Reference*.

Blatz-Ko Hyperelastic Option in the *Structural Analysis Guide*.

**ETUBE --**

Extended tube model. Five material constants ( $NPTS = 5$ ) are required.

**References:**

Extended Tube Material Constants (TB,HYPER,,,,ETUBE) in the *Element Reference*

Extended Tube Model in the *Theory Reference for the Mechanical APDL and Mechanical Applications*

**FOAM --**

Hyperfoam (Ogden) model. For  $NPTS$ , the default = 1 and the maximum is such that  $NTEMP \times NPTS \times 3 = 1000$ .

**References:**

Ogden Compressible Foam Hyperelastic Material Constants (TB,HYPER,,,,FOAM) in the *Element Reference*.

Ogden Compressible Foam Hyperelastic Option in the *Structural Analysis Guide*.

**GENT --**

Gent model. For  $NPTS$ , the default = 3 and the maximum = 3.

**References:**

Gent Hyperelastic Material Constants (TB,HYPER,,,,GENT) in the *Element Reference*.

Gent Hyperelastic Option in the *Structural Analysis Guide*.

**MOONEY --**

Mooney-Rivlin model (default). You can choose a two-parameter Mooney-Rivlin model with  $NPTS = 2$  (default), or a three-, five-, or nine-parameter model by setting  $NPTS$  equal to one of these values.

**References:**

Mooney-Rivlin Hyperelastic Material Constants (TB,HYPER,,,,MOONEY) in the *Element Reference*.

Mooney-Rivlin Hyperelastic Option in the *Structural Analysis Guide*.

**NEO --**

Neo-Hookean model. For  $NPTS$ , the default = 2 and the maximum = 2.

**References:**

Neo-Hookean Hyperelastic Material Constants (TB,HYPER,,,,NEO) in the *Element Reference*.

Neo-Hookean Hyperelastic Option in the *Structural Analysis Guide*.

**OGDEN --**

Ogden model. For  $NPTS$ , the default = 1 and the maximum is such that  $NTEMP \times NPTS \times 3 = 1000$ .

**References:**

Ogden Hyperelastic Material Constants (TB,HYPER,,,,OGDEN) in the *Element Reference*.

Ogden Hyperelastic Option in the *Structural Analysis Guide*.

#### **POLY --**

Polynomial form model. For *NPTS*, the default = 1 and the maximum is such that  $NTEMP \times NPTS = 1000$ .

#### **References:**

Polynomial Form Hyperelastic Material Constants (TB,HYPER,,,,POLY) in the *Element Reference*.

Polynomial Form Hyperelastic Option in the *Structural Analysis Guide*.

#### **RESPONSE --**

Experimental response function model. For *NPTS*, the default = 0 and the maximum is such that  $NTEMP \times NPTS + 2 = 1000$ .

#### **References:**

Response Function Hyperelastic Material Constants (TB,HYPER,,,,RESPONSE) in the *Element Reference*.

Response Function Hyperelastic Option (TB,HYPER,,,,RESPONSE) in the *Structural Analysis Guide*.

Experimental Response Functions in the *Theory Reference for the Mechanical APDL and Mechanical Applications*

#### **YEOH --**

Yeoh model. For *NPTS*, the default = 1 and the maximum is such that  $NTEMP \times NPTS \times 2 = 1000$ .

#### **References:**

Yeoh Hyperelastic Material Constants (TB,HYPER,,,,YEOH) in the *Element Reference*.

Yeoh Hyperelastic Option in the *Structural Analysis Guide*.

#### **USER --**

User-defined hyperelastic model. See the *ANSYS Guide to User Programmable Features* for details.

#### **References:**

User-Defined Hyperelastic Material Constants (TB,HYPER,,,,USER) in the *Element Reference*.

User-Defined Hyperelastic Option in the *Structural Analysis Guide*.

## **MOONEY Specifications**

#### ***NTEMP:***

Number of temperatures for which data will be provided. Default = 6, Maximum = 6.

#### ***NPTS:***

(Not used for explicit dynamic elements.)

#### ***TBOPT:***

Mooney-Rivlin material option, applicable to explicit dynamic elements [PLANE162](#), [SHELL163](#), [SOLID164](#), and [SOLID168](#).

**0 --**

Direct input of hyperelastic material constants. This value is the default.

**1 --**

Reserved for future use.

**2 --**

Material constants to be calculated by the LS-DYNA program from experimental data. This option is only valid for explicit dynamic elements.

**References:**

[Nonlinear Elastic Models](#) in the *ANSYS LS-DYNA User's Guide*.

**Mullins Effect Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1. Maximum must be a value such that  $(NTEMP \times NPTS) \leq 1000$ .

**NPTS:**

Number of data points to be specified for a given temperature.

**TBOPT:**

Mullins effect option:

**PSE2 --**

Pseudo-elastic model with modified Ogden-Roxburgh damage function. Requires  $NPTS = 3$ .

**References:**

[Mullins Effect](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

[Mullins Effect Constants \(TB,CDM\)](#) in the *Element Reference*.

[Mullins Effect Material Model](#) in the *Structural Analysis Guide*.

---

**Viscoelasticity Tables****PRONY Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1; Max = 100.

**NPTS:**

Number of pairs of Prony series. Default = 1 pair; Max = 100 pairs.

The total number of data points allowed is 1000.

**TBOPT:**

Defines the relaxation behavior for viscoelasticity.

**1 --**

(or SHEAR) relaxation behavior of the shear response.

**2 --**

(or BULK) relaxation behavior of the volumetric response.



**References:**

Viscoelastic Material Model in the *Element Reference*.

**SHIFT Specifications****NTEMP:**

Allows one temperature for which data will be provided.

**NPTS:**

Number of material constants to be entered as determined by the shift function specified by *TBOPT*.

**3** --

for *TBOPT* = 1 or WLF

**2** --

for *TBOPT* = 2 or TN

$n_f$  --

for *TBOPT* = 3 or FICT, where  $n_f$  is the number of partial fictive temperatures

**TBOPT:**

Defines the shift function

**1 or WLF** --

William-Landel-Ferry shift function

**2 or TN** --

Tool-Narayanaswamy shift function

**3 or FICT** --

Tool-Narayanaswamy with fictive temperature shift function

**100** --

(or USER) User-defined shift function.

**References:**

Viscoelastic Material Model in the *Element Reference*.

**Multiphysics Tables****BH Specifications****NTEMP:**

Not used.

**NPTS:**

Number of data points to be specified. Default = 20, Maximum = 500

**TBOPT:**

Not used.

**References:**

Magnetic Materials in the *Element Reference*.

Additional Guidelines for Defining Regional Material Properties and Real Constants in the *Low-Frequency Electromagnetic Analysis Guide*.

## CNDE Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

High-Frequency Electromagnetic Material Models in the *Element Reference*.

## CNDM Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

High-Frequency Electromagnetic Material Models in the *Element Reference*.

## DPER Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used for HF118, HF119, and HF120.

Permittivity matrix options for PLANE223, SOLID226, and SOLID227:

**0** --

Permittivity matrix at constant strain [ $\epsilon^S$ ] (used as supplied)

**1** --

Permittivity matrix at constant stress [ $\epsilon^T$ ] (converted to [ $\epsilon^S$ ] form before use)

**References:**

Anisotropic Electric Permittivity Material Model (TB,DPER) in the *Element Reference*

## HFFDLD Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

High-Frequency Electromagnetic Materials in the *Element Reference*.

Specifying Material Properties in the *High-Frequency Electromagnetic Analysis Guide*.

**HFLM Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1, Maximum = 20

**NPTS:**

Number of data points to be specified for a given temperature. Default = 1, Maximum = 100

**TBOPT:**

Not used.

**References:**

FLUID116 in the *Element Reference*.

**LSEM Specifications****NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

High-Frequency Electromagnetic Materials in the *Element Reference*.

Specifying Material Properties in the *High-Frequency Electromagnetic Analysis Guide*.

**MUR Specifications****NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Anisotropic relative permeability options:

**0** --

Input permeability matrix.

**1** --

Generate permeability matrix for B-H nonlinear material with uniform dc internal magnetic field

**2** --

Generate permeability matrix for B-H nonlinear material with nonuniform dc internal magnetic field

**References:**

High-Frequency Electromagnetic Material Models in the *Element Reference*.

## PIEZ Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Piezoelectric matrix options.

**0** --

Piezoelectric stress matrix [e] (used as supplied)

**1** --

Piezoelectric strain matrix [d] (converted to [e] form before use)

**References:**

[Piezoelectric Material Model \(TB,DPER\)](#) in the *Element Reference*.

[Piezoelectric Analysis](#) in the *Coupled-Field Analysis Guide*.

## PZRS Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Piezoresistive matrix options

**0** --

Piezoresistive stress matrix (used as supplied)

**1** --

Piezoresistive strain matrix (used as supplied)

**References:**

[Piezoresistive Material Model \(TB,PZRS\)](#) in the *Element Reference*.

[Piezoresistive Analysis](#) in the *Coupled-Field Analysis Guide*.

---

## **Porous Media Tables**

### PM Specifications

**NTEMP:**

The number of temperatures. Default = 1. Maximum must be a value such that  $(NTEMP \times NPTS) \leq 1000$ .

**NPTS:**

The number of material constants. Default = 4. Maximum must be a value such that  $(NTEMP \times NPTS) \leq 1000$ .

**TBOPT:**

Porous media options:

**PERM --**

Permeability

**BIOT --**

Biot coefficient

**References:**

Porous Media Constants (TB,PM) in the *Element Reference*.

Pore-Fluid-Diffusion-Structural Analysis in the *Coupled-Field Analysis Guide*.

Porous Media Flow in the *Theory Reference for the Mechanical APDL and Mechanical Applications*

Also see VM260 in the *Verification Manual*.

---

## ANSYS LS-DYNA Tables

**COMP Specifications****NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

Composite Damage Model in the *ANSYS LS-DYNA User's Guide*.

**DISCRETE Specifications****NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Explicit spring-damper (discrete) material options.

**0 --**

Linear elastic spring (translational or rotational elastic spring). This value is the default.

**1 --**

Linear viscous damper (linear translational or rotational damper)

**2 --**

Elastoplastic spring (elastoplastic translational or rotational spring with isotropic hardening)

**3 --**

Nonlinear elastic spring (nonlinear elastic translational or rotational spring with arbitrary force/displacement response moment/rotation dependency)

- 4 -- Nonlinear viscous damper (nonlinear damping with arbitrary force/velocity response moment/rotational velocity dependency)
- 5 -- General nonlinear spring (general nonlinear translational or rotational spring with arbitrary loading and unloading definitions)
- 6 -- Maxwell viscoelastic spring (Maxwell viscoelastic translational or rotational spring)
- 7 -- Inelastic tension or compression-only spring (inelastic tension or compression only, translational or rotational spring)

**References:**

[Spring-Damper \(Discrete\) Models](#) in the *ANSYS LS-DYNA User's Guide*.

**EOS Specifications*****NTEMP:***

Not used.

***NPIS:***

Not used.

***TBOPT:***

Equation of state (explicit dynamics elements only - no default, must specify).

- 1 -- Johnson-Cook material model - for strain, strain rate, and temperature dependent impact/forming analyses.
- 2 -- Null material model - for allowing equation of state to be considered without computing deviatoric stresses.
- 3 -- Zerilli-Armstrong material model - for metal forming processes in which the stress depends on strain, strain rate, and temperature.
- 4 -- Bamman material model - for metal forming processes with strain rate and temperature dependent plasticity. Does not require an additional equation of state (*EOSOPT* is not used).
- 5 -- Steinberg material model - for modeling high strain rate effects in solid elements with failure.

**References:**

[Equation of State Models](#) in the *ANSYS LS-DYNA User's Guide*.

**EVISC Specifications*****NTEMP:***

Not used.

***NPIS:***

Not used.

**TBOPT:**

Not used.

**References:**

Nonlinear Elastic Models in the *ANSYS LS-DYNA User's Guide*.

## FCON Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1, Maximum = 20.

**NPTS:**

Number of data points to be specified for a given temperature. Default = 1, Maximum = 100.

**TBOPT:**

Not used.

**References:**

FLUID116 in the *Element Reference*.

## FOAM Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Foam material options for explicit dynamics elements (no default - must specify).

**1** --

Rigid, closed cell, low density polyurethane foam material model.

**2** --

Highly compressible urethane foam material model.

**3** --

Energy absorbing foam material model.

**4** --

Crushable foam material model.

**References:**

Foam Models in the *ANSYS LS-DYNA User's Guide*.

## GCAP Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

Pressure Dependent Plasticity Models in the *ANSYS LS-DYNA User's Guide*.

## HONEY Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Not used.

**References:**

Foam Models in the *ANSYS LS-DYNA User's Guide*.

## PLAW Specifications

**NTEMP:**

Not used.

**NPTS:**

Not used.

**TBOPT:**

Plasticity options for explicit dynamics elements (no default - must specify).

**1** --

Isotropic/kinematic hardening model.

**2** --

Strain rate dependent plasticity model used for metal and plastic forming analyses.

**3** --

Anisotropic plasticity model (Barlat and Lian).

**4** --

Strain rate dependent plasticity model used for superplastic forming analyses.

**5** --

Strain rate dependent isotropic plasticity model used for metal and plastic forming analyses.

**6** --

Anisotropic plasticity model (Barlat, Lege, and Brem) used for forming processes.

**7** --

Fully iterative anisotropic plasticity model for explicit shell elements only.

**8** --

Piecewise linear plasticity model for explicit elements only.

**9** --

Elastic-plastic hydrodynamic model for explicit elements only.

**10** --

Transversely anisotropic FLD (flow limit diagram) model for explicit elements only.

**11** --

Modified piecewise linear plasticity model for explicit shell elements only.

**12** --

Elastic viscoplastic thermal model for explicit solid and shell elements only.



**References:**

Nonlinear Inelastic Models in the *ANSYS LS-DYNA User's Guide*.

Pressure Dependent Plasticity Models in the *ANSYS LS-DYNA User's Guide*.

---

## ***Special Material Tables***

**CZM Specifications*****NTEMP:***

Number of temperatures for which data will be provided. Default = 1.

***NPTS:***

Number of data points to be specified for a given temperature.

***TBOPT:***

Cohesive zone material options.

**EXPO --**

Exponential material behavior (valid for interface elements only).

**CBDD --**

Bilinear material behavior with linear softening characterized by maximum traction and maximum separation (valid for contact elements only).

**CBDE --**

Bilinear material behavior with linear softening characterized by maximum traction and critical energy release rate (valid for contact elements only).

**References:**

Interface Delamination and Failure Simulation in the *Structural Analysis Guide*.

Cohesive Zone Material Constants (TB,CZM) in the *Element Reference*.

**EXPE Specifications*****NTEMP:***

Not used.

***NPTS:***

Not used.

***TBOPT:***

Experimental data type:

**UNIAXIAL --**

Uniaxial experimental data.

**BIAXIAL --**

Equibiaxial experimental data.

**SHEAR --**

Pure shear experimental data (also known as planar tension).

**VOLUME --**

Volumetric experimental data.

**SSHEAR --**

Simple shear experimental data.

**UNITENSION --**

Uniaxial tension experimental data.

**UNICOMPRESSION --**

Uniaxial compression experimental data.

**References:**

[Experimental Data \(TB,EXPE\)](#) in the *Element Reference*.

[Experimental Response Functions](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*

See also the **TBFIELD** command documentation for information about defining field-dependent experimental data.

**FCLI Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1. Maximum = 10.

**NPTS:**

Number of data points to be specified for a given temperature. Default = 16 when *TBOPT* = 1. Default = 9 when *TBOPT* = 2.

**TBOPT:**

Material strength limit definition:

**1 --**

Define stress-strength limits.

**2 --**

Define strain-strength limits.

**References:**

[Material Strength Limits \(TB,FCLI\)](#) in the *Element Reference*.

**FLUID Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1; Max = 20.

**NPTS:**

Number of data points to be specified for a given temperature.

**TBOPT:**

Fluid material options:

**LIQUID --**

Define material constants for a liquid material.

**GAS --**

Define material constants for a gas material.

**PVDATA --**

Define pressure-volume data for a fluid material.

**References:**

Fluid Material Models (TB,FLUID) in the *Element Reference*.

Fluid Material Models in the *Theory Reference for the Mechanical APDL and Mechanical Applications*.

**FRIC Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1; Max = 40.

*NTEMP* is not used for the following situations:

- Isotropic or orthotropic friction defined in terms of field data (**TBFIELD** command)
- User-defined friction (*TBOPT* = USER)

**NPTS:**

Number of data points to be specified for user-defined friction (*TBOPT* = USER). Not used for *TBOPT* = ISO or *TBOPT* = ORTHO.

**TBOPT:**

Friction options:

**ISO --**

Isotropic friction (one coefficient of friction, MU). This option is valid for all 2-D and 3-D contact elements. (Default.)

**ORTHO --**

Orthotropic friction (two coefficients of friction, MU1 and MU2). This option is valid for [CONTA173](#), [CONTA174](#), [CONTA175](#), [CONTA176](#), and [CONTA177](#).

**USER --**

User defined friction. This option is valid for [CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#), [CONTA175](#), [CONTA176](#), [CONTA177](#), and [CONTA178](#) elements.

**References:**

[Contact Friction](#) in the *Element Reference*.

See also the **TBFIELD** command for more information on defining a coefficient of friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity.

**GASKET Specifications****NTEMP:**

Number of temperatures for which data will be provided. Default = 1.

**NPTS:**

Number of data points to be specified for a given temperature. Default = 5 for *TBOPT* = PARA. Default = 1 for all other values of *TBOPT*.

**TBOPT:**

Gasket material options.

**PARA --**

Gasket material general parameters.

**COMP --**

Gasket material compression data.

**LUNL --**

Gasket linear unloading data.

**NUNL --**

Gasket nonlinear unloading data.

**TSS --**

Transverse shear data.

**References:**

[Gasket Materials](#) in the *Element Reference*.

[Gasket Joints Simulation](#) in the *Structural Analysis Guide*.

---

## User Tables

### STATE Specifications

**NTEMP:**

Not used.

**NPTS:**

Number of state variables. Maximum = 1000

**TBOPT:**

Not used.

**References:**

[User-Defined Material Constants \(TB,USER\)](#) and [Implicit Creep Equations](#) in the *Element Reference*.

[User Defined Material](#) and [Implicit Creep Procedure](#) in the *Structural Analysis Guide*.

### USER Specifications

**NTEMP:**

Number of temperatures for which data will be provided. Default = 1, Maximum value of *NTEMP* is such that  $NTEMP \times NPTS = 1000$

**NPTS:**

Number of data points to be specified for a given temperature. Default = 48, Maximum value of *NPTS* is such that  $NTEMP \times NPTS = 1000$

**TBOPT:**

Not used.

**References:**

[User-Defined Material Constants \(TB,USER\)](#) in the *Element Reference*.

[User-Defined Material Model](#) in the *Structural Analysis Guide*.

---

## Notes

**TB** activates a data table to be used with subsequent **TBDATA** or **TBPT** commands. The table space is initialized to zero values. Data from this table are used for certain nonlinear material descriptions as well as

for special input for some elements. See [Material Data Tables \(Implicit Analysis\)](#) in the *Element Reference* for a description of table types (*Lab*) or the elements that require the table for special data. See [Material Models](#) in the *ANSYS LS-DYNA User's Guide* for a description of data table input required for explicit dynamic materials. See the **MP** command for linear material property input.

The type of data table specified remains active until the **TB** command is reissued. More than one type of data table may be defined for each material (for example, MISO and CREEP), except that only one type of plasticity/elasticity may be used for each material.

The relationship between SDAMP, DMPRAT, and MP,DAMP is as follows:

SDAMP	DMPRAT	MP,DAMP
s	s/2	s/(2πf)

Where s is the damping value specified in TB,SDAMP and f is the corresponding frequency.

This command is also valid in SOLUTION.

### Product Restrictions

Command Option Lab	Available Products
AHYPER	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
ANAN	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
ANEL	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
ANIS	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
BH	MP ME ST <> <> <> <> <> EM <> <> PP <> EME MFS
BISO	MP ME ST <> PRN <> <> <> <> <> <> DY PP <> EME MFS
BKIN	MP ME ST <> PRN <> <> <> <> <> <> DY PP <> EME MFS
BOYC	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
CAST	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
CHAB	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
CNDE	MP ME ST <> <> <> <> <> <> EH <> PP <> EME MFS
CNDM	MP ME ST <> <> <> <> <> <> EH <> PP <> EME MFS
COMP	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
CONC	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
CREEP	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
CZM	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
DISC	<> <> <> <> <> <> <> <> <> <> DY PP <> <> <>
DP	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
DPER	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
EDP	MP ME ST <> <> <> <> <> <> <> DY PP <> EME MFS
ELASTIC	MP ME ST PR PRN <> <> <> <> <> <> <> DY PP <> EME MFS
EOS	<> <> <> <> <> <> <> <> <> <> DY PP <> <> <>



## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models  
 Main Menu>Preprocessor>Material Props>Material Models  
 Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

### TBCOPY, *Lab*, *MATF*, *MATT*

**Copies a data table from one material to another.**

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

#### *Lab*

Data table label (see the **TB** command for valid labels, and see "[Notes](#)" (p. 1699) for *Lab* = ALL).

#### *MATF*

Material reference number where data table is to be copied from.

#### *MATT*

Material reference number where data table is to be copied to.

## Notes

The **TBCOPY** command, with *Lab* = ALL, copies all of the nonlinear data defined by the **TB** command. If you copy a model that includes both yield behavior constants and linear constants (for example, a BKIN model), **TBCOPY,ALL** and **MPCOPY** are used together to copy the entire model. All input data associated with the model is copied, that is, all data defined through the **TB** and **MP** commands.

Also, if you copy a material model using the [Material Model Interface \(Edit> Copy\)](#), both the commands **TBCOPY,ALL** and **MPCOPY** are issued, regardless of whether the model includes linear constants only, or if it includes a combination of linear and yield behavior constants.

This command is also valid in SOLUTION.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models  
 Main Menu>Preprocessor>Material Props>Material Models  
 Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

### TBDATA, *STLOC*, *C1*, *C2*, *C3*, *C4*, *C5*, *C6*

**Defines data for the material data table.**

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

#### *STLOC*

Starting location in table for entering data. For example, if *STLOC* = 1, data input in the *C1* field applies to the first table constant, *C2* applies to the second table constant, etc. If *STLOC*=5, data input in the

*C1* field applies to the fifth table constant, etc. Defaults to the last location filled + 1. The last location is reset to 1 with each **TB** or **TBTEMP** command.

*C1, C2, C3, . . . , C6*

Data values assigned to six locations starting with *STLOC*. If a value is already in this location, it is re-defined. A blank value leaves the existing value unchanged.

## Notes

Defines data for the table specified on the last **TB** command at the temperature specified on the last **TBTEMP** command (if applicable). The type of data table specified in the last **TB** command determines the number of data values needed in **TBDATA**. Data values are linearly interpolated for temperatures that fall between user defined **TBTEMP** values. See [Material Data Tables \(Implicit Analysis\)](#) of the *Element Reference* for the number of data values required for different material behavior options.

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models**

**Main Menu>Preprocessor>Material Props>Material Models**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models**

---

**TBDELE**, *Lab, MAT1, MAT2, INC*

**Deletes previously defined material data tables.**

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

*Lab*

Data table label. (See the **TB** command for valid labels.) If ALL, delete all data tables.

*MAT1, MAT2, INC*

Delete tables for materials *MAT1* to (*MAT2* defaults to *MAT1*) in steps of *INC* (defaults to 1). If *MAT1*=ALL, ignore *MAT2* and *INC* and delete data tables for all materials.

## Notes

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models**

**Main Menu>Preprocessor>Material Props>Material Models**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models**

---



**TBEO**, *Par*, *Value*

Sets special options or parameters for material data tables.

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

**Par**

Parameter name:

**CAPCREEPREG**

Available for the viscoplasticity/creep model (**TB,CREEP**), allows two creep models to be specified via the same material ID when used with the Extended Drucker-Prager model (**TB,EDP**).

**Value**

Parameter value:

**SHEA**

Use the shear stress-state creep model with the Extended Drucker-Prager model. Valid only when *Par* = CAPCREEPREG.

**COMP**

Use the compaction stress-state creep model with the Extended Drucker-Prager model. Valid only when *Par* = CAPCREEPREG.

**Notes**

Issue the **TBEO** command *after* activating the data table (**TB**) but *before* defining data for the table (**TBDATA**) or a point on a nonlinear data curve (**TBPT**).

**Menu Paths**

This command cannot be accessed from a menu.

**TBFIELD**, *Type*, *Value*

Defines values of field variables for the material data tables.

[PREP7: Data Tables](#)

MP ME ST <> PRN <> <> <> <> DY PP <> EME MFS

**Type**

Type of field variable:

**FREQ**

A frequency will be specified in *Value*

**TEMP**

A temperature will be specified in *Value*

**TIME**

A time will be specified in *Value*

**NPRES**

A normal pressure will be specified in *Value*

**SLDA**

A total sliding distance (algebraic) will be specified in *Value*

**SLDI**

A total sliding distance (absolute) will be specified in *Value*

**SLRV**

A sliding velocity will be specified in *Value*

**Value**

The field value to be referenced (use this command multiple times to enter values of different field variables).

**Notes**

You use this command with **TB,ELASTIC**, or **TB,SDAMP** to define multiple field values for frequency and/or temperature dependent material data tables. You can also use this command with **TB,FRIC** to define friction that is dependent on temperature, time, normal pressure, sliding distance, and sliding velocity.

**Note**

When defining friction, keep in mind the following points:

- The TEMP value specified on this command corresponds to the average temperature on the contact surface for contact elements [CONTA171](#), [CONTA172](#), [CONTA173](#), [CONTA174](#), [CONTA175](#), [CONTA176](#), and [CONTA177](#). For contact element [CONTA178](#), the TEMP value corresponds to the average temperature of the nodes.
- The TIME value specified on this command corresponds to the analysis time specified on the **TIME** command.
- The algebraic sliding distance (SLDA) specified on this command is the total sliding distance (the algebraic sum) as reported in the element output definitions table for the contact elements (for example, TASS and TASR output items for [CONTA174](#)).
- The absolute sliding distance (SLDI) specified on this command is the total accumulated sliding distance (the absolute sum) as reported in the element output definitions table for the contact elements (for example, AASS and AASR output items for [CONTA174](#)).
- When used with **TB,FRIC**, field variables defined by **TBFIELD** are only available for isotropic friction ( $TBOPT = ISO$ ) and orthotropic friction ( $TBOPT = ORTHO$ ); they are not available for user-defined friction ( $TBOPT = USER$ ).

Enter the appropriate **TB,ELASTIC**, **TB,SDAMP**, or **TB,FRIC** option first to define your data tables as field variable-dependent. Then issue the **TBFIELD** command to define your field values. Data values must be defined in ascending order for all field quantities. If a field value is to be held constant, it should be defined only once; subsequent definitions will be ignored. There is no limit on the number of values you can specify. The specified field value remains active until the next **TBFIELD** command is input. Once you define the field value(s), use **TBDATA** to define your data for the data tables.

See [Understanding How ANSYS Interpolates Field Variables](#) in the *Element Reference* for more information on the interpolation scheme used for field-dependent material properties.

See [Full Harmonic Response Analysis](#) in the *Structural Analysis Guide* for more information on using **TBFIELD** with **TB,ELASTIC** or **TB,SDAMP**.

See [Contact Friction](#) in the *Element Reference* for more information on using **TBFIELD** with **TB,FRIC**.

## Menu Paths

This command cannot be accessed from a menu.

**TBFT**, *Oper*, *ID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*

Performs material curve-fitting operations.

[PREP7:Materials](#)

MP ME ST <> PRN <> <> <> <> DY PP <> EME MFS

Use material curve fitting to evaluate your experimental data and correlate it to the nonlinear material models built into ANSYS. You apply coefficients to your curve data, determine a fit with existing model, and write the data according to the table configurations outlined in the **TB** command. Curve fitting is available for nonlinear models associated with the material behaviors listed below:

For hyperelastic material behavior, see "[TBFT Specifications for Hyperelastic Models](#)" (p. 1703), below.

For creep material behavior, see "[TBFT Specifications for Creep Models](#)" (p. 1706), below.

For viscoelastic material behavior, see "[TBFT Specifications for Viscoelastic Models](#)" (p. 1708), below.

## TBFT Specifications for Hyperelastic Models

**TBFT**, *Oper*, *ID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*

### *Oper*

The specific curve-fitting operation:

#### **FADD**

Define a constitutive model.

#### **FDEL**

Delete a constitutive model.

#### **FSET**

Write data related to a constitutive model to the database (same as **TB** command).

#### **SET**

Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

#### **CDEL**

Deletes coefficients at current reference temperature. Applicable only for temperature dependent coefficients.

#### **SOLVE**

Solve for coefficients.

#### **FIX**

Fix (hold constant) the coefficient you specify in *Option4*.

#### **EADD**

Add experimental data.

**EDEL**

Delete experimental data.

**LIST**

List all data associated with the material model represented by the material *ID* number.

**ID**

The material reference number (same as *MAT* argument used in the **TB** command). Valid entry is any number greater than zero (default = 1) but less than 100,000.

**Option1**

For curve fit function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE or FIX) this field specifies the category (HYPER).

For adding or deleting your experiment (*Oper* = EADD or EDEL), this field specifies the experimental data type. Valid entries are: UNIA, BIAX, SHEA, SSHE, and VOLU (default = UNIA).

**Option2**

For curve fit function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE, or FIX), this field specifies constitutive model type. The valid entries are listed in [Table 255: Hyperelastic Options \(p. 1705\)](#) below.

When you need to specify a filename from which to get experimental data (*Oper* = EADD), place that string here. Valid entry is any filename string. You can enter the entire `path\filename.extension` string and leave the next two fields (*Option3* and *Option4*) blank, or you can specify the name here, the extension in the next field, and the path following.

**Option3**

For *Oper* = FADD, FDEL, FSET, CDEL, SET, SOLVE or FIX, some of the cases specified in *Option2* will require that the polynomial order be specified. The applicable values for the order specification are listed in [Table 255: Hyperelastic Options \(p. 1705\)](#).

If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the file extension.

**Option4**

When you are working on a specific coefficient (*Oper* = FIX), this field specifies the index of that coefficient. Valid entries vary from 1 to *n*, where *n* is the total number of coefficients (default = 1).

For *Oper* = SET, see [Table 254: Set Operations \(p. 1704\)](#), below.

If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the directory/path specification.

If *Oper* = SOLVE, this value specifies the curve-fitting procedure. Valid entries are 0 for unnormalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

**Option5**

When you are working on a specific coefficient (*Oper* = FIX), this field specifies the index of that coefficient. Valid entries vary from 1 to *N*, where *N* is the total number of coefficients (default = 1)

For *Oper* = SET, see [Table 254: Set Operations \(p. 1704\)](#), below.

**Table 254 Set Operations**

Purpose	Option4	Option5
Set the value of the coefficient.	Index of coefficient	Value of that coefficient

Purpose	Option4	Option5
Turn temperature dependency ON/OFF*	tdep	1 for ON and 0 for OFF
Set reference temperature	tref	Temperature value

\* - Temperature data should be supplied in the same units as the TREF command.

If *Oper* = SOLVE, use this field to specify the number of iterations to be used in the calculation of the coefficients. Valid entry is any positive integer (default = 1000)

#### Option6

If *Oper* = SOLVE, specifies the allowed tolerance in residual change to stop an iteration. Valid entry is 0.0 to 1.0 (default = 0.0).

#### Option7

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid entry is 0 to 1 (default = 0).

**Table 255 Hyperelastic Options**

Option1	Option2	Option3
HYPER	MOON	2, 3, 5, 9
HYPER	POLY	1 to N
HYPER	OGDEN	1 to N
HYPER	YEOH	1 to N
HYPER	BOYC	NA
HYPER	GENT	NA
HYPER	NEO	NA
HYPER	BLAT	NA
HYPER	FOAM	1 to N

Following table summarizes the format for hyperelastic operations via the **TBFT** command:

### **Hyperelastic Model Command Summary**

Oper	ID	Option1	Option2	Option3	Option4	Option5	Option6	Option7
FADD	ID	HYPER	Option	Order				
FDEL	ID	HYPER	Option	Order				
FSET	ID	HYPER	Option	Order				
SET	ID	HYPER	Option	Order	Index	Value		
SOLVE	ID	HYPER	Option	Order	Norm Flag	Num Iter	RTOL	CTOL
FIX	ID	HYPER	Option	Order	Index	Fix / Unfix Flag		
EADD	ID	ExpType	File	Extension	Directory			

Oper	ID	Option1	Option2	Option3	Option4	Option5	Option6	Option7
EDEL	ID	Exp Type / Index						

## TBFT Specifications for Creep Models

**TBFT**, *Oper*, *ID*, *Option1*, *Option2*, *Option3*, ..., *Option7*

### *Oper*

The specific curve-fitting operation:

#### **FADD**

Define a constitutive model.

#### **FDEL**

Delete a constitutive model.

#### **FSET**

Write data related to a constitutive model to the database (same as **TB** command).

#### **SET**

Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

#### **CDEL**

Deletes coefficients at current reference temperature. Applicable only for temperature dependent coefficients.

#### **SOLVE**

Solve for coefficients.

#### **FIX**

Fix (hold constant) the coefficient you specify in *Option4*.

#### **EADD**

Add experimental data.

#### **EDEL**

Delete experimental data.

#### **LIST**

List all data associated with the material model represented by the material *ID* number.

### *ID*

The material reference number (same as *MAT* argument used in the **TB** command). Valid entry is any number greater than zero (default = 1) but less than 100,000.

### *Option1*

For curve fit function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE or FIX) this field specifies the category (CREEP).

For adding or deleting your experiment (*Oper* = EADD or EDEL), this field specifies the experimental data type (CREEP).

### *Option2*

For curve fit function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE, or FIX), this field specifies constitutive model type. The valid entries are listed in [Table 257: Creep Options \(p. 1708\)](#) below.

When you need to specify a filename from which to get experimental data (*Oper* = EADD), place that string here. Valid entry is any filename string. You can enter the entire `path\filename.extension`

string and leave the next two fields (*Option3* and *Option4*) blank, or you can specify the name here, the extension in the next field, and the path following.

#### **Option3**

If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the file extension.

#### **Option4**

When you are working on a specific coefficient (*Oper* = FIX), this field , specifies the index of that coefficient. Valid entries vary from 1 to *n*, where *n* is the total number of coefficients (default = 1).

For *Oper* = SET, see [Table 256: Set Operations \(p. 1707\)](#), below.

If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the directory/path specification.

If *Oper* = SOLVE, this value specifies the curve-fitting procedure. Valid entries are 0 for unnormalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

#### **Option5**

If *Oper* = SOLVE, use this field to specify the number of iterations to be used in the calculation of the coefficients. Valid entry is any positive integer (default = 1000)

If you specify a coefficient to be held constant (*Oper* = FIX), enter a 1 to FIX the coefficient, and a 0 to allow it to vary (turn fixing OFF).

For *Oper* = SET, see [Table 256: Set Operations \(p. 1707\)](#), below.

**Table 256 Set Operations**

<b>Purpose</b>	<b>Option4</b>	<b>Option5</b>
Set the value of the coefficient	Index of coefficient	Value of coefficient
Turn temperature dependency ON/OFF*	tdep	1 for ON and 0 for OFF
Set reference temperature	tref	Temperature value

\* - When tdep is OFF, the Arrhenius term (e.g. C4) in the strain hardening creep equation is calculated. Because TBTEMP is not included in the curve-fitting regimen, the experimental data (in absolute temperature) must be supplied using the /TEMP command.

\* - When tdep is set ON, the Arrhenius term is set to 0 and the constants are separately computed for each temperature. The curve-fitting regimen uses **TBTEMP**. In this case, you must supply the experimental data using the same units as the **TREF** command.

#### **Option6**

If *Oper* = SOLVE, specifies the allowed tolerance in residual change to stop an iteration. Valid entry is 0.0 to 1.0 (default = 0.0).

**Option7**

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid entry is 0 to 1 (default = 0).

**Table 257 Creep Options**

Category	Name	Option
CREEP	SHAR	NA
CREEP	THAR	NA
CREEP	GEXP	NA
CREEP	GGRA	NA
CREEP	GBLA	NA
CREEP	MTHA	NA
CREEP	MSHA	NA
CREEP	GGAR	NA
CREEP	EXPO	NA
CREEP	NORT	NA
CREEP	PSTH	NA
CREEP	PSRP	NA
CREEP	GTHA	NA

Following table summarizes the format for creep operations via the **TBFT** command:

**Creep Model Command Summary**

Oper	ID	Option1	Option2	Option3	Option4	Option5	Option6	Option7
FADD	ID	CREEP	Option	NA				
FDEL	ID	CREEP	Option	NA				
FSET	ID	CREEP	Option	NA				
SET	ID	CREEP	Option	NA	Index	Value		
SOLVE	ID	CREEP	Option	NA	Norm Flag	Num Iter	RTOL	CTOL
FIX	ID	CREEP	Option	NA	Index	Fix / Unfix Flag		
EADD	ID	ExpType	File	Extension	Directory			
EDEL	ID	ExpType / Index						

**TBFT Specifications for Viscoelastic Models**

**TBFT**, *Oper*, *ID*, *Option1*, *Option2*, *Option3*, ..., *Option7*

**Oper**

The specific curve-fitting operation:



**FCASE**

Define a case/constitutive model for viscoelasticity (prony).

**FADD**

Define a constitutive model.

**FDEL**

Delete a constitutive model.

**FSET**

Write data related to a constitutive model to the database (same as **TB** command).

**SET**

Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

**CDEL**

Deletes coefficients at current reference temperature. Applicable only for temperature dependent coefficients.

**SOLVE**

Solve for coefficients.

**FIX**

Fix (hold constant) the coefficient you specify in *Option4*.

**EADD**

Add experimental data.

**EDEL**

Delete experimental data.

**LIST**

List all data associated with the material model represented by the material *ID* number.

**ID**

The material reference number (same as *MAT* argument used in the **TB** command). Valid entry is any number greater than zero (default = 1) but less than 100,000.

**Option1**

This option is set to VISCO for adding coefficients (*Oper* = FADD). The FADD operation is used to specify the shear order, the bulk order or the shift option.

This option (*Oper* = CASE) is set to NEW or FINI. The FADD commands are always enclosed by **TBFT,FCASE, ID, NEW** and **TBFT,FCASE, ID, FINI**. See "[Material Curve Fitting](#)" in the *Structural Analysis Guide* for more information.

For other curve fit function operations (*Oper* = FDEL, FSET, SET, CDEL, SOLVE or FIX) this field is set to CASE.

For adding or deleting your experiment (*Oper* = EADD or EDEL), this field specifies the experiment type. Valid entries are: SDEC (Shear Modulus vs. Time) or BDEC (Bulk Modulus vs. Time).

**Option2**

For defining your viscoelastic case (*Oper* = FCASE), you specify PVHE.

For curve fit function operations (*Oper* = FDEL, FSET, SET, CDEL, SOLVE, or FIX), this field specifies the case name.

To create a new case ( $Oper = FADD$ ), the valid options are listed in [Table 259: Viscoelastic Options \(p. 1711\)](#) below.

When you need to specify a filename from which to get experimental data ( $Oper = EADD$ ), place that string here. Valid entry is any filename string. You can enter the entire `path\filename.extension` string and leave the next two fields ( $Option3$  and  $Option4$ ) blank, or you can specify the name here, the extension in the next field, and the path following.

#### **Option3**

For  $Oper = FCASE$ , this field specifies the CASE name.

For  $Oper = FADD$ , this field specifies the order

If a filename for experimental data is being specified in  $Option2$  ( $Oper = EADD$ ), this field will contain the file extension.

#### **Option4**

When you are working on a specific coefficient ( $Oper = FIX$ ), this field specifies the index of that coefficient. Valid entries vary from 1 to  $n$ , where  $n$  is the total number of coefficients (default = 1).

For  $Oper = SET$ , see [Table 258: Set Operations \(p. 1710\)](#), below.

You can also specify TREF to indicate the reference temperature, or COMP for a partial/complete solution (only for bulk, only for shear, or all coefficients).

If a filename for experimental data is being specified in  $Option2$  ( $Oper = EADD$ ), this field will contain the directory/path specification.

If  $Oper = SOLVE$ , this value specifies the curve-fitting procedure. Valid entries are 0 for unnormalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

#### **Option5**

For  $Oper = SET$ , see [Table 258: Set Operations \(p. 1710\)](#), below.

**Table 258 Set Operations**

Purpose	Option4	Option5
Set the value of the coefficient	Index of coefficient	Value of coefficient
Turn temperature dependency ON/OFF	tdep	1 for ON and 0 for OFF
Set reference temperature	tref	Temperature value
Set current component to solve for	comp	pshea, pbulk, or pvhe

If  $Oper = SOLVE$ , use this field to specify the number of iterations to be used in the calculation of the coefficients. Valid entry is any positive integer (default = 1000).

If you are specifying a coefficient to be held constant ( $Oper = FIX$ ), a 1 FIXES the specified coefficient, and a 0 allows it to vary (turns fixing OFF).

#### **Option6**

If  $Oper = SOLVE$ , specifies the allowed tolerance in residual change to stop an iteration. Valid entry is 0.0 to 1.0 (default = 0.0).

**Option7**

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid entry is 0 to 1 (default = 0).

**Table 259 Viscoelastic Options**

Category	Name	Option
VISCO	PSHEAR	1 to N
VISCO	PBULK	1 to N
VISCO	SHIFT	TN, WLF, NONE

Following table summarizes the format for viscoelastic operations via the **TBFT** command:

**Viscoelastic Models**

Oper	ID	Option1	Option2	Option3	Option4	Option5	Option6	Option7
FCASE	ID	NEW	PVHE	Case Name				
FCASE	ID	FINI						
FADD	ID	VISCO	Option	Order				
FDEL	ID	CASE	Case Name	NA				
FSET	ID	CASE	Case Name	NA				
SET	ID	CASE	Case Name	NA	Index	Value		
SOLVE	ID	CASE	Case Name	NA	Norm Flag	Num Iter	RTOL	CTOL
FIX	ID	CASE	Case Name	NA	Index	Fix / Unfix Flag		
EADD	ID	ExpType	File	Extension	Directory			
EDEL	ID	ExpType / Index						

**Notes**

This command provides tools for comparing experimental material data to the program-provided calculated data for various nonlinear material options. Based on curve-fitting comparisons and error norms, you choose the model to use during the solution phase of the analysis according to the best fit. All of the capabilities of the TBFT-series of commands are accessible interactively via the standard material GUI. See "[Material Curve Fitting](#)" in the *Structural Analysis Guide* for more information.

You can display material model data associated with both the **TB** command and the **TBFT,FSET** command by issuing **TBLIST,ALL,ALL**.

Material model data associated with the *latest* **TB** or **TBFT,FSET** command overwrites previous data.

You can display material model data associated with both the **TB** command and the **TBFT,FSET** command by issuing **TBLIST,ALL,ALL**.

The capability to fix coefficients (*Option4* = FIX) applies only to nonlinear curve fits (as listed in Table 10.3: "Hyperelastic Curve-Fitting Model Types").

The uniaxial, biaxial, and shear experimental data use *engineering* stress. The volumetric data uses *true* stress. See the *Element Reference* for details on experimental data for creep and viscoelasticity.

## Menu Paths

**Main Menu> Preprocessor> Material Props> Material Models**

---

## TBLE

Specifies "Data table properties" as the subsequent status topic.

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Notes

This is a status (**STAT**) topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

**Utility Menu>List>Status>Preprocessor>Data Tables**

---

## TBLIST, Lab, MAT

Lists the material data tables.

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

### *Lab*

Data table label. (See the **TB** command for valid labels.) Defaults to the active table. If ALL, list data for all labels.

### *MAT*

Material number to be listed (defaults to the active material). If ALL, list data tables for all materials.

## Notes

This command is a utility command, valid anywhere.

## Menu Paths

Utility Menu>List>Properties>Data Tables

---

**TBMODIF**, *ROW*, *COL*, *VALUE*

**Modifies data for the material data table (GUI).**

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

*ROW*, *COL*

The row and column numbers of the table entry to be modified.

*VALUE*

The new value to be used in the *ROW*, *COL* location.

## Notes

The **TBMODIF** command modifies data for the table specified on the last **TB** command.

For temperature-dependent data, the temperature specified on the last **TBTEMP** command is used.

**TBMODIF** is a command generated by the Graphical User Interface (GUI). It appears in the log file (Job-name.LOG) if a **TB** material data table is graphically edited in spreadsheet fashion.

The **TBMODIF** command is *not* intended to be typed in directly during an analysis session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is also valid in SOLUTION.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

**TBPLOT**, *Lab*, *MAT*, *TBOPT*, *TEMP*, *SEGN*

**Displays the material data table.**

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM EH DY PP <> EME MFS

*Lab*

Data table label. Valid labels are: MKIN, KINH, MELAS, MISO, BKIN, BISO, BH, GASKET, and JOIN. Defaults to the active table label. For B-H data, also valid are: NB to display NU-B<sup>2</sup>, MH to display MU vs. H, and SBH, SNB, SMH to display the slopes of the corresponding data.

*MAT*

Material number to be displayed (defaults to the active material).

**TBOPT**

Gasket material or joint element material option to be plotted.

The following gasket material options are valid only when *Lab* = GASKET:

**ALL**

Plots all gasket data.

**COMP**

Plots gasket compression data only.

**LUNL**

Plots gasket linear unloading data with compression curve.

**NUNL**

Plots gasket nonlinear unloading data only.

The following joint element material options are valid only when *Lab* = JOIN:

**JNSA**

Plots nonlinear stiffness data that is applicable to all relevant directions.

**JNSn**

Plots only the specified nonlinear stiffness data. The "n" can be 1, 4, or 6. For example, JNS4 plots only the nonlinear stiffness data specified in the local direction 4 (ROTX).

**JNDA**

Plots nonlinear damping data that is applicable to all relevant directions.

**JNDn**

Plots only the specified nonlinear damping data. The "n" can be 1, 4, or 6. For example, JND4 plots only the nonlinear damping data specified in the local direction 4 (ROTX).

**JNFA**

Plots nonlinear hysteretic friction data that is applicable to all relevant directions.

**JNFn**

Plots only the specified nonlinear hysteretic friction data. The "n" can be 1, 4, or 6. For example, JNF4 plots only the nonlinear hysteretic friction data specified in local direction 4 (ROTX).

**TEMP**

Specific temperature at which gasket data or joint element material data will be plotted (used only when *Lab* = GASKET or JOIN). Use *TEMP* = ALL to plot gasket data or joint element material data at all temperatures.

**SEGN**

Segment number of plotted curve (valid only when *Lab* = GASKET):

**NO**

Segment number is not added to plotted curve (default).

**YES**

Segment number is added to plotted curve. This option is ignored if the number of data points in a curve exceeds 20.

## Notes

Only data for stress-strain, B-H, gasket curves, or joint element nonlinear material model curves can be displayed.

The *TBOPT* and *TEMP* values are valid only when *Lab* = GASKET or JOIN.

The *SEGN* value is valid only when *Lab* = GASKET.

This command is valid in any processor.

## Menu Paths

Utility Menu>Plot>Data Tables

**TBPT**, *Oper*, *X1*, *X2*, *X3*, ..., *XN*

**Defines a point on a nonlinear data curve.**

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

### *Oper*

Operation to perform:

#### **DEFI**

Defines a new data point (default). The point is inserted into the table in ascending order of *X1*. If a point already exists with the same *X1* value, it is replaced.

#### **DELE**

Deletes an existing point. The *X1* value must match the *X1* value of the point to be deleted (*XN* is ignored).

*X1*, *X2*, ..., *XN*

The *N* components of the point. *N* depends on the type of data table. Except for **TB,EXPE** all other **TB** Tables support only 2 components.

## Notes

**TBPT** defines a point on a nonlinear data curve (such as a stress-strain curve, B-H curve, etc.) at the temperature specified on the last **TBTEMP** command. The meaning of the values depends on the type of data table specified on the last **TB** command (MISO, BH, etc.).

This command is also valid in SOLUTION.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

**TBTEMP**, *TEMP*, *KMOD*

**Defines a temperature for a material data table.**

[PREP7: Data Tables](#)

MP ME ST PR PRN <> <> <> EM <> DY PP <> EME MFS

**TEMP**

Temperature value (defaults to 0.0 if *KMOD* is blank).

**KMOD**

If blank, *TEMP* defines a new temperature. If an integer, 1 to NTEMP (from the **TB** command), modify that previously defined temperature to the *TEMP* value, unless *TEMP* is blank, then that previously defined temperature is reactivated. Use **TBLIST** to list temperatures and data. The next **TBDATA** or **TBPT** commands also add or change the data at this temperature. If *KMOD* = CRIT (and *TEMP* is blank), the next **TBDATA** values are failure-criteria keys. If *KMOD* = STRAIN (and *TEMP* is blank), the next **TBDATA** values are strains as described for the MKIN property option (see [Material Data Tables \(Implicit Analysis\)](#) of the [Element Reference](#)).

## Notes

The **TBTEMP** command defines a temperature to be associated with the data on subsequent **TBPT** or **TBDATA** commands.

The defined temperature remains active until the next **TBTEMP** command is issued.

Data values must be defined with the temperatures in ascending order. Temperatures previously associated with a data table may also be modified.

This command is also valid in SOLUTION.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models**

**Main Menu>Preprocessor>Material Props>Material Models**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models**

---

**TCHG**, *ENAME1*, *ENAME2*, *ETYPE2*

**Converts 20-node degenerate tetrahedral elements to their 10-node non-degenerate counterparts.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**ENAME1**

Name (or the number) of the 20-node tetrahedron element that you want to convert. This argument is required.

**ENAME2**

Name (or the number) of the 10-node tetrahedron element to which you want to convert the *ENAME1* elements. This argument is required.



**ETYPE2**

Element TYPE reference number for *ENAME2*. If *ETYPE2* is 0 or is not specified, ANSYS chooses the element TYPE reference number for *ENAME2*. See the "Notes" (p. 1717) section for details. This argument is optional.

**Notes**

The **TCHG** command allows you to specify conversion of *any* selected 20-node brick that is degenerated into a tetrahedron to a 10-node tetrahedron.

The **TCHG** command is useful when used in with the **MOPT, PYRA** command. Twenty-node pyramid shaped elements may be used in the same volume with 10-node tetrahedra.

Performing a conversion is likely to create circumstances in which more than one element type is defined for a single volume.

If specified, *ETYPE2* will usually be the same as the local element TYPE number (**ET**, *ITYPE*) that was assigned to *ENAME2* with the **ET** command. You can specify a unique number for *ETYPE2* if you prefer. Although *ETYPE2* is optional, it may be useful when two or more *ITYPES* have been assigned to the same element (for example, if two **SOLID187** elements have been established in the element attribute tables for the current model, use the *ETYPE2* argument to distinguish between them). If *ETYPE2* is nonzero and it has *not* already been assigned to an element via **ET**, ANSYS assigns the *ETYPE2* value to *ENAME2* as its element TYPE reference number.

If *ETYPE2* is 0 or is not specified, ANSYS determines the element TYPE reference number for *ENAME2* in one of these ways:

- If *ETYPE2* is 0 or is not specified, and *ENAME2* *does not appear* in the element attribute tables, ANSYS uses the next available (unused) location in the element attribute tables to determine the element TYPE reference number for *ENAME2*.
- If *ETYPE2* is 0 or is not specified, and *ENAME2* *appears* in the element attribute tables, ANSYS uses *ENAME2* 's existing element TYPE reference number for *ETYPE2* . (If there is more than one occurrence of *ENAME2* in the element attribute tables (each with its own TYPE reference number), ANSYS uses the first *ENAME2* reference number for *ETYPE2* .)

See [Meshing Your Solid Model](#) of the *Modeling and Meshing Guide* for detailed information about converting degenerate tetrahedral elements.

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Modify Mesh>Change Tets**

---

**/TEE**, *Label*, *Fname*, *Ext*, --

**Writes a list of commands to a specified file at the same time that the commands are being executed.**

APDL: Macro Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Label*

Indicates how ANSYS is to interpret this **/TEE** command:

#### **NEW --**

Signals the beginning of the command text that is to be written to *Fname*. If *Fname* already exists, specifying NEW causes the contents of *Fname* to be overwritten.

#### **APPEND --**

Indicates that you want to append to *Fname* the command text that follows.

#### **END --**

Signals the end of the command text that is to be written to or appended to *Fname*.

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

### *Ext*

Filename extension (8 character maximum).

If you plan to execute the file as if it were an ANSYS command, use the extension *.mac*.

--

Unused field.

## Notes

You can use the **/TEE** command to record a macro to a specified file at the same time that the macro is being executed. It is similar to the UNIX **tee** command.

For more information about the **/TEE** command, see the "Introducing APDL" of the *ANSYS Parametric Design Language Guide*.

The following example illustrates the use of the **/TEE** command. If you issue these commands:

```
/tee,new,myfile,mac
et,1,42,0,0,1
ex,1,3e7
/tee,end
/tee,append,myfile,mac
n,1,8
n,5,11
fill
ngen,5,5,1,5,1,0,1
/tee,end
```

the content of **myfile.mac** is:

```
et,1,42,0,0,1
ex,1,3e7
```

```
n,1,8
n,5,11
fill
ngen,5,5,1,5,1,0,1
```

This command is valid in any processor, but only during an interactive run.

## Menu Paths

**This command cannot be accessed from a menu.**

**TERM**, *Kywr*d, *Opt1*, *Opt2*, *Opt3*

**Specifies various terminal driver options.**

[DISPLAY: Driver Options](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

If *Kywr*d = **COPY**, command format is **TERM,COPY,NCOPY**

**NCOPY**

Activate hard copy device for *NCOPY* (0,1,2, etc.) copies.

If *Kywr*d = **LOOP**, command format is **TERM,LOOP,NLOOP,PAUSE**. Used only with **PLOT,ALL**

**NLOOP**

Loop *NLOOP* times back to beginning of file when end of file is reached.

**PAUSE**

Pause *PAUSE* seconds between plots.

If *Kywr*d = **NOPROM**, command format is **TERM,NOPROM,KEY**

**KEY**

Prompt key:

**0**

Display prompt line for prompt.

**1**

Use terminal bell for prompt.

## Notes

Used only with terminal driver names on [/SHOWDISP](#) command.

This command is also valid in PREP7.

## Menu Paths

**This command cannot be accessed from a menu.**

**THOPT**, *Refopt*, *REFORMTOL*, *NTABPOINTS*, *TEMPMIN*, *TEMPMAX***Nonlinear transient thermal solution option.**

SOLUTION: Analysis Options

MP ME &lt;&gt; PR &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

***Refopt***

Matrix reform option.

**FULL**

Use the full Newton-Raphson solution option (default).

**QUASI**Use a selective reform solution option based on *REFORMTOL*.**LINEAR**

Use a linear solution option with no matrix reform.

***REFORMTOL***

Property change tolerance for Matrix Reformation (.05 default). The thermal matrices are reformed if the maximum material property change in an element (from the previous reform time) is greater than the reform tolerance.

***NTABPOINTS***

Number of points in Fast Material Table (64 default).

***TEMPMIN***Minimum temperature for Fast Material Table. Defaults to the minimum temperature defined by the **MPTEMP** command for any material property defined.***TEMPMAX***Maximum temperature for Fast Material Table. Defaults to the maximum temperature defined by the **MPTEMP** command for any material property defined.**Notes**

This solution logic is only supported by the ICCG and the JCG solvers (**EQSLV** command). You must issue the **THOPT** command before the **EQSLV** command to use the QUASI option.

For *Refopt* = QUASI, results from a restart may be different than results from a single run because the stiffness matrices are always recreated in a restart run, but may or may not be in a single run (depending on the behavior resulting from the *REFORMTOL* setting). Additionally, results may differ between two single runs as well, if the matrices are reformed as a result of the *REFORMTOL* setting.

For *Refopt* = QUASI, midside node temperatures are not calculated if 20-node thermal solid elements (**SOLID90** or **SOLID279**) are used.

Because the matrices are reformed based on material properties, only *Refopt* = FULL should be used for the radiosity solver method.

**THOPT** settings take precedence over **SOLCONTROL** settings.

**ANTYPE**,,RESTART (including multiframe restart) is not supported by **THOPT**, LINEAR option.

**Distributed ANSYS Restriction** The *Refopt* = QUASI and *Refopt* = LINEAR options are not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options  
Main Menu>Solution>Analysis Type>Analysis Options

**TIFF**, *Kywrđ*, *OPT*

Provides TIFF file Export for ANSYS Displays.

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Kywrđ*

Specifies various TIFF file export options.

### COMP

If *Kywrđ* = COMP, then *OPT* controls data compression for the output file. If COMP = 0, then compression is off. If COMP = 1 (default), then compression is on.

### ORIENT

If *Kywrđ* = ORIENT, then *OPT* will determine the orientation of the entire plot. *OPT* can be either Horizontal (default) or Vertical.

### COLOR

If *Kywrđ* = COLOR, then *OPT* will determine the color attribute of the saved file. *OPT* can be 0, 1, or 2, corresponding to Black and White, Grayscale, and Color (default), respectively.

### TMOD

If *Kywrđ* = TMOD, then *OPT* will determine the text method. *OPT* can be either 1 or 0, corresponding to bitmap text (default) or line stroke text, respectively.

### DEFAULT

If *Kywrđ* = DEFAULT, then all of the default values, for all of the *Kywrđ* parameters listed above, are active.

*OPT*

*OPT* can have the following names or values, depending on the value for *Kywrđ* (see above).

### 1 or 0

If *Kywrđ* = COMP, a value of 1 (on) or 0 (off) will control compression for the TIFF file.

### Horizontal, Vertical

If *Kywrđ* = ORIENT, the terms Horizontal or Vertical determine the orientation of the plot.

### 0, 1, 2

If *Kywrđ* = COLOR, the numbers 0, 1, and 2 correspond to Black and White, Grayscale and Color, respectively.

### 1, 0

If *Kywrđ* = TMOD, the values 1 and 0 determine whether bitmap (1) or stroke text (0) fonts will be used

## Menu Paths

Utility Menu>PlotCtrls>HardCopy>ToFile

**TIME**, *TIME*

**Sets the time for a load step.**

[SOLUTION: Load Step Options](#)

MP ME ST PR PRN <> <> <> EM <> DY PP <> EME MFS

**TIME**

Time at the end of the load step.

**Command Default**

Previous *TIME* + 1.0 (at each load step), corresponding to the load step number.

**Notes**

Associates the boundary conditions at the end of the load step with a particular *TIME* value.

*TIME* must be a positive, nonzero, monotonically increasing quantity that "tracks" the input history. Units of time should be consistent with those used elsewhere (for properties, creep equations, etc.).

Typically, for the first load step *TIME* defaults to 1. However, for the first load step of a reduced transient analysis (**ANTYPE**,TRANS and **TRNOPT**,REDUC) or mode superposition transient analysis (**ANTYPE**,TRANS and **TRNOPT**,MSUP), the **TIME** command is ignored and a static solution is performed at *TIME* = 0.

For a full transient analyses, the command's default behavior does not apply. You must specify a time for each load step and it must be greater than the time at the end of the prior load step.

*TIME* does not apply to modal (**ANTYPE**,MODAL), harmonic response (**ANTYPE**,HARMIC), or substructure (**ANTYPE**,SUBSTR) analyses.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Basic**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substps**

**Main Menu>Solution>Time Controls>Solution Time**

**TIMERANGE**, *TMIN*, *TMAX*

**Specifies the time range for which data are to be stored.**

[POST26: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**TMIN**

Minimum time (defaults to first time (or frequency) point on the file).

**TMAX**

Maximum time (defaults to last time (or frequency) point on the file).

**Command Default**

Include all time (or frequency) points in the range.

**Notes**

Defines the time (or frequency) range for which data are to be read from the file and stored in memory. Use the **NSTORE** command to define the time increment.

**Menu Paths**

**Main Menu>TimeHist Postpro>Settings>Data**

**TIMINT**, *Key, Lab*

**Turns on transient effects.**

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

**Key**

Transient effects key:

**OFF**

No transient effects (static or steady-state).

**ON**

Include transient (mass or inertia) effects.

**Lab**

Degree of freedom label:

**ALL**

Apply this key to all appropriate labels (default).

**STRUC**

Apply this key to structural DOFs.

**THERM**

Apply this key to thermal DOFs.

**ELECT**

Apply this key to electric DOFs.

**MAG**

Apply this key to magnetic DOFs.

**FLUID**

Apply this key to fluid DOFs.

**Command Default**

Include transient effects (ON) if **ANTYPE,TRANS**, exclude transient effects (OFF) if **ANTYPE,STATIC**.

## Notes

Indicates whether this load step in a full transient analysis should use time integration, that is, whether it includes transient effects (e.g. structural inertia, thermal capacitance) or whether it is a static (steady-state) load step for the indicated DOFs. Transient initial conditions are introduced at the load step having *KEY* = ON. Initial conditions are then determined from the previous two substeps. Zero initial velocity and acceleration are assumed if no previous substeps exist. See the *Structural Analysis Guide*, the *Thermal Analysis Guide*, and the *Low-Frequency Electromagnetic Analysis Guide* for details.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters**

**Main Menu>Solution>Analysis Type>Sol'n Controls>Transient**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay**

**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters**

## TIMP, ELEM, CHGBND, IMLEVEL

Improves the quality of tetrahedral elements that are not associated with a volume.

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **ELEM**

Identifies the tetrahedral elements to be improved. Valid values are ALL and P. If *ELEM* = ALL (default), improve all selected tetrahedral elements. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

### **CHGBND**

Specifies whether to allow boundary modification. Boundary modification includes such things as changes in the connectivity of the element faces on the boundary and the addition of boundary nodes. (Also see the Notes section below for important usage information for *CHGBND*.)

**0**

Do not allow boundary modification.

**1**

Allow boundary modification (default).

### **IMLEVEL**

Identifies the level of improvement to be performed on the elements. (Improvement occurs primarily through the use of face swapping and node smoothing techniques.)

**0**

Perform the least amount of swapping/smoothing.

**1**

Perform an intermediate amount of swapping/smoothing.



- 2 Perform the greatest amount of swapping/smoothing.
- 3 Perform the greatest amount of swapping/smoothing, plus additional improvement techniques (default).

## Notes

The **TIMP** command enables you to improve a given tetrahedral mesh by reducing the number of poorly-shaped tetrahedral elements (in particular, the number of sliver tetrahedral elements)--as well as the overall number of elements--in the mesh. It also improves the overall quality of the mesh.

**TIMP** is particularly useful for an imported tetrahedral mesh for which no geometry information is attached.

Regardless of the value of the *CHGBND* argument, boundary mid-nodes can be moved as long as you are not using p-method analysis. When *CHGBND* = 0 and you are using p-method analysis, boundary mid-nodes cannot be moved. (ANSYS issues an error message if it would be necessary to move boundary mid-nodes in order to generate valid quadratic elements.)

When loads or constraints have been placed on boundary nodes or mid-nodes, and boundary mid-nodes are later moved, ANSYS issues a warning message to let you know that it will not update the loads or constraints.

No boundary modification is performed if shell or beam elements are present in the mesh, even when *CHGBND* = 1.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Modify Mesh>Improve Tets>Detached Elems**

**TINTP**, *GAMMA*, *ALPHA*, *DELTA*, *THETA*, *OSLM*, *TOL*, --, --, *AVSMOOTH*, *ALPHAF*, *ALPHAM*

**Defines transient integration parameters.**

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

### **GAMMA**

Amplitude decay factor for 2nd order transient integration, e.g., structural dynamics (used only if *ALPHA*, *DELTA*, *ALPHAF*, and *ALPHAM* are blank). Defaults to 0.005.

### **ALPHA**

2nd order transient integration parameter (used only if *GAMMA* is blank). Defaults to 0.2525.

### **DELTA**

2nd order transient integration parameter (used only if *GAMMA* is blank). Defaults to 0.5050.

### **THETA**

1st order transient (e.g., thermal transient) integration parameter. Defaults to 0.5 if **SOLCONTROL** is OFF. Defaults to 1.0 if **SOLCONTROL** is ON.

**OSLM**

Specifies the oscillation limit criterion for automatic time stepping of 1st order transients (e.g., thermal transients). Defaults to 0.5 with a tolerance of *TOL*.

**TOL**

Tolerance applied to *OSLM*. Defaults to 0.0.

--,--

Unused fields.

**AVSMOOTH**

Smooth flag option:

**0**

Include smoothing of initial velocity (1st order system) or initial acceleration (2nd order system) (default).

**1**

Do not include smoothing.

**ALPHAF**

Interpolation factor in HHT algorithm for force and damping terms (used only if *GAMMA* is blank). Defaults to 0.005.

**ALPHAM**

Interpolation factor in HHT algorithm for inertial term (used only if *GAMMA* is blank). Defaults to 0.0.

## Notes

Used to define the transient integration parameters. For more information on transient integration parameters, refer to the [Theory Reference for the Mechanical APDL and Mechanical Applications](#).

For structural transient analyses, you may choose between the Newmark and HHT time integration methods (see the **TRNOPT** command). In this case, if *GAMMA* is input and the integration parameters *ALPHA*, *DELTA*, *ALPHAF*, and *ALPHAM* are left blank, the program will calculate the integration parameters. Alternatively, you can input these integration parameters directly on this command. However, for the unconditional stability and second order accuracy of the time integration, these parameters should satisfy a specific relationship, as described in [Description of Structural and Other Second Order Systems of the Theory Reference for the Mechanical APDL and Mechanical Applications](#).

In a transient piezoelectric analysis, required input for this command is *ALPHA* = 0.25, *DELTA* = 0.5, and *THETA* = 0.5. For a coupled electromagnetic-circuit transient analysis, use *THETA* = 1.0, the default value, to specify the backward Euler method.

The default values given for this command assume **SOLCONTROL,ON** (the default). If your analysis is using **SOLCONTROL,OFF**, you may need to adjust some defaults to prevent the solution from diverging. See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL,ON** and **SOLCONTROL,OFF**.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay**

**Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Transient**  
**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay**  
**Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters**

---

**/TITLE**, *Title*

**Defines a main title.**

[DATABASE: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Title**

Input up to 72 alphanumeric characters. Parameter substitution may be forced within the title by enclosing the parameter name or parametric expression within percent (%) signs.

## Notes

The title is carried through the printout and written on various files. The title written to a file is the title defined at that time. Special characters may be used within the title text. Subtitles may also be defined [[/STITLE](#)].

This command is valid in any processor.

## Menu Paths

**Utility Menu>File>Change Title**

---

**/TLABEL**, *XLOC*, *YLOC*, *Text*

**Creates annotation text (GUI).**

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**XLOC**

Text *X* starting location (-1.0 < *X* < 1.6).

**YLOC**

Text *Y* starting location (-1.0 < *Y* < 1.0).

**Text**

Text string (60 characters maximum). Parameter substitution may be forced within the text by enclosing the parameter name or parametric expression within percent (%) signs.

## Notes

Defines annotation text to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (*Jobname.LOG*) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the [/INPUT](#) command).

All text is shown on subsequent displays unless the annotation is turned off or deleted. Use the [/TSPEC](#) command to set the attributes of the text.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

---

**TOCOMP**, *Refname*, *Type*, *NUMLC*, *LCARR*

**Defines single or multiple compliance as the topological optimization function.**

OPTIMIZATION: Specifications

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Refname*

Reference name (8 character string). **TOCOMP**,*Refname* with the other fields blank deletes the defined compliance function.

### *Type*

Sets the type of compliance function. Valid types are:

#### **SINGLE**

Sets the topological optimization function for *Refname* to single compliance. *NumLC* is the load case number.

#### **MULTIPLE**

Sets the topological optimization function to weighted summation of individual compliances. *NumLC* is the number of load cases considered

### *NUMLC*

For *Type* = Single, *NumLC* is the load case identifier. For *Type* = Multiple, *NumLC* is the total number of load cases considered.

### *LCARR*

Used when *Type* = Multiple. The array (of dimension *NumLC*) defining the load case weights. Array name must be enclosed in % signs: %WEIGHT%.

## Notes

The function defined using **TOCOMP** is used as an objective or constraint for the topological optimization, as defined further in **TOVAR**. When used as the objective, the optimization distributes the material so that the compliance measure is minimized (stiffness is maximized).

## Menu Paths

Main Menu>Topological Opt>Set Up>Advanced Opt>Topo Function

Main Menu>Topological Opt>Set Up>Basic Opt

---

**TODEF**, *ACCUR*

**Defines parameters for and initializes topological optimization.**

OPTIMIZATION: Specifications

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**ACCUR**

Accuracy used for termination and convergence checking. Default = 0.0005.

**Notes**

Initializes the topological optimization problem and defines the accuracy for the solution. Issue this command before **TOLOOP**.

**Menu Paths**

Main Menu>Topological Opt>Run

**TOEXE**

**Executes one topological optimization iteration.**

OPTIMIZATION: Run

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**Notes**

Runs one topological optimization iteration, leading to the prediction of a new shape, defined by means of element densities. Before issuing **TOEXE**, you must perform a static or modal analysis solution depending on the type of objective and constraints you defined. **TOEXE** performs a convergence test based on relative objective and element density change with an accuracy as specified by **TODEF**.

**Menu Paths**

This command cannot be accessed from a menu.

**TOFFST**, *VALUE*

**Specifies the temperature offset from absolute zero to zero.**

SOLUTION: Analysis Options

AUX12: General Radiation

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

**VALUE**

Degrees between absolute zero and zero of temperature system used (should be positive).

## Notes

Specifies the difference (in degrees) between absolute zero and the zero of the temperature system used. Absolute temperature values are required in evaluating certain expressions, such as for creep, swelling, radiation heat transfer, [MASS71](#), etc. (The offset temperature is not used in evaluating emissivity.) Examples are 460° for the Fahrenheit system and 273° for the Celsius system. The offset temperature is internally included in the element calculations and does not affect the temperature input or output. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Temperature Units**

**Main Menu>Preprocessor>Material Props>Temperature Units**

**Main Menu>Preprocessor>Radiation Opts>Solution Opt**

**Main Menu>Radiation Opt>Radiosity Meth>Solution Opt**

**Main Menu>Solution>Analysis Type>Analysis Options**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Temperature Units**

**Main Menu>Solution>Radiation Opts>Solution Opt**

---

**TOFREQ**, *Refname*, *Type*, *Nfreq*, *Frqarr*, *Targval*

**Defines single or mean frequency formulation as the topological optimization function.**

[OPTIMIZATION: Specifications](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Refname*

Reference name (8 character string). **TOFREQ**,*Refname* with other fields blank deletes the defined compliance function.

### *Type*

Defines type of frequency topological optimization function. Valid types are:

#### **SINGLE**

Defines single frequency as the topological optimization function.

#### **WEIGHTED**

Defines a weighted summation of individual frequencies as the topological optimization function.

#### **RECIPROCAL**

Uses a reciprocal formulation as the topological optimization function.

#### **EUCLIDEAN**

Uses a Euclidean Norm formulation as the topological optimization function.

### *Nfreq*

For *Type* = SINGLE, *Nfreq* is the frequency identifier. For *Type* = WEIGHTED, RECIPROCAL, or EUCLIDEAN, *Nfreq* is the total number of frequencies considered.

### *Frqarr*

For *Type* = WEIGHTED or RECIPROCAL, *Frqarr* is the array of dimension *Nfreq* that defines the weights for mean frequency formulation. For *Type* = EUCLIDEAN, *Frqarr* is the array (of dimension *Nfreq*)

defining the target values for mean frequency formulation. The array name must be enclosed in % signs: %ARRAY%.

### ***Targval***

For *Type* = RECIPROCAL, *Targval* specifies the target value for mean frequency formulation.

<b>Type</b>	<b>Nfreq</b>	<b>Frqarr</b>	<b>Targval</b>
SINGLE	Frequency identifier	N/A	N/A
WEIGHTED	Total number of frequencies	Defines weights	N/A
RECIPROCAL	Total number of frequencies	Defines weights	Target value
EUCLIDEAN	Total number of frequencies	Defines target values	N/A

## **Notes**

Single, when the frequency function is set as the objective (**TOVAR**), distributes material so that the single frequency specified is maximized.

Weighted (available only as an objective function) distributes material so that the specified mean frequency formulation is maximized.

Reciprocal (available only as an objective function) is a mean frequency formulation that defines a smoother function than Single. It is best used when two modes whose eigenfrequencies occur in the given mean formulation exchange their orders during optimization. The eigenfrequency that is closest to the target frequency (set with *Targval*) experiences the largest increase and is maximized.

Euclidean Norm Formulation (available only as an objective function) is used to shift single or multiple eigenfrequencies up. The mean eigenfrequency is utilized to smooth the optimization process. This must be used as the objective where the optimization procedure distributes material so that the mean frequency is minimized.

If the optimization problem does not converge, try specifying a different type of frequency formulation. Each formulation uses different methods to calculate the frequency optimization, so depending on your specific problem, one might work better than another. See the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for more information on these formulations.

## **Menu Paths**

**Main Menu>Topological Opt>Set Up>Advanced Opt>Topo Function**

**TOGRAPH**, *Type*, *Refname*

**Plots iteration solution of topological optimization.**

OPTIMIZATION: Display

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

*Type*

**OBJ**

Plot objective history versus iteration number.

**CON**

Plot constraint history versus iteration number.

**Refname**

Used if multiple constraints were defined. Defaults to the first constraint name.

**Notes**

Plots all iterations of the topological optimization solution, using either objective or constraint history.

**Menu Paths**

**Main Menu>Topological Opt>Graph History**

---

**TOLIST**

**Lists all topological optimization functions currently defined.**

OPTIMIZATION: Display

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**Notes**

Provides a list of all defined topological functions, in the order they were defined.

**Menu Paths**

**Main Menu>Topological Opt>Set Up>Advanced Opt>List Functions**

---

**TOLOOP**, *NITER*, *PLOT*

**Execute several topological optimization iterations.**

OPTIMIZATION: Run

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

***NITER***

Number of iterations to be performed (maximum of 100). Default is 1.

***PLOT***

Sets display of topological densities for each iteration:

**0**

Do not display the results of each iteration.

**1**

Display the results of each iteration.

**Command Default**

**TOLOOP**,1,0.



## Notes

Invokes a macro to solve, postprocess, and plot each iteration. For compliance cases, you must write at least one load step (using **LSWRITE**) before issuing this command. **LSSOLVE** is used for static analyses if there are multiple load steps. The Block Lanczos eigensolver is used for modal analyses. **PLNSOL**, **TOPO** and **TOEXE** are used for each iteration. The macro terminates when either the number of iterations or the convergence criteria is met, as specified in **TODEF**, *ACCUR*.

## Menu Paths

Main Menu>Topological Opt>Run

**\*TOPER**, *ParR*, *Par1*, *Oper*, *Par2*, *FACT1*, *FACT2*, *CON1*

Operates on table parameters.

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *ParR*

Name of the resulting table parameter. The command will create a table array parameter with this name. Any existing parameter with this name will be overwritten.

### *Par1*

Name of the first table parameter.

### *Oper*

The operation to be performed: ADD. The operation is:  $ParR(i,j,k) = FACT1*Par1(i,j,k) + FACT2 *Par2(i,j,k) + CON1$

### *Par2*

Name of the second table parameter.

### *FACT1*

The first table parameter multiplying constant. Defaults to 1.

### *FACT2*

The second table parameter multiplying constant. Defaults to 1.

### *CON1*

The constant increment for offset. Defaults to 0.

## Notes

**\*TOPER** operates on table parameters according to:  $ParR(i,j,k) = FACT1*Par1(i,j,k) + FACT2 *Par2(i,j,k) + CON1$

*Par1* and *Par2* must have the same dimensions and the same variable names corresponding to those dimensions. *Par1* and *Par2* must also have identical index values for rows, columns, etc.

If you want a local coordinate system for the resulting array, you must dimension it as such using the **\*DIM** command before issuing **\*TOPER**.

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Array Operations>Table Operations

---

### TOPLOT, *AVRG*

Plot current topological density distribution.

OPTIMIZATION: Display

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### *AVRG*

Sets **TOPLOT** to show smoothed nodal solution or non-smoothed element solution.

**0**

Plot smoothed nodal solution. Default.

**1**

Plot non-smoothed element solution.

## Notes

Invokes a macro to postprocess the current element pseudo densities for topological optimization. **TOPLOT,0** uses **PLNSOL,TOPO**, and **TOPLOT,1** uses **PLESOL,TOPO** to plot the current density distribution predicted by the topological optimization.

## Menu Paths

Main Menu>Topological Opt>Plot Dens Unavg

Main Menu>Topological Opt>Plot Densities

---

### TOPRINT, *Type, Refname*

Print iteration solution history of topological optimization.

OPTIMIZATION: Display

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### *Type*

**OBJ**

Print objective history versus iteration number.

**CON**

Print constraint history versus iteration number.

#### *Refname*

Used if multiple constraints were defined. Defaults to the first constraint name.

## Notes

Prints one iteration of the topological optimization solution, using either objective or constraint history.

## Menu Paths

Main Menu>Topological Opt>Print History

---

## TORQ2D

Calculates torque on a body in a magnetic field.

POST1: Magnetics Calculations

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

## Notes

**TORQ2D** invokes an ANSYS macro which calculates mechanical torque on a body in a magnetic field. The body must be completely surrounded by air (symmetry permitted), and a closed path [**PATH**] passing through the air elements surrounding the body must be available. A counterclockwise ordering of nodes on the **PPATH** command will give the correct sign on the torque result. The macro is valid for 2-D planar analysis.

The calculated torque is stored in the parameter TORQUE. A node plot showing the path is produced in interactive mode. The torque is calculated using a Maxwell stress tensor approach. Path operations are used for the calculation, and all path items are cleared upon completion. See the **TORQC2D** command for torque calculation based on a circular path.

## Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>Torque

---

## TORQC2D, RAD, NUMN, LCSYS

Calculates torque on a body in a magnetic field based on a circular path.

POST1: Magnetics Calculations

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

### **RAD**

Radius of the circular path. The nodes for the path are created at this radius.

### **NUMN**

Number of nodes to be created for the circular path. The greater the number of nodes, the higher the accuracy of the torque evaluation. Defaults to 18.

### **LCSYS**

(Optional) Local coordinate system number to be used for defining the circular arc of nodes and the path. Defaults to 99. (If a local system numbered 99 already exists, it will be overwritten by this default.)

## Notes

**TORQC2D** invokes an ANSYS macro which calculates the mechanical torque on a body using a circular path. It is used for a circular or cylindrical body such as a rotor in an electric machine. The body must be centered about the global origin and must be surrounded by air elements. The air elements surrounding the path at radius *RAD* must be selected, and elements with a high-permeability material should be unselected prior to using the macro. The macro is valid for 2-D planar analyses only. For a harmonic analysis, the macro calculates the time-average torque. Radial symmetry models are allowed, i.e., the model need not be a full 360° model.

The calculated torque is stored in the parameter TORQUE. If the model is not a full 360° model, TORQUE should be multiplied by the appropriate factor (such as 4.0 for a 90° sector) to obtain the total torque. A node plot showing the path is produced in interactive mode.

The torque is calculated via a circular path integral of the Maxwell stress tensor. The circular path and the nodes for the path are created by the macro at the specified radius *RAD*. Path operations are used for the calculation, and all path items are cleared upon completion. See the **TORQ2D** command for torque calculation based on an arbitrary, non-circular path.

## Menu Paths

**Main Menu>General Postproc>Elec&Mag Calc>Path Based>Circular Torq**

---

**TORQSUM**, *Cnam1*, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*, *Cnam9*

Summarizes electromagnetic torque calculations on element components.

[POST1: Magnetics Calculations](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

*Cnam1*, *Cnam2*, *Cnam3*, . . . , *Cnam9*

Names of existing element components for which Maxwell or virtual work boundary conditions were applied in the preprocessor. Must be enclosed in single quotes (e.g., 'CNAM1') when the command typed in the command input box.

## Notes

**TORQSUM** invokes an ANSYS macro that summarizes the Maxwell and virtual work torque values. The element components must have had appropriate Maxwell or virtual work boundary conditions established in the preprocessor prior to solution in order to retrieve torques (see the **FMAGBC** command). The torque values are stored on a per-element basis for the adjacent air layer elements surrounding the components and are retrieved and summed by the macro. For a harmonic analysis, the calculated torque represents a time-average value.

**TORQSUM** is valid only for 2-D planar analysis.

## Menu Paths

**Main Menu>General Postproc>Elec&Mag Calc>Component Based>Torque**

---

**TORUS**, *RAD1*, *RAD2*, *RAD3*, *THETA1*, *THETA2*

**Creates a toroidal volume.**

PREP7: Primitives

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***RAD1*, *RAD2*, *RAD3***

Three values that define the radii of the torus. You can specify the radii in any order. The smallest of the values is the inner minor radius, the intermediate value is the outer minor radius, and the largest value is the major radius. (There is one exception regarding the order of the radii values--if you want to create a solid torus, specify zero or blank for the inner minor radius, in which case the zero or blank *must* occupy either the *RAD1* or *RAD2* position.) At least two of the values that you specify must be positive values; they will be used to define the outer minor radius and the major radius. See the diagram in the Notes section for a view of a toroidal sector showing all radii.

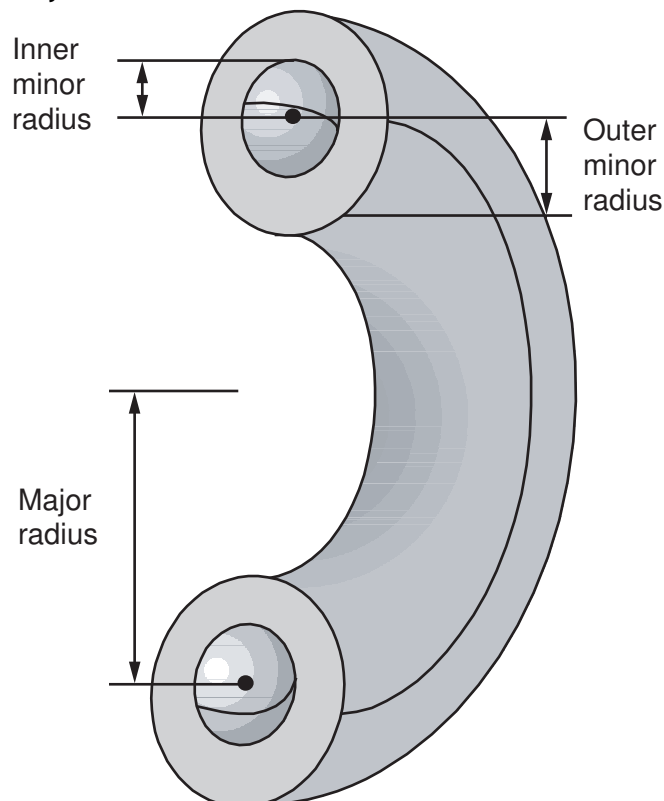
***THETA1*, *THETA2***

Starting and ending angles (either order) of the torus. Used for creating a toroidal sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0° and the ending angle defaults to 360°.

**Notes**

Defines a toroidal volume centered about the working plane origin. A solid torus of 360° will be defined with four areas, each area spanning 180° around the major and minor circumference.

To create the toroidal sector shown below, the command **TORUS**,5,1,2,0,180 was issued. Since "1" was the smallest radii value specified, it defined the inner minor radius; since "2" was the intermediate radii value specified, it defined the outer minor radius; and since "5" was the largest radii value specified, it defined the major radius. The values "0" and "180" defined the starting and ending angles of the torus.



## Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Torus

---

## TOSTAT

Displays topological optimization status and results information.

OPTIMIZATION: Display

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

## Notes

Displays information for defined functions from **TOVAR**, **TODEF**, and **TOTYPE**, and results information such as final volume, number of iterations, and convergence status.

## Menu Paths

Main Menu>Topological Opt>Set Up>Advanced Opt>Status

Main Menu>Topological Opt>Status

---

## TOTAL, *NTOT*, *NRMDF*

Specifies automatic MDOF generation.

SOLUTION: Master DOF

MP ME ST PR PRN <> <> <> <> <> PP <> EME MFS

### *NTOT*

Total number of master degrees of freedom to be used in the analysis, including specified (NS, see below) master degrees of freedom. *NTOT* must be greater than NS if any automatic generation is to be done.

### *NRMDF*

Rotational masters key:

**0**

Include all degrees of freedom in automatic master selection.

**1**

Exclude rotational degrees of freedom (and VOLT degrees of freedom in a piezoelectric analysis) from automatic selection.

## Command Default

Do not use any automatically generated MDOF.

## Notes

Specifies automatic master degree of freedom (MDOF) generation. The limit on the number of MDOF is equal to the maximum in-memory wavefront size (see the *Basic Analysis Guide*). If NS is defined as the number of master degrees of freedom specified with the **M** or **MGEN** command, *NTOT*-NS additional master

degrees of freedom will be automatically generated *during the solution phase* if **TOTAL** is used. NS may be zero, i.e., all master degrees of freedom can be automatically generated. After the solution phase, generated masters become specified masters ( $NS = NTOT$ ) so that they may be listed, displayed, modified, etc. The **TOTAL** command is ignored in subsequent solutions unless masters are deleted, such that  $NS < NTOT$ . If used in SOLUTION, this command is valid only within the first load step.

During the matrix triangulation (wavefront) operation, the first  $NTOT$  degrees of freedom are temporarily identified as masters and then are replaced as degrees of freedom with lower K/M ratios are found. Degrees of freedom matching the user specified set (if any) are permanently identified. The wavefront builds to  $NTOT$  and will have a minimum (and final) value of  $NTOT$ . The final set of automatic masters identified will be those corresponding to the lowest modes of the structure.

Constrained degrees of freedom are excluded from the automatic master selection. Constraints may be defined to prevent undesirable modes from being present (thus preventing the corresponding MDOF from being selected). For example, if symmetry constraints are imposed, degrees of freedom producing only symmetric modes will be selected. In-plane rotational degrees of freedom for shell elements lying in a global plane are automatically excluded. All rotational degrees of freedom can be excluded during the automatic selection if desired.

If automatic master selection is used in the reduced linear transient dynamic (**ANTYPE,TRANS**) analysis or the reduced harmonic response (**ANTYPE,HARMIC**) analysis, be sure to force the selection [**M**] of any degrees of freedom having nonzero displacement or force inputs. If automatic master selection is used in the super-element generation pass (**ANTYPE,SUBSTR**), be sure to force the selection of connection points to nonsuper-elements.

Automatically selected masters are shown in the solution listing (and not in preprocessing listings) as follows:

- in the reduced eigenvector solution for modal (**ANTYPE,MODAL**).
- in the reduced displacement solution for harmonic response (**ANTYPE,HARMIC**).
- in the reduced displacement solution for linear transient dynamic (**ANTYPE,TRANS**).
- in the matrix or load vector printout for substructures (**ANTYPE,SUBSTR**).

In the substructure generation pass (**ANTYPE,SUBSTR**), a mass matrix must be available if the **TOTAL** option is to be used.

The **TOTAL** command is used in conjunction with a specific solver developed in an earlier version of the ANSYS program. ANSYS automatically chooses this solver when the **TOTAL** command is executed. Other solvers are not available with the **TOTAL** command.

We recommend that you define as many MDOF as you can based on your knowledge of the dynamic characteristics of the structure and let the program choose a few additional masters with the **TOTAL** command.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Master DOFs>Program Selected**

**Main Menu>Solution>Master DOFs>Program Selected**

**TOTYPE**, *Type*

**Specifies solution method for topological optimization.**

[OPTIMIZATION: Specifications](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**Type**

Choose the solution method to use:

**OC**

Use Optimality Criteria (OC) approach. (Default)

**SCP**

Use Sequential Convex Programming (SCP) approach.

**Notes**

The OC approach is applicable to problems with only volume as the constraint (or “volume as the only constraint”). The SCP approach is applicable to all valid combinations of objectives and constraints. See [TOVAR](#) for a description of valid combinations.

**Menu Paths**

**Main Menu>Topological Opt>Run**

**TOVAR**, *Refname, Type, LOWER, UPPER, Boundtype*

**Specifies the objective and constraints for the topological optimization problem.**

[OPTIMIZATION: Specifications](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**Refname**

Reference name (8 character string), previously defined with [TOCOMP](#) and [TOFREQ](#), or VOLUME (reserved name; default).

**Type**

Valid types for this command are:

**OBJ**

Specifies the objective for the topological optimization problem. The *Refname*(s) specified must already be defined ([TOCOMP](#), *Refname*, or [TOFREQ](#)) before defining your constraint(s). *LOWER*, *UPPER*, and *Boundtype* are not used if the specified *Refname* is the objective. To specify total volume as the objective function, use VOLUME as the *Refname*. For single or multiple compliance, see [TOCOMP](#). For frequency formulations (single, weighted mean, reciprocal mean, or Euclidean normal), see [TOFREQ](#).

**CON**

Specifies the constraint for the topological optimization problem. Requires a previously defined objective function ([TOVAR](#), *Refname*, OBJ). *LOWER*, *UPPER*, and *Boundtype* must be specified as listed below.



**DEL**

Deletes the previously defined topological optimization objective or constraint named in *Refname*. *LOWER*, *UPPER*, and *Boundtype* are not used.

**LOWER**

Lower bound for the constraint (*Type* = CON).

**UPPER**

Upper bound for the constraint (*Type* = CON). Default is no defined upper bound.

**Boundtype**

For *Type* = CON, specifies whether the specified bounds are actual values, or indicate percentages.

**PERCENT**

Indicates that the values specified in *LOWER* and *UPPER* should be treated as percentages of the original value. The original value is 100%; only values less than 100 are valid. Depending on the type of constraint, this value indicates a decrease or increase of the initial values. For example, specify 50 to reduce the volume by 50%, or specify 30 to increase the compliance by 30%.

**ACTUAL**

Indicates that the values specified in *LOWER* and *UPPER* should be treated as actual values.

**Command Default**

TOVAR,VOLUME,OBJ

**Notes**

You must set the objective first, then set the constraint(s).

If a single or multiple compliance function (see **TOCOMP**) is specified as objective function, only the VOLUME function is allowed as a constraint.

If a single, weighted mean, reciprocal mean, or euclidean norm frequency (see **TOFREQ**) objective is selected, only the VOLUME function is allowed as a constraint.

If the VOLUME function is selected as the objective function, then only a single or multiple compliance constraint (see **TOCOMP**) is allowed as constraint function. In this case, a multiple single compliance constraint definition is also possible. LOWER is not applicable if *Type* = OBJ or DEL, and is not used if *Boundtype* = PERCENT.

UPPER is not applicable if *Type* = OBJ or DEL. If *Boundtype* = PERCENT, and *RefName* = VOLUME, UPPER specifies the percentage of volume reduction. If *Boundtype* = PERCENT, and *RefName* is a single or multiple compliance function, UPPER specifies the percentage of compliance increase.

If *Boundtype* = PERCENT, the valid range for *UPPER* is [5%, 95%].

**Menu Paths**

**Main Menu>Topological Opt>Set Up>Advanced Opt>Topo Objective**

**Main Menu>Topological Opt>Set Up>Advanced Opt>TopoConstraint>By Percentage**

**Main Menu>Topological Opt>Set Up>Advanced Opt>TopoConstraint>By Value**

**Main Menu>Topological Opt>Set Up>Advanced Opt>TopoConstraint>Delete**

**Main Menu>Topological Opt>Set Up>Basic Opt**

**TRANS**, *Fname*, *Ext*, --

Reformats File .GRPH for improved performance with plotters.

[DISPLAY: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to TRAN33.

***Ext***

Filename extension (8 character maximum).

--

Unused field.

**Notes**

Reformats current *Fname* .GRPH data (based on color) for improved performance with pen plotters.

**Menu Paths**

It is part of the **DISPLAY** command.

**TRANSFER**, *KCNTO*, *INC*, *NODE1*, *NODE2*, *NINC*

Transfers a pattern of nodes to another coordinate system.

[PREP7: Nodes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***KCNTO***

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system.

***INC***

Increment all nodes in the given pattern by *INC* to form the transferred node pattern.

***NODE1*, *NODE2*, *NINC***

Transfer nodes from pattern beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and the pattern is all selected nodes [NSEL]. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component may be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

**Notes**

Transfers a pattern of nodes from one coordinate system to another. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. Coordinate values are interpreted in the active coordinate system and are transferred directly.

A model generated in one coordinate system may be transferred to another coordinate system. The user may define several coordinate systems (translated and rotated from each other), generate a model in one coordinate system, and then repeatedly transfer the model to other coordinate systems. The model may be generated in any type of coordinate system (Cartesian, cylindrical, etc.) and transferred to any other type of coordinate system. Coordinate values (X, Y, Z, or R,  $\theta$ , Z, or etc.) of the model being transferred are interpreted in the *active* coordinate system type, regardless of how they were generated. Values are transferred directly and are interpreted according to the type of coordinate system being transferred to. For example, transferring from a Cartesian coordinate system to a cylindrical coordinate system (not recommended) would cause X = 2.0 and Y = 3.0 values to be directly interpreted as R = 2.0 and  $\theta$  = 3.0 values, respectively.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Nodes**

**\*TREAD**, *Par*, *Fname*, *Ext*, *--*, *NSKIP*

**Reads data from an external file into a table array parameter.**

[APDL: Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Par*

Table array parameter name as defined by the **\*DIM** command.

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

File name has no default.

### *Ext*

Filename extension (8 character maximum).

Extension has no default.

### *--*

Unused field.

### *NSKIP*

Number of comment lines at the beginning of the file being read that will be skipped during the reading. Default = 0.

## Notes

Use this command to read in a table of data from an external file into an ANSYS table array parameter. The external file may be created using a text editor or by an external application or program. The external file must be in tab-delimited, blank-delimited, or comma-delimited format to be used by **\*TREAD**. The ANSYS TABLE type array parameter must be defined before you can read in an external file. See **\*DIM** for more information.

This command is not applicable to 4- or 5-D tables.

## Menu Paths

Utility Menu>Parameters>Array Parameters>Read from File

### TREF, TREF

Defines the reference temperature for the thermal strain calculations.

[SOLUTION: Load Step Options](#)

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

#### TREF

Reference temperature for thermal expansion.

#### Note

If the uniform temperature ([TUNIF](#)) is not specified, it is also set to this value.

## Command Default

Reference temperature is 0.0 degrees.

## Notes

Defines the reference temperature for the thermal strain calculations in structural analyses and explicit dynamic analyses. Thermal strains are given by  $\alpha \cdot (T - TREF)$ , where  $\alpha$  is the coefficient of thermal expansion (for more on this see the [Theory Reference for the Mechanical APDL and Mechanical Applications](#)). Input the strain via ALPX, ALPY, ALPZ (the secant or mean coefficient value), or CTEX, CTEY, CTEZ (the instantaneous coefficient value), or the thermal strain value (THSX, THSY, THSZ). T is the element temperature. If  $\alpha$  is temperature-dependent, TREF should be in the range of temperatures you define using the [MPTEMP](#) command.

Reference temperatures may also be input *per material* by specifying a value on the [MP](#) material property command:

[MP](#),REFT,*MAT*,*C0*.

Only a constant (non-temperature-dependent) value is valid. The value input on the [TREF](#) command applies to all materials not having a specified material property definition.

To convert temperature-dependent secant coefficients of thermal expansion (SCTE) data (properties ALPX, ALPY, ALPZ) from the definition temperature to the reference temperature defined via a [TREF](#) (or [MP](#),REFT) command, issue the [MPAMOD](#) command.

This command is also valid in PREP7.

## Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Reference Temp

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Reference Temp

**Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Reference Temp**  
**Main Menu>Solution>Define Loads>Settings>Reference Temp**  
**Main Menu>Solution>Load Step Opts>Other>Reference Temp**  
**Main Menu>Solution>Loading Options>Reference Temp**

## **/TRIAD**, *Lab*

Shows the global XYZ coordinate triad on displays.

GRAPHICS: Labeling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Lab*

Display triad as follows:

#### **ORIG**

Display triad at global origin (default).

#### **OFF**

Turn off triad display.

#### **LBOT**

Display triad in lower left screen corner.

#### **RBOT**

Display triad in lower right screen corner.

#### **LTOP**

Display triad in upper left screen corner.

#### **RTOP**

Display triad in upper right screen corner.

## **Notes**

For efficiency, ANSYS 3-D graphics logic maintains a single data structure (segment), which includes the triad as a 3-D data object. If a 3-D device is involved (**/SHOW,3D**), and the ANSYS graphics are not being displayed as multi-plots, then the triad location is determined by the view settings for Window #1. A request for triad display anywhere except for the origin could yield an improper display in windows 2 through 5. The program displays the same segment in all windows. The view settings of each window constitute the only difference in the display in the active windows.

This command is valid in any processor.

## **Menu Paths**

**Utility Menu>PlotCtrls>Window Controls>Reset Window Options**  
**Utility Menu>PlotCtrls>Window Controls>Window Options**

**/TRLCY**, *Lab*, *TLEVEL*, *N1*, *N2*, *NINC***Specifies the level of translucency.**

GRAPHICS:Style

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***Lab***

Apply translucency level to the items specified by the following labels:

**ELEM**Elements. Use *N1*, *N2*, *NINC* fields for element numbers.**AREA**Solid model areas. Use *N1*, *N2*, *NINC* fields for area numbers.**VOLU**Solid model volumes. Use *N1*, *N2*, *NINC* fields for volume numbers.**ISURF**

Isosurfaces (surfaces of constant stress, etc., value). Translucency varies with result value, to a maximum of the specified translucency level.

**CM**Component group. Use *N1* for component name, ignore *N2* and *NINC*.**CURVE**Filled areas under curves of line graphs. Use *N1*, *N2*, *NINC* fields for curve numbers.**ZCAP**If **/TYPE**,*WN*,*ZCAP* is the current display type, then **/TRLCY**,*ZCAP*,*TLEVEL* will display the model in window *WN* with the portion of the model in front of the section plane displayed at the translucency level *TLEVEL*.**ON, OFF**

Sets the specified translucency display on or off. All other fields are ignored.

***TLEVEL***

Translucency level: 0.0 (opaque) to 1.0 (transparent).

***N1*, *N2*, *NINC***Used only with labels as noted above. Apply translucency level to *Lab* items numbered *N1* to *N2* (defaults to *N1*) in steps of *NINC* (defaults to 1). If *N1* is blank or ALL, apply specified translucency level to entire selected range. If *Lab* is CM, use component name for *N1* and ignore *N2* and *NINC*. A value of *N1* = P allows you to graphically pick elements, areas, and volumes. You can then assign translucency levels to the entities via the picker. The *Lab* and *TLEVEL* fields are ignored when translucency is applied by picking.**Command Default**

Zero translucency (opaque) level.

**Notes**Specifies the level of translucency for various items. Issue **/TRLCY**,DEFA to reset the default (0) translucency levels. This command is valid only on selected 2-D and 3-D graphics devices; see in the *Basic Analysis Guide* for more information on applying translucency.

For 2-D devices, ANSYS displays only the visible faces of the items being displayed. The information behind the facing planes is not displayed. Issuing the **/SHRINK** command will force the hardware to display information behind the translucent items.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Translucency

**TRNOPT**, *Method*, *MAXMODE*, *Dmpkey*, *MINMODE*, *MCout*, *TINTOPT*

**Specifies transient analysis options.**

[SOLUTION: Dynamic Options](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS  
[Product Restrictions](#)

### *Method*

Solution method for the transient analysis:

#### **FULL**

Full method (default).

#### **REDUC**

Reduced method.

#### **MSUP**

Mode superposition method.

#### **VT**

Variational Technology method.

### *MAXMODE*

Largest mode number to be used to calculate the response (for *Method* = MSUP). Defaults to the highest mode calculated in the preceding modal analysis.

### *Dmpkey*

Damping option (for *Method* = REDUC):

#### **DAMP**

Include the effects of damping if present (default).

#### **NODAMP**

Ignore the effects of damping, even if present.

### *MINMODE*

Smallest mode number to be used (for *Method* = MSUP). Defaults to 1.

### *MCout*

Modal coordinates output key (valid only for the mode superposition method MSUP):

#### **NO**

No output of modal coordinates (default).

#### **YES**

Output modal coordinates to the text file `jobname.MCF`.

**TINTOPT**

Time integration method for the transient analysis:

**NMK or 0**

Newmark algorithm (default).

**HHT or 1**

HHT algorithm (valid only for the full transient method).

**Notes**

Specifies transient analysis (**ANTYPE**,**TRANS**) options. If used in **SOLUTION**, this command is valid only within the first load step. Use the **TINTP** command to set transient integration parameters.

The **VT** option is valid for either thermal or structural analyses, where it attempts to reduce the total number of iterations. Both linear and nonlinear structural transient analyses are supported. The **VT** option is a **FULL** transient solution.

To include residual vectors in your mode-superposition transient analysis (*Method*=**MSUP**), specify **RESVEC,ON**.

This command is also valid in **PREP7**.

**Product Restrictions**

The **VT** option is only available with a HPC license.

Additional product restrictions for the **TRNOPT** command are shown in the table below.

Command Option Method	Available Products
FULL	MP ME ST <> <> <> <> <> EM <> <> PP <> EME MFS
REDUC	MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS
MSUP	MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

**Menu Paths**

**Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options**  
**Main Menu>Preprocessor>Loads>Analysis Type>New Analysis**  
**Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient**  
**Main Menu>Solution>Analysis Type>Analysis Options**  
**Main Menu>Solution>Analysis Type>New Analysis**  
**Main Menu>Solution>Analysis Type>Sol'n Controls>Transient**



**TRPDEL**, *NTRP1*, *NTRP2*, *TRPINC*

Deletes particle flow or charged particle trace points.

POST1:Trace Points

MP <> <> <> <> <> <> FL EM <> <> PP <> EME <>

*NTRP1*, *NTRP2*, *TRPINC*

Delete points from *NTRP1* to *NTRP2* (defaults to *NTRP1*) in steps of *TRPINC* (defaults to 1). If *NTRP1* = ALL, *NTRP2* and *TRPINC* are ignored and all trace points are deleted. If *NTRP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**Notes**

Deletes particle flow or charged particle trace points defined with the **TRPOIN** command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>General Postproc>Plot Results>Dele Trace Pt

**TRPLIS**, *NTRP1*, *NTRP2*, *TRPINC*, *Opt*

Lists the particle flow or charged particle trace points.

POST1:Trace Points

MP <> <> <> <> <> <> FL EM <> <> PP <> EME <>

*NTRP1*, *NTRP2*, *TRPINC*

List points from *NTRP1* to *NTRP2* (defaults to *NTRP1*) in steps of *TRPINC* (defaults to 1). If *NTRP1* = ALL, *NTRP2* and *TRPINC* are ignored and all trace points are listed. If *NTRP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

*Opt*

*Opt* = LOC lists the trace point number location (X, Y, Z). Default.

*Opt* = PART lists the trace point number particle settings (velocity, charge, mass).

**Notes**

Lists the particle flow or charged particle trace points in the active display coordinate system [**DSYS**]. Trace points are defined with the **TRPOIN** command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu>General Postproc>Plot Results>List Trace Pt

**TRPOIN**, *X, Y, Z, VX, VY, VZ, CHRG, MASS*

Defines a point through which a particle flow or charged particle trace will travel.

[POST1:Trace Points](#)

MP <> <> <> <> <> <> FL EM <> <> PP <> EME <>

***X, Y, Z***

Coordinate location of the trace point (in the active coordinate system). If  $X = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

***VX, VY, VZ***

Particle velocities in the X, Y and Z directions (in the active coordinate system).

***CHRG***

Particle charge.

***MASS***

Particle mass.

## Notes

Defines a point through which a particle flow or charged particle trace [**PLTRAC**] will travel. Multiple points (50 maximum) may be defined which will result in multiple flow traces. Use **TRPLIS** to list the currently defined trace points and **TRPDEL** to delete trace points.

The VX, VY, VZ, CHRG, and MASS arguments only apply to charged particles.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Plot Results>Defi Trace Pt

---

**TRTIME**, *TIME, SPACING, OFFSET, SIZE, LENGTH*

Defines the options used for the **PLTRAC** (particle flow or charged particle trace) command.

[POST1:Animation](#)

MP ME ST PR PRN <> <> FL EM <> <> PP <> EME MFS

***TIME***

Total Trace Time (seconds) (defaults to 0, which is the full flow trace).

***SPACING***

Particle spacing in seconds (defaults to 0).

***OFFSET***

Particle offset in seconds (defaults to 0). It is used internally in the **ANFLOW** macro to produce an animation of particle flow in a flowing fluid or charged particle motion in an electric or magnetic field.

***SIZE***

Particle size (defaults to 0, which is a line).

***LENGTH***

Particle length fraction (defaults to .1).

## Command Default

Full particle flow or charged particle trace.

## Notes

The **TRTIME** command varies the type of **PLTRAC** display produced. Particle flow or charged particle traces follow a particle's path in the forward and backward direction of travel. The DOF selected determines the color of the particle trace. *SPACING* defines the particle spacing in seconds from adjacent particles in the stream line. *OFFSET* defines the offset in seconds from the spacing set by the *SPACING* argument.

*LENGTH* defines the particle length fraction. The default value (.1), means the particle occupies 10% of the flow region, and the other 90% is a color-coded line.

*SIZE* sets the radius of the particle. Use *SPACING*, *OFFSET* and *LENGTH* only when *SIZE* is nonzero (i.e., the particle is bigger than the line).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>General Postproc>Plot Results>Time Interval

## TSHAP, *Shape*

Defines simple 2-D and 3-D geometric surfaces for target segment elements.

[PREP7: Elements](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

### *Shape*

Specifies the geometric shapes for target segment elements [TARGE169](#) and [TARGE170](#).

#### **LINE**

Straight line (2-D, 3-D) (Default for 2-D)

#### **PARA**

Parabola (2-D, 3-D)

#### **ARC**

Clockwise arc (2-D)

#### **CARC**

Counterclockwise arc (2-D)

#### **CIRC**

Complete circle (2-D)

#### **TRIA**

Three-node triangle (3-D) (Default for 3-D)

#### **TRI6**

Six-node triangle (3-D)

#### **QUAD**

Four-node quadrilateral (3-D)

**QUA8**

Eight-node quadrilateral (3-D)

**CYLI**

Cylinder (3-D)

**CONE**

Cone (3-D)

**SPHE**

Sphere (3-D)

**PILO**

Pilot node (2-D, 3-D)

**POINT**

Point (rigid surface node) (2-D, 3-D)

## Notes

Use this command to specify the target segment shapes for the rigid target surface associated with surface-to-surface contact ([TARGE169](#), [CONTA171](#), [CONTA172](#) (2-D) and [TARGE170](#), [CONTA173](#), [CONTA174](#) (3-D)), 3-D beam-to-beam contact ([TARGE170](#) and [CONTA176](#)), and 3-D line-to-surface contact ([TARGE170](#) and [CONTA177](#)). Once you issue **TSHAP**, all subsequent target elements generated via the direct element generation technique will have the same shape, until you issue **TSHAP** again with a different *Shape* value.

## Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

**/TSPEC**, *TCOLOR*, *TSIZE*, *TXTHIC*, *PANGLE*, *IANGLE*

Creates annotation text attributes (GUI).

[GRAPHICS: Annotation](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**TCOLOR**Text color ( $0 \leq TCOLOR \leq 15$ ):**0**

Black.

**1**

Red-Magenta.

**2**

Magenta.

**3**

Blue-Magenta.

**4**

Blue.

**5**

Cyan-Blue.

- 6**  
Cyan.
- 7**  
Green-Cyan.
- 8**  
Green.
- 9**  
Yellow-Green.
- 10**  
Yellow.
- 11**  
Orange.
- 12**  
Red.
- 13**  
Dark Gray.
- 14**  
Light Gray.
- 15**  
White.

***TSIZE***

Text size factor.

***TXTHIC***

Text thickness key:

- 1**  
normal.
- 2**  
twice as thick.
- 3**  
three times as thick.
- 4**  
four times as thick.

***PANGLE***

Text path angle ( $0.0 < angle < 360.0$ ).

***IANGLE***

Text italic angle ( $0.0 < angle < 45.0$ ).

## Notes

Defines annotation text attributes to control certain characteristics of the text created via the **/TLABEL** command. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

---

### TSRES, Array

Defines an array of key times at which the time-stepping strategy changes.

SOLUTION: Load Step Options

MP ME ST PR PRN <> <> FL EM <> DY PP <> EME MFS

#### Array

Identifies an  $N \times 1 \times 1$  array parameter containing the key times at which the heat transfer time-stepping strategy changes (the time step is reset to the initial time step based on **DELTIM** or **NSUBST** settings). The array name must be enclosed by % signs (e.g., %array%). See **\*DIM** for more information on array parameters.

## Notes

Time values in the array parameter must be in ascending order and must not exceed the time at the end of the load step as defined on the **TIME** command. The time increment between time points in the array list must be larger than the initial time step defined on the **DELTIM** or **NSUBST** command. Time values must also fall between the beginning and ending time values of the load step. For multiple load step problems, you must either change the parameter values to fall between the beginning and ending time values of the load step or reissue the command with a new array parameter. To clear the array parameter specification, issue **TSRES,ERASE**. Results can be output at the requested time points if the array or time values in the array are also specified in the **OUTRES** command using *FREQ=%array%*. Use this command to reset the time-stepping strategy within a load step. You may need to reset the time-stepping strategy when using tabular time-varying boundary conditions.

See [Steady-State Thermal Analysis](#) of the *Thermal Analysis Guide* for more information on applying boundary conditions via tabular input. See [Transient Thermal Analysis](#) of the *Thermal Analysis Guide* for more information on defining the key time array.

## Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substps

---

**TUNIF**, *TEMP*

**Assigns a uniform temperature to all nodes.**

SOLUTION: FE Body Loads

MP ME ST PR PRN DS DSS FL EM EH DY PP <> EME MFS

**TEMP**

Uniform temperature assigned to the nodes. If a *TEMP* value is not specified, the uniform temperature is set to zero.

**Command Default**

Set the uniform temperature to the reference temperature (defined via the **TREF** command).

**Notes**

**TUNIF** is a convenient form of the more general **BFUNIF** command.

In a transient or nonlinear thermal analysis, the uniform temperature is used during the first iteration of a solution as follows:

- as the starting nodal temperature (except where temperatures are explicitly specified [**D**, **DK**]),
- to evaluate temperature-dependent material properties.

In a structural analysis or an explicit dynamic analysis, the uniform temperature is used as the *default* temperature for thermal strain calculations and material property evaluation (except where body load temperatures are specified (**BF**, **BFE**, **BFK**, **LDREAD**). In other scalar field analyses, the uniform temperature is used for material property evaluation.

Because **TUNIF** (or **BFUNIF**,*TEMP*) is step-applied in the first iteration, issue a **BF**,*ALL*,*TEMP*,*Value* command to ramp on a uniform temperature load.

When the **TUNIF** command is used in an explicit dynamic analysis, you cannot apply temperature loading via the **EDLOAD**,*TEMP* command. Furthermore, temperature loading defined by **TUNIF** cannot be listed or deleted by the **EDLOAD** command.

The command default sets the uniform temperature to the reference temperature defined via the **TREF** command only (and *not* the **MP**,*REFT* command).

If using the command default to set the uniform temperature (to the reference temperature set via **TREF**), you can convert temperature-dependent secant coefficients of thermal expansion (SCTEs) from the definition temperature to the uniform temperature. To do so, issue the **MPAMOD** command.

This command is also valid in PREP7.

**Menu Paths**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>Uniform Temp**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>Uniform Temp**

**Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Temperature>Uniform Temp**

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>Uniform Temp  
 Main Menu>Preprocessor>Loads>Define Loads>Settings>Uniform Temp  
 Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Uniform Temp  
 Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>Uniform Temp  
 Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>Uniform Temp  
 Main Menu>Solution>Define Loads>Apply>Structural>Temperature>Uniform Temp  
 Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>Uniform Temp  
 Main Menu>Solution>Define Loads>Settings>Uniform Temp  
 Main Menu>Solution>Loading Options>Uniform Temp

---

## TVAR, *KEY*

Changes time to the cumulative iteration number.

[POST26: Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **KEY**

Time key:

**0**

Time is used for the variable *TIME*.

**1**

NCUMIT is used for the variable *TIME*.

## Command Default

TIME is the variable *TIME*.

## Notes

Changes the meaning of the time variable to the cumulative iteration number (NCUMIT) variable. Data can be read from the file, printed, and displayed as a function of NCUMIT rather than time. All POST26 descriptions applying to TIME then apply to NCUMIT.

## Menu Paths

Main Menu>TimeHist Postpro>Settings>Data

---

**/TXTRE**, *Lab, NUM, N1, N2, NINC*

Controls application of texture to selected items.

[GRAPHICS: Style](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **Lab**

You can apply texture according to the following labels:

#### **ELEM**

Apply texture to elements *N1* through *N2* in steps of *NINC*.



**AREA**

Apply texture to areas *N1* through *N2* in steps of *NINC*.

**VOLU**

Apply texture to volumes *N1* through *N2* in steps of *NINC*.

**CM**

Apply texture to the component named in *N1.N2* and *NINC* are ignored.

**ON, OFF**

Sets the specified texture display on or off. All other fields are ignored.

**File**

If Lab = File, the command format is **/TXTRE**, File, *Key\_Index*, *Fname*, *Fext*, --, *Format* (This variant of the command is applicable to 2-D drivers).

***Key\_Index***

The texture index associated with the file. If the number fifty-one (51) is used, the imported bitmap will be used as the window's logo.

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

***Fext***

Filename extension (8 character maximum).

--

Unused field.

***Format***

The file format. If *Format* = 0, the file is a pixmap (UNIX) or Bitmap (PC). The file cannot contain a compressed image, and the PC file must be 8 or 24 bit BI\_RGB format. If *Format* = 1 or JPEG, then the file is in JPEG (Joint Photographic Experts Group) format. If *Format* = 2 or PNG, then the file is in PNG (Portable Network Graphics) format.

**NUM**

Select the texture index number from the following list:

**0**

No Texturing

**1**

Aluminum

**2**

Aluminum, Brushed

**3**

Steel With Bumps

**4**

Steel, Embossed

**5**

Iron

**6**

Steel, Pattern

- 7** Steel, Riveted
- 8** Steel, Scratched
- 9** Tin
- 10** Metal
- 11** Steel, Etched
- 12** Metal, Hot
- 13** Iron, Grainy
- 14** Metal, Rusty
- 15** Brick
- 16** Block
- 17** Wood
- 18** Wood, Light
- 19** Wood, Walnut
- 20** Plastic, Hard Blue
- 21** Plastic, Light Blue
- 22** Plastic, Hard Red
- 31** Gold
- 32** Brass
- 33** Silver
- 34** Plastic, Black
- 35** Plastic, Ivory

- 36**  
Plastic, Blue
- 37**  
Plastic, Red
- 38**  
Plastic, Yellow
- 39**  
Plastic, Green
- 40**  
Plastic, Brown

***N1, N2, NINC***

Apply texture to *Lab* items numbered *N1* through *N2* in steps of *NINC* (defaults to 1). If *Lab* = CM, then *N1* is used to for the component name and *N2* and *NINC* are ignored. If *Lab* = ELEM, AREA, or VOLU and *N1* = blank or ALL, then the specified texture will be applied to all entities of type *Lab*. If *N1* = P, then graphical picking is enabled.

**Command Default**

No texture (/TXTRE,DEFA)

**Notes**

This command is available for 3-D Open GL devices. 2-D devices are supported **only** for the *Lab* = *File* variation of the command, allowing imported bitmaps to be used for texturing and annotation. Textures can affect the speed of many of your display operations. You can increase the speed by temporarily turning the textures off (**Utility Menu > PlotCtrls > Style > Texturing(3D) > Display Texturing**). This menu selection toggles your textures on and off. When textures are toggled off, all of the texture information is retained and reapplied when texturing is toggled back on.

For some displays, the texture will appear distorted because of a technique used to enhance 3-D displays (/DV3D,TRIS,1). Disabling this function (/DV3D,TRIS,0) will improve the quality of some texture displays. Disabling the TRIS option of the /DV3D command will slow down 3-D displays significantly. Be sure to reapply the TRIS option after you obtain a satisfactory output.

Specifying /TXTRE,DEFA removes all texturing.

**Menu Paths**

**Utility Menu > PlotCtrls > Style > Texturing(3D)**

**/TYPE**, *WN*, *Type*

**Defines the type of display.**

GRAPHICS:Style

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***WN***

Window number (or ALL) to which command applies (defaults to 1).

**Type**

Display type. Defaults to ZBUF for raster mode displays or BASIC for vector mode displays:

**BASIC or 0**

Basic display (no hidden or section operations).

**SECT or 1**

Section display (plane view). Use the **/CPLANE** command to define the cutting plane.

**HIDC or 2**

Centroid hidden display (based on item centroid sort).

**HIDD or 3**

Face hidden display (based on face centroid sort).

**HIDP or 4**

Precise hidden display (like HIDD but with more precise checking). Because all facets are sorted, this mode can be extremely slow, especially for large models.

**CAP or 5**

Capped hidden display (same as combined SECT and HIDD with model in front of section plane removed).

**ZBUF or 6**

Z-buffered display (like HIDD but using software Z-buffering).

**ZCAP or 7**

Capped Z-buffered display (same as combined SECT and ZBUF with model in front of section plane removed).

**ZQSL or 8**

ZSLICE Z-buffered display (same as SECT but the edge lines of the remaining 3-D model are shown).

**HQSL or 9**

ZSLICE precise hidden display (like ZQSL but using precise hidden).

## Command Default

ZBUF for raster mode displays; BASIC for vector mode displays.

## Notes

Defines the type of display, such as section display or hidden-line display. Use the **/DEVICE** command to specify either raster or vector mode.

The SECT, CAP, ZCAP, ZQSL, and HQSL options produce section displays. The section or "cutting" plane is specified on the **/CPLANE** command as either normal to the viewing vector at the focus point (default), or as the working plane.

When you use PowerGraphics, the section display options (Section, Slice, and Capped) use different averaging techniques for the interior and exterior results. Because of the different averaging schemes, anomalies may appear at the transition areas. In many cases, the automatically computed MIN and MAX values will differ from the full range of interior values. You can lessen the effect of these anomalies by issuing **AVRES,,FULL** (**Main Menu > General Post Proc > Options for Outp**). This command sets your legend's automatic contour interval range according to the minimum and maximum results found throughout the entire model.

With PowerGraphics active (**/GRAPHICS,POWER**), the averaging scheme for surface data with interior element data included (**AVRES,,FULL**) and multiple facets per edge (**/EFACET,2** or **/EFACET,4**) will yield differing minimum and maximum contour values depending on the Z-Buffering options (**/TYPE,,6** or **/TYPE,,7**). When the Section data is not included in the averaging schemes (**/TYPE,,7**), the resulting absolute value for the midside node is significantly smaller.

The HIDD, HIDD, HIDD, ZBUF, ZQSL, and HQLS options produce displays with "hidden" lines removed. Hidden lines are lines obscured from view by another element, area, etc. The choice of non-Z-buffered hidden-line procedure types is available only for raster mode [**/DEVICE**] displays. For vector mode displays, all non-Z-buffered "hidden-line" options use the same procedure (which is slightly different from the raster procedures). Both geometry and postprocessing displays may be of the hidden-line type. Interior stress contour lines within solid elements can also be removed as hidden lines, leaving only the stress contour lines and element outlines on the visible surfaces. Midside nodes of elements are ignored on postprocessing displays. Overlapping elements will not be displayed.

The ZBUF, ZCAP, and ZQSL options use a specific hidden-line technique called software Z-buffering. This technique allows a more accurate display of overlapping surfaces (common when using Boolean operations or **/ESHAPE** on element displays), and allows smooth shaded displays on all interactive graphics displays. Z-buffered displays can be performed faster than HIDD and CAP type displays for large models. See also the **/LIGHT**, **/SHADE**, and **/GFILE** commands for additional options when Z-buffering is used.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

### TYPE, ITYPE

Sets the element type attribute pointer.

PREP7: Meshing

PREP7: Elements

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### ITYPE

Assign this type number to the elements (defaults to 1).

## Command Default

ITYPE = 1.

## Notes

Activates an element type number to be assigned to subsequently defined elements. This number refers to the element type number (*ITYPE*) defined with the **ET** command. Type numbers may be displayed [**/PNUM**].

In some cases, ANSYS can proceed with a meshing operation even when no logical element type has been assigned via **TYPE** or **xATT,,,TYPE**. For more information, see the discussion on setting element attributes in [Meshing Your Solid Model](#) in the *Modeling and Meshing Guide*.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs**  
**Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes**  
**Main Menu>Preprocessor>Modeling>Operate>Extrude>Elem Ext Opts**

---

## TZAMESH, *Tvolu*, *SIZE*, *NDIV*

Meshes the areas of a volume to create Trefftz nodes.

[PREP7: Trefftz Domain](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

### *Tvolu*

Component name for the Trefftz volume. The component name must be enclosed in single quotes in the **TZAMESH** command line.

### *SIZE*

Edge length between Trefftz nodes on surface boundaries (i.e. lines). The areas of the component volume will be meshed with this specification to create the Trefftz nodes. If *SIZE* is zero (or blank), use *NDIV*.

### *NDIV*

Number of element divisions along the surface boundary lines. Defaults to 2.

## Notes

The command macro is used to create Trefftz nodes by meshing the surface areas of the selected volumes. The nodes of the meshed surface areas are grouped into the node component TZ\_NOD. There is no solid model associativity with the Trefftz volume and the Trefftz nodes. The Trefftz nodes (component TZ\_NOD) are used to create the Trefftz substructure (**TZEGEN** command).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Trefftz Domain>Mesh TZ Geometry**

---

## TZDELE

Deletes the Trefftz superelement, associated constraint equations and all supporting Trefftz files.

[PREP7: Trefftz Domain](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

## Notes

Deletes the Trefftz superelement, associated constraint equations and all supporting Trefftz files previously generated with the **TZEGEN** command.

You should delete the Trefftz superelement if you are going to create new Trefftz nodes and generate a new Trefftz superelement.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Trefftz Domain>Superelement>Delete TZ

---

## TZEGEN

**Generates a Trefftz domain substructure and defines a Trefftz superelement for use in electrostatic analysis.**

[PREP7: Trefftz Domain](#)

MP <> <> <> <> <> <> <> EM <> <> PP <> EME <>

## Notes

The command generates a Trefftz domain substructure from the Trefftz node component (TZ\_NOD) and the flagged infinite surfaces of the exterior finite element domain [**SF** command with the INF surface load label].

The Trefftz nodes should be uniformly scattered between the modeled components and the exterior of the finite element mesh. The nodes must not be attached to any underlying electrostatic finite elements. The nodes may be created automatically from a solid model volume using the **TZEGEN** command macro.

The substructure named `Jobname.sub` is created and automatically brought into the problem as a super-element (**MATRIX50**) using the next available element number. Also created is a set of constraint equations relating the Trefftz nodes to the surface nodes of the flagged exterior finite element domain. The **TZEGEN** command creates temporary files during the substructure generation. These include the following:

- `Jobname.TZN` - Trefftz nodes
- `Jobname.TZE` - Trefftz surface facets on the FE boundary
- `Jobname.TZX` - Surface nodes of the FE boundary
- `Jobname.TZM` - Trefftz material

The **TZEGEN** macro is only valid for 3-D electrostatics analysis. In addition, the bounding surface of the finite element domain must contain no symmetry planes.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Trefftz Domain>Superelement>Generate TZ

---





---

## U Commands

**/UCMD**, *Cmd*, *SRNUM*

**Assigns a user-defined command name.**

[APDL: Abbreviations](#)

MP ME ST <> <> <> <> <> EM EH <> PP <> EME MFS

### Argument Descriptions

#### *Cmd*

User-defined command name. Only the first four characters are significant. Must not conflict with any ANSYS command name or any user "unknown command" macro name.

#### *SRNUM*

User subroutine number (1 to 10) programmed for this command. For example, the command **/UCMD,MYCMD,3** will execute subroutine USER03 whenever the command **MYCMD** is entered. Use a blank command name to disassociate *SRNUM* from its command. For example, **/UCMD,,3** removes **MYCMD** as a command.

### Notes

Assigns a user-defined command name to a user-programmable (system-dependent) subroutine. This feature allows user-defined commands to be programmed into the ANSYS program. Once programmed, this command can be input to the program like other commands, and can also be included in the ANSYS start-up file. See **\*ULIB** for another way of defining user commands.

Up to 10 subroutines are available for user-defined commands (USER01 to USER10). Users must have system permission, system access, and knowledge to write, compile, and link the appropriate subprocessors into the ANSYS program at the site where it is to be run. All routines should be written in FORTRAN. For more information on FORTRAN compilers please refer to either the [ANSYS, Inc. Windows Installation Guide](#) or the [ANSYS, Inc. UNIX/Linux Installation Guide](#) for details specific to your platform or operating system. The USER01 routine is commented and should be listed from the distribution media (system dependent) for more details. Issue **/UCMD,STAT** to list all user-defined command names. Since a user-programmed command is a non-standard use of the program, the verification of any ANSYS run incorporating these commands is entirely up to the user. In any contact with ANSYS customer support regarding the performance of a custom version of the ANSYS program, you should explicitly state that a user programmable feature has been used. See the [Advanced Analysis Techniques Guide](#) for a general description of user-programmable features and [Guide to ANSYS User Programmable Features](#) for a detailed description of these features.

This command is valid only at the Begin Level.

### Menu Paths

**This command cannot be accessed from a menu.**

**/UDOC**, *WIND*, *Class*, *Key*,

**Determines position and content for the multi-legend options.**

GRAPHICS:Labeling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WIND**

The window number to which the command applies. (defaults to 1)

**Class**

The type (and relative importance) of legend item being displayed:

**CNTR**

Contour legend. This legend item is controlled separately from the other legend items (see note below).

**DATE**

The items in the DATE class include the date and time, or the ANSYS graphical logo (**/PLOPTS,LOGO,1**). This item is shown by default in all plots.

**GWIN**

The items in the GWIN class include the entity acronyms that appear in the legend of a multiplot of entities (Nodes, Elements, Keypoints, Lines, Areas, Volumes). GWIN items are shown by default for all **GPLOT** displays.

**TYPE**

Items in the TYPE class include the plot type (e.g. ELEMENTS, MATERIALS, NODAL SOLUTIONS, etc.). TYPE items are shown by default in all plots.

**TYP2**

Items in the TYP2 class include supplementary type information, such as DMAX and SMAX for nodal solutions. TYP2 items are shown by default in all plots.

**INUM**

Items in the INUM class include the number labels generated by the **/PNUM** command. This class is displayed by default in all plots that contain **/PNUM** information.

**BCDC**

The items in the BCDC class include labels created by the **/PBC** command. This class is shown by default in all plots which contain **/PBC** information.

**VECT**

Items in the VECT class include labels created by the **PLVECT** command. This class is shown by default for all **PLVECT** plots.

**SURF**

The items in the SURF class include labels from the **/PSF** legend. This class is shown by default on all plots of surface boundary conditions.

**BODY**

Items from the BODY class include labels from the **/PBF** legend. This class is shown by default in all plots of body forces.

**PSTA**

Items from the PSTA class include stress scaling statistics, such as the **/SSCALE** setting. This class is not shown as the default for any type of plot, and must be specifically referenced to display the included data.

**VIEW**

The items in the VIEW class include view statistics. This class is not shown as the default for any type of plot, and must be specifically referenced to display the included data.

**MISC**

The items in the MISC class include supplementary labels like /EXPANDED and Stress Section Cross Section. This class is not shown as the default for any type of plot, and must be specifically referenced to display the included data.

**KEY**

Switch:

<b>OFF or 0 --</b>	Do not display the legend.
<b>LEFT, RIGHT, TOP or BOTTOM --</b>	If the value for <i>Class</i> is CNTR, these are the four acceptable values for the contour legend position.
<b>LEFT or RIGHT --</b>	If any value other than CNTR is used for <i>Class</i> , these are the two acceptable values for the text data.

**Notes**

The legend classes conform to the controls specified in the window options panel (**PlotCtrls> Window Controls> Window Options**). In many instances, the legend controls specified with the **/PLOPTS** command will take precedence and override **/UDOC** specifications. For instance:

**/PLOPTS,LEG1,OFF** will disable the TYPE, TYP2, INUM, and MISC classes, regardless of the **/UDOC** settings.

**/PLOPTS,LEG2,OFF** will disable the VIEW class, regardless of the **/UDOC** settings.

**/PLOPTS,LEG3,OFF** will disable the PSTA class, regardless of the **/UDOC** settings.

All items in a class are listed with the same X coordinate (except for contours). The contents of the text classes are dumped onto the display window from top to bottom, in order of class importance.

The font specification for text items that are included in the user-specified legends are controlled with the **/DEVICE** command (**PlotCtrls> Font Controls> Anno/Graph Font**).

The floating point values for the data presented in the legend(s) are controlled by the **/GFORMAT** command.

**Menu Paths**

**Utility Menu>PlotCtrls>Style>MultiLegend Options>Contour Legend**  
**Utility Menu>PlotCtrls>Style>MultiLegend Options>Text Legend**

---

**/UI**, *Func*, *Type*, *Format*, *Screen*, *Color*, *Krev*, *Orient*, *Compress*, *Quality*

**Activates specified GUI dialog boxes.**

SESSION: Run Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Func**

Label identifying the dialog box to be activated:

**HELP**

Activates the online help system. Valid only in non-UI graphics mode (**/MENU**,GRPH).

**VIEW**

Activates the Pan, Zoom, Rotate dialog box

**WPSE**

Activates the Working Plane Settings dialog box.

**WPVI**

Activates the Offset Working Plane dialog box.

**RESULT**

Activates the Query Picking Menu for reviewing results.

**QUERY**

Activates the Query Picked Entities (preprocess) dialog box.

**COPY**

Activates the Hard Copy dialog box.

**ANNO**

Activates the 2D Annotation dialog box.

**AN3D**

Activates the 3D Annotation dialog box.

**SELECT**

Activates the Select Entities dialog box.

**NSEL**

Activates a picking menu to select nodes.

**ESEL**

Activates a picking menu to select elements.

**KSEL**

Activates a picking menu to select keypoints.

**LSEL**

Activates a picking menu to select lines.

**ASEL**

Activates a picking menu to select areas.

**VSEL**

Activates a picking menu to select volumes.

**REFRESH**

Refreshes the graphics window (non-UI mode only).

**COLL**

Controls the collapse of the ANSYS Main Menu when a **FINISH** command is issued. See *Type* below for a discussion of the arguments.

**Type**

Label identifying the type of select operation. Valid only for the following *Func* labels; NSEL, ESEL, KSEL, LSEL, ASEL, and VSEL:

**S**

Select a new set.

**R**

Reselect a set from the current set.

**A**

Additionally select a set and extend the current set.

**U**

Unselect a set from the current set.

Label identifying the type of results data to be queried. Valid only for *Func* = RESULT:

**NODE**

Nodal solution data (h-elements only).

**ELEMENT**

Element solution data.

Label specifying the behavior of the ANSYS Main Menu after a **FINISH** command is issued. Note that this does not affect user interaction with the Main Menu. Valid only for *Func* = COLL:

**YES, 1 or blank**

Allows the Main Menu to collapse after **FINISH** command.

**NO or 0**

Prevents Main Menu collapse after **FINISH** command.

If *Func* = **COPY**, and *Type* = **SAVE**, command format is */UI,Func,Type,Format,Screen,Color,Krev,Orient,Compress,Quality*. The remaining fields (after *Type*) identify the portion of the screen, the type of file, and the method of display that the portion of the screen is to be saved as.

**Format****PSCR**

Encapsulated Postscript File Format.

**TIFF**

Tagged Image File Format.

**EPSI**

Encapsulated Postscript with TIFF preview.

**BMP**

(PC only) Bitmap (Windows) file format.

**WMF**

(PC only) Windows Metafile format.

**EMF**

(PC only) Enhanced Metafile format.

**JPEG**

JPEG (Joint Photographic Experts Group) file format.

**Screen****FULL**

Saves the entire screen in the specified format.

**GRAPH**

Saves only the ANSYS Graphic window.

### *Color*

#### **MONO**

A two color (black and white) file is saved.

#### **GRAY**

The specified file format is saved in gray scale.

#### **COLOR**

The file is saved at the specified color depth.

### *Krev*

#### **NORM**

Saves file as shown on the screen.

#### **REVERSE**

Saves file with the background color reversed.

### *Orient*

#### **LANDSCAPE**

Saves file in landscape mode.

#### **PORTRAIT**

Saves file in portrait mode.

### *Compress*

#### **YES**

Compresses TIFF files and EPS files with TIFF preview (default).

#### **NO**

Saves files with no compression.

### *Quality*

#### **1,2,,,100**

JPEG quality index, with 100 being the maximum quality level.

## **Notes**

Allows you to activate specified GUI dialog boxes directly in either GUI or non-GUI mode.

The **/UI** command itself is valid in any processor, however certain dialog boxes are accessible only in a particular processor (e.g., **/UI,RESULT,...** is valid only in the General Postprocessor).

ANSYS JPEG software is based in part on the work of the Independent JPEG Group, Copyright 1998, Thomas G. Lane.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## **Menu Paths**

**Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Create Rezone Area**

---

**UIMP**, *MAT*, *Lab1*, *Lab2*, *Lab3*, *VAL1*, *VAL2*, *VAL3*

**Defines constant material properties (GUI).**

[PREP7:Materials](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**MAT**

Material number.

**Lab1**, **Lab2**, **Lab3**

Material property labels (see the **MP** command for valid labels).

**VAL1**, **VAL2**, **VAL3**

Values corresponding to three labels.

## Notes

Defines constant material properties. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.LOG`) if material properties are specified using the Material Properties dialog box. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models**

**Main Menu>Preprocessor>Material Props>Material Models**

**Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models**

**/UIS**, *Label*, *VALUE*

**Controls the GUI behavior.**

[SESSION:Run Controls](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Label**

Behavior control key:

**BORD**

Controls the functionality of the mouse buttons for dynamic viewing mode only. When *Label* = BORD, the three values that follow control the functionality of the LEFT, MIDDLE and RIGHT buttons, respectively (see below).

**MSGPOP**

Controls which messages from the ANSYS error message subroutine are displayed in a message dialog box.

**REPLOT**

Controls whether or not an automatic replot occurs after functions affecting the model are executed.

**ABORT**

Controls whether or not ANSYS displays dialog boxes to show the status of an operation in progress and to cancel that operation.

**DYNA**

Controls whether the dynamic mode preview is a bounding box or the edge outline of the model. This label only applies to 2-D display devices (i.e., **/SHOW,XII** or **/SHOW,WIN32**). This "model edge outline" mode is only supported in PowerGraphics [**/GRAPHICS,POWER**] and is intended for element, line, results, area, or volume displays.

**PICK**

Controls how graphical entities are highlighted from within the ANSYS Select menu.

**POWER**

Controls whether or not PowerGraphics is active when the GUI is initiated. The ANSYS program default status is PowerGraphics "ON"; this command is used (placed in the `start.ans` file) when full graphics is desired on start up.

**DPRO**

Controls whether or not the ANSYS input window displays a dynamic prompt. The dynamic prompt shows the correct command syntax for the command, as you are entering it.

**UNDO**

Controls whether or not the session editor includes nonessential commands or comments in the file it creates. You can use this option to include comments and other materials in the session editor file.

**LEGE**

Controls whether or not the multi-legend is activated when you start the GUI. The multi-legend allows you specify the location of your legend items in each of the five graphics windows. You can place this option in your `start.ans` file and have the GUI start with the legend items in a pre-specified location.

**PBAK**

Controls whether or not the background shading is activated when you start the GUI. Background shading. You can place this option in your `start.ans` file and control whether or not background shading is activated.

**ZPIC**

Controls the sorting order for entities that are coincident (directly in front of or behind each other) to a picked spot on your model. When you pick a spot on your model that could indicate two or more entities, a message warns you of this condition, and a list of the coincident entities can be generated. The *VALUE* term (below) will determine the sort order.

**HPOP**

Controls the prioritization of your GUI windows when the contents are ported to a plot or print file (**/UI,COPY,SAVE**). OpenGL (3D) graphics devices require that the ANSYS Graphics Screen contents be set in front of all overlying windows in order to port them to a printer or a file. This operation can sometimes conflict with **/NOERASE** settings. See the *VALUE* term (below) to determine the available control options.

**VALUE**

Values controlling behavior if *Label* = BORD:

(These values control the operation according to syntax : **/UIS,BORD,LEFT,MIDDLE,RIGHT**)

**1**

PAN, controls dynamic translations.

**2**

ZOOM, controls zoom, and dynamic rotation about the view vector.



- 3** ROTATE, controls dynamic rotation about the screen X and Y axes.

---

**Note**

You can designate any value for any button, or designate the same value for all three buttons. If no value is specified, default is LEFT = PAN, MIDDLE = ZOOM and RIGHT = ROTATE.

Values controlling behavior if *Label* = MSGPOP:

- 0** All messages displayed.
- 1** Only notes, warnings, and errors displayed.
- 2** Only warnings and errors displayed (default).
- 3** Only errors displayed.

Values controlling behavior if *Label* = REPLOT:

- 0** No automatic replot.
- 1** Automatic replot (default).

Values controlling behavior if *Label* = ABORT:

- ON** Display status and cancellation dialog boxes (default).
- OFF** Do not display status and cancellation dialog boxes.
- 1** Same as ON.
- 0** Same as OFF.

Values controlling behavior if *Label* = DYNA:

- 0** Use model edge outline when possible (default).
- 1** Use bounding box preview.

Values controlling behavior if *Label* = PICK:

- 0** Picked keypoints and nodes are enclosed by a square. Picked lines are overlaid by a thicker line. Picked areas, volumes, and elements (non-point/non-line) are redrawn with highlighting colors.

However, if the pick is a box, circle, or polygon pick, the highlighting for all entities consists only of a square placed around the entity's centroid.

- 1**  
Picked entities are not highlighted.
- 2**  
5.1 highlighting (that is, no XOR).
- 3**  
Picked entities are highlighted as in  $VALUE = 0$ , except that, for a box, circle, or polygon pick, the picked areas, volumes, and elements (non-point/non-line) are redrawn with highlighting colors. This technique is slower than the  $VALUE = 0$  technique.

Values controlling behavior if *Label* = POWER:

- 0**  
Start GUI in Full Graphics mode.
- 1**  
Start GUI in PowerGraphics mode (default).

Values controlling behavior if *Label* = DPRO:

- 0 or OFF**  
Do not display the dynamic prompt.
- 1 or ON**  
Display the dynamic prompt (default).

Values controlling behavior if *Label* = UNDO:

- 0 or None**  
Do not suppress any commands (default).
- 1 or Comment**  
Write the nonessential commands to the session editor file as comments (with a ! at the beginning).
- 2 or Remove**  
Do not write nonessential commands or comments.

Values controlling behavior if *Label* = LEGE:

- 0 or OFF**  
Start GUI with the enhanced legend off (default).
- 1 or ON**  
Start GUI with the enhanced legend capability activated.

Values controlling behavior if *Label* = PBAK:

- 0 or OFF**  
Start the GUI with the no background shading (default).
- 1 or ON**  
Start the GUI with background shading activated.

Values controlling behavior if *Label* = HPOP:

**0 or OFF**

No rewrite operations are performed to compensate for items that obscure or overlay the graphics window (default).

**1 or ON**

The Graphics screen contents are replotted to ensure that they are situated in front of all other windows. If **/NOERASE** is detected, this operation is suppressed.

## Notes

Controls certain features of the Graphical User Interface (GUI), including whether the ANSYS program displays dialog boxes to show you the status of an operation (such as meshing or solution) in progress and to enable you to cancel that operation. Issue **/UIS,STAT** for current status. Issue **/UIS,DEFA** to reset default values for all labels. Issue **/UIS,Label,STAT** and **/UIS,Label,DEFA** for status and to reset a specific *Label* item.

A **/UIS,HPOP,1** command employs a fast redraw method which does not allow entering the legend logic for a **/PLOPTS,INFO,1** or **/PLOPTS,INFO,2** command. However, the legend is redrawn for **/PLOPTS,INFO,3** because that command also allows a fast redraw.

This command is valid in any processor.

## Menu Paths

Utility Menu>MenuCtrls>Message Controls

Utility Menu>PlotCtrls>Device Options

---

**\*ULIB**, *Fname*, *Ext*, --

Identifies a macro library file.

[APDL: Macro Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

***Ext***

Filename extension (8 character maximum).

**--**

Unused field.

## Command Default

No macro library file.

## Notes

Identifies a macro library file for the **\*USE** command. A library of macros allows blocks of often used ANSYS commands to be stacked and executed from a single file. The macro blocks must be enclosed within block identifier and terminator lines as shown in the example below. If you want to add comment lines to a macro block, you may place them anywhere *within* the macro block. (This includes placing them directly on the lines where the macro block identifier and the macro block terminator appear, as shown in the example.) Do not place comment lines (or any other lines) outside of a macro block.

```

ABC! Any valid alphanumeric name (8 characters maximum)
! identifying this data block
---! ANSYS data input commands
---
---
/EOF! Terminator for this data block
XYZ! Identify another data block (if desired)
---! ANSYS data input commands
---
---
/EOF! Terminator for this data block
(etc.)

```

The name of the macro library file is identified for reading on the **\*ULIB** command. The name of the macro block is identified on the **\*USE** command. The commands within the macro block are copied to a temporary file (of the macro block name) during the **\*USE** operation and executed as if a macro file of that name had been created by the user. The temporary file is deleted after it has been used. Macro block names should be acceptable filenames (system dependent) and should not match user created macro file names, since the user macro file will be used first (if it exists) before the library file is searched. Macro blocks may be stacked in any order. Branching [**\*GO** or **\*IF**] external to the macro block is not allowed.

This command is valid in any processor.

## Menu Paths

Utility Menu>Macro>Execute Data Block

## UNDELETE, *Option*, *Nstart*, *Nend*

**Removes results sets from the group of sets selected for editing.**

[AUX3: Results Files](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *Option*

Specifies which sets are to be removed from the selected sets.

#### **SET**

Specifies one or more particular sets in the results file that are to be removed from the group of sets selected for deletion.

#### **ALL**

Removes all selected sets that are currently selected for deletion.

### *Nstart*

The first set to be removed from the set selected for deletion.

***Nend***

The final set to be removed from the set selected for deletion. This field is used only if operating on more than one sequential set.

**Notes**

Use this command if you have previously marked a set for deletion (with the **DELETE** command) and now wish to keep that set instead of deleting it.

**Menu Paths**

**This command cannot be accessed from a menu.**

**UNDO**, *Kywrđ*

**Allows the user to modify or save commands issued since the last RESUME or SAVE command.**

[DATABASE: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Kywrđ***NEW**

Create an editable GUI window that allows the user to alter the commands issued since the most recent **SAVE** or **RESUME** operations (GUI only).

**Notes**

The **UNDO** command brings up the session editor, a text window that displays all of the program operations since the last **SAVE** or **RESUME** command. You can modify command parameters, delete whole sections of text and even save a portion of the command string to a separate file. The file is named `jobname000.cmds`, with each subsequent save operation incrementing the filename by one digit.

**Note**

The session editor file can be changed only by the session editor. If you rename your database file outside of ANSYS and then resume that database, the session editor will display the old filename.

For more information on the session editor, see [Using the Session Editor](#) in the *Operations Guide*.

**Menu Paths**

**Main Menu>Preprocessor>Session Editor**

**/UNITS**, *Label*, *LENFACT*, *MASSFACT*, *TIMEFACT*, *TEMPFACT*, *TOFFSET*, *CHARGEFACT*, *FORCEFACT*, *HEATFACT*

**Annotates the database with the system of units used.**

[DATABASE: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Label***

Label to denote the system of units used in this job:

**USER**

User-defined system (default).

**SI**

International system (m, kg, s, K).

**MKS**

MKS system (m, kg, s, °C).

**uMKS**

μMKS system (μm, kg, s, °C).

**CGS**

CGS system (cm, g, s, °C).

**MPA**

MPA system (mm, Mg, s, °C).

**BFT**

U. S. Customary system using feet (ft, slug, s, °F).

**BIN**

U. S. Customary system using inches (in, lbf\*s<sup>2</sup>/in, s, °F).

If *Label* = USER, the remaining fields on this command may be used to enter conversion factors that are appropriate for the user-defined system of units.

***LENFACT***

Conversion factor to meter (m). Default = 1.

***MASSFACT***

Conversion factor to kilogram (kg). Default = 1.

***TIMEFACT***

Conversion factor to second (s). Default = 1.

***TEMPFACT***

Conversion factor to Kelvin (K). Default = 1.

***TOFFSET***

Temperature offset from absolute zero in degrees Kelvin. Default = 0.

***CHARGEFACT***

Conversion factor to Coulomb. Default = 1.

***FORCEFACT***

Conversion factor to Newton (N). Default = 1.

***HEATFACT***

Conversion factor to Joule (J). Default = 1.

## Command Default

User-defined units.

## Notes

Allows the user to set a marker in the database indicating the system of units used. The setting may be reviewed with the **/STATUS** command at the Begin level. The units label and conversion factors on this command are for user convenience only and have no effect on the analysis or data. That is, **/UNITS** will not convert database items from one system to another (e.g., from U. S. Customary to SI, etc.). The units setting will be written to the file of IGES data [**IGESOUT** or **CDWRITE**], which can then be read by many programs that read IGES files. The user must still use consistent units for the results to be valid.

If you choose the MKS system of units, the EPZRO option for the **EMUNIT** command is set to 8.85 e-12 F/m. (EPZRO specifies alternate free-space permittivity.)

For micro-electromechanical systems (MEMS), where dimensions are on the order of microns, see the conversion factors in [System of Units](#) in the *Coupled-Field Analysis Guide*.

If you use the ANSYS ADAMS Interface to export model information to the ADAMS program, the **/UNITS** command is required to ensure the correct transfer of data between ANSYS and ADAMS. You may choose a predefined unit system label (*Label* = SI, CGS, etc.) or you can select the user-defined system option (*Label* = USER) and input the appropriate conversion factors (*LENFACT*, *MASSFACT*, *TIMEFACT*, and *FORCEFACT*). The conversion factors will be written to the ADAMS input file *Jobname.MNF* in order to correctly generate the load. For more information, see [Export to ADAMS](#) in the *Advanced Analysis Techniques Guide*.

All differences between the base solution units used by the ANSYS and CFX solvers will be noted in the ANSYS output file. Unit conversions are automatically applied to all loads transferred unless *Label* = USER. Unit conversions are not applied to any of the loads transferred between the ANSYS and CFX solvers if they use a user-defined unit system.

This command is valid in any processor.

## Menu Paths

**This command cannot be accessed from a menu.**

---

## UNPAUSE

Restores use of a temporarily released product license.

SESSION: Run Controls

MP ME ST PR PRN DS DSS FL EM EH DY PP VT EME MFS

## Notes

The **UNPAUSE** command restores use of a temporarily released (paused) product license. The command is valid only *after* a previously issued **PAUSE** command.

When use of the product license is paused via the **PAUSE** command, no other operation (other than **SAVE** or **/EXIT**) is possible until you issue the **UNPAUSE** command.

For more information, see the documentation for the **PAUSE** command and the *ANSYS, Inc. Licensing Guide*.

## Menu Paths

**This command cannot be accessed from a menu.**

### UPCOORD, *FACTOR*, *Key*

**Modifies the coordinates of the active set of nodes, based on the current displacements.**

**SOLUTION:** Load Step Options

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

#### **FACTOR**

Scale factor for displacements being added to nodal coordinates. If *FACTOR* = 1.0, the full displacement value will be added to each node, 0.5, half the displacement value will be added, etc. If *FACTOR* = -1, the full displacement value will be subtracted from each node, etc.

#### **Key**

Key for zeroing displacements in the database:

#### **OFF**

Do not zero the displacements (default).

#### **ON**

Zero the displacements.

## Notes

The **UPCOORD** command uses displacements stored in the ANSYS database, and not those contained within the results file, *Jobname.RST*. Nodal coordinates are updated each time the command is issued. After updating, both the nodal displacements and rotations are set to zero if *Key* = ON.

For structural solutions with an updated mesh, unless the coefficient matrix is otherwise reformed (e.g., a new analysis or **NLGEOM,ON**) it should first be reformed by issuing a **KUSE,-1** command.

**UPCOORD** should not be issued between load steps in structural analysis.

For a multiphysics simulation where a CFD or electromagnetic field is being coupled to a structure undergoing large displacements, all (or a portion) of the surrounding field mesh may take part in the structural solution to "move" with the displacing structure. You can use the **UPCOORD** command with a suitable *FACTOR* to update the coordinates of the nodes using the newly computed displacements. The mesh will now conform with the displaced structure for subsequent field solutions. However, the mesh should always be restored to its original location by using an **UPCOORD,FACTOR** command before performing any subsequent structural solutions. This is true for both repeated linear solutions, and for nonlinear restarts. (All saved displacements are relative to the original mesh location.)

This command is not intended to replace either the large displacement or birth and death logic.

This command is also valid in PREP7.



## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>Updt Node Coord**  
**Main Menu>Solution>Load Step Opts>Other>Updt Node Coord**

---

**UPGEOM**, *FACTOR*, *LSTEP*, *SBSTEP*, *Fname*, *Ext*, --

**Adds displacements from a previous analysis and updates the geometry of the finite element model to the deformed configuration.**

[PREP7: Elements](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***FACTOR***

Multiplier for displacements being added to coordinates. The value 1.0 will add the full value of the displacements to the geometry of the finite element model. Defaults to 1.0.

### ***LSTEP***

Load step number of data to be imported. Defaults to the last load step.

### ***SBSTEP***

Substep number of data to be imported. Defaults to the last substep.

### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The field must be input (no default).

### ***Ext***

Filename extension (8 character maximum).

The extension must be an RST extension.

--

Unused field.

## Notes

This command updates the geometry of the finite element model according to the displacement results of the previous analysis and creates a revised geometry at the deformed configuration. This command works on all nodes (default) or on a selected set of nodes. If this command is issued repeatedly, it creates a revised geometry of the finite element model in a cumulative fashion, i.e., it adds displacement results on the previously generated deformed geometry. The solid model geometry is not updated by this command.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Update Geom**

---

\*USE

---

**\*USE**, *Name*, ARG1, ARG2, ARG3, ARG4, ARG5, ARG6, ARG7, ARG8, ARG9, AR10, AR11, AR12, AR13, AR14, AG15, AR16, AR17, AR18

**Executes a macro file.**

APDL: Macro Files

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Name*

Name (32 characters maximum, beginning with a letter) identifying the macro file or a macro block on a macro library file.

### *ARG1, ARG2, ARG3, . . . , AR18*

Values passed into the file or block where the parameters ARG1 through ARG9 and AR10 through AR18 are referenced. Values may be numbers, alphanumeric character strings (up to 32 characters enclosed in single quotes), parameters (numeric or character) or parametric expressions. See below for additional details.

## Notes

Causes execution of a macro file called *Name*, or, if not found, a macro block "*Name*" on the macro library file [**\*ULIB**]. Argument values (numeric or character) are passed into the file or block and substituted for local parameters ARG1, ARG2, ..., AR18. The file *Name* may also be executed as an "unknown command" (i.e., without the **\*USE** command name) as described below.

A macro is a sequence of ANSYS commands (as many as needed) recorded in a file or in a macro block in a library file (specified with the **\*ULIB** command). The file or block is typically executed with the **\*USE** command. In addition to command, numerical and alphanumeric data, the macro may include parameters which will be assigned numerical or alphanumeric character values when the macro is used. Use of the macro may be repeated (within a do-loop, for example) with the parameters incremented. A macro is defined within a run by "enclosing" a sequence of data input commands between a **\*CREATE** and a **\*END** command. The data input commands are passive (not executed) while being written to the macro file. The macro file (without **\*CREATE** and **\*END**) can also be created external to ANSYS.

Up to 99 specially named scalar parameters called ARG1 to AR99 are *locally* available to each macro. Note that the prefix for the first 9 parameters is "ARG," while the prefix for the last 90 is "AR." A *local* parameter is one which is not affected by, nor does it affect, other parameters, even those of the same name, which are used outside of the macro. The only way a local parameter can affect, or be affected by, parameters outside the macro is if values are passed out of, or into, the macro by an argument list. Parameters ARG1 through AR18 can have their values (numeric or character) passed via the argument list on the **\*USE** command (ARG1 through AR19 can be passed as arguments on the "unknown command" macro). Parameters AR19 through AR99 (AR20 through AR99 in the "unknown command" macro) are available solely for use within the macro; they cannot be passed via an argument list. Local parameters are available to do-loops and to **/INPUT** files processed within the macro. In addition to an ARG1--AR99 set for each macro, another ARG1--AR99 set is available external to all macros, local to "non-macro" space.

A macro is exited after its last line is executed. Macros may be nested (such as a **\*USE** or an "unknown command" within a macro). Each nested macro has its own set of 99 local parameters. Only one set of local parameters can be active at a time and that is the set corresponding to the macro currently being executed or to the set external to all macros (if any). When a nested macro completes execution, the previous set of local parameters once again becomes available. Use **\*STATUS,ARGX** to view current macro parameter values.

An alternate way of executing a macro file is via the "unknown command" route. If a command unknown to the ANSYS program is entered, a search for a file of that name (plus a `.MAC` suffix) is made. If the file exists, it is executed, if not, the "unknown command" message is output. Thus, users can write their own commands in terms of other ANSYS commands. The procedure is similar to issuing the **\*USE** command with the unknown command in the *Name* field. For example, the command **CMD,10,20,30** is internally similar to **\*USE,CMD,10,20,30**. The macro file named `CMD.MAC` will be executed with the three parameters. The **\*USE** macro description also applies to the "unknown command" macro, except that various directories are searched and a suffix (`.MAC`) is assumed. Also, a macro library file is not searched.

A three-level directory search for the "unknown command" macro file may be available (see the [Operations Guide](#)). The search order may be: 1) a high-level system directory, 2) the login directory, and 3) the local (working) directory. Use the **/PSEARCH** command to change the directory search path. For an "unknown command" **CMD**, the first file named `CMD.MAC` found to exist in the search order will be executed. The command may be input as upper or lower case, however, it is converted to upper case before the file name search occurs. On systems that uniquely support both upper and lower case file names, the file with the matching lower case name will be used if it exists, otherwise, the file with the matching upper case name will be used. All macro files placed in the `apdl` directory must be upper case.

Note, since undocumented commands exist in the ANSYS program, the user should issue the command intended for the macro file name to be sure the "unknown command" message is output in the processor where it's to be used. If the macro is to be used in other processors, the other processors must also be checked.

This command is valid in any processor.

## Menu Paths

Utility Menu>Macro>Execute Data Block

**/USER**, *WN*

Conveniently resets **/FOCUS** and **/DIST** to **USER**.

GRAPHICS: Views

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*WN*

Window number (or ALL) to which command applies (defaults to 1).

## Notes

Conveniently resets scale parameters to **USER** on the **/FOCUS** and **/DIST** commands. Scale parameters will be internally respecified to those used for the last display. Convenient when the last scale parameters were automatically calculated. User specified parameters hold until changed or removed [**/AUTO**]. Parameters may be reset on the individual commands after this command has been issued.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>View Settings>Automatic Fit Mode

**USRCAL**, *Rnam1*, *Rnam2*, *Rnam3*, *Rnam4*, *Rnam5*, *Rnam6*, *Rnam7*, *Rnam8*, *Rnam9***Allows user-solution subroutines to be activated or deactivated.**

SOLUTION: Load Step Options

MP ME ST &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; &lt;&gt; PP &lt;&gt; EME MFS

***Rnam1*, *Rnam2*, *Rnam3*, . . . , *Rnam9***

User-defined solution subroutine names to be activated. Up to nine may be defined on one command or multiple commands may be used. If *Rnam1* = ALL, activate all valid user subroutines. If *Rnam1* = NONE, deactivate all valid user subroutines. All characters are required:

**USREFL**

Allows user defined scalar field (body force) loads.

**USERCV**

Allows user defined convection (surface) loads.

**USERPR**

Allows user defined pressure (surface) loads.

**USERFX**

Allows user-defined heat flux (surface) loads.

**USERCH**

Allows user-defined charge density (surface) loads.

**USERFD**

Computes the complex load vector for the frequency domain logic.

**USEROU**

Allows user supplied element output.

**USERMC**

Allows user control of the hygrothermal growth).

**USOLBEG**

Allows user access before each solution.

**ULDBEG**

Allows user access before each load step.

**USSBEG**

Allows user access before each substep.

**UITBEG**

Allows user access before each equilibrium iteration.

**UITFIN**

Allows user access after each equilibrium iteration.

**USSFIN**

Allows user access after each substep.

**ULDFIN**

Allows user access after each load step.

**USOLFIN**

Allows user access after each solution.

**UANBEG**

Allows user access at start of ANSYS run.

**UANFIN**

Allows user access at end of ANSYS run.

**UELMATX**

Allows user access to element matrices and load vectors.

**USWVVEAC**

Allows user access to ocean loading.

## Command Default

No user-solution subroutines are active (even if linked into the program).

## Notes

Allows certain user-solution subroutines to be activated or deactivated (system-dependent). This command only affects the subroutines named. Other user subroutines (such as user elements, user creep, etc.) have their own activation controls described with the feature.

The routines are commented and should be listed after performing a custom installation from the distribution media for more details. See also the [Advanced Analysis Techniques Guide](#) for a general description of user-programmable features.

Users must have system permission, system access, and knowledge to write, compile, and link the appropriate subroutines into the ANSYS program at the site where it is to be run. All routines should be written in FORTRAN. (For more information on FORTRAN compilers please refer to either the [ANSYS, Inc. Windows Installation Guide](#) or the [ANSYS, Inc. UNIX/Linux Installation Guide](#) for details specific to your platform or operating system.) Issue **USRCAL,STAT** to list the status of these user subroutines. Since a user-programmed subroutine is a nonstandard use of the program, the verification of any ANSYS run incorporating these commands is entirely up to the user. In any contact with ANSYS customer support regarding the performance of a custom version of the ANSYS program, you should explicitly state that a user programmable feature has been used.

This command is also valid in PREP7.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Other>User Routines**

**Main Menu>Solution>Load Step Opts>Other>User Routines**

---

**USRDOF**, *Action, DOF1, DOF2, DOF3, DOF4, DOF5, DOF6, DOF7, DOF8, DOF9, DOF10*

**Specifies the degrees of freedom for the user-defined element **USER300**.**

[PREP7: Elements](#)

MP ME ST <> <> <> <> FL EM EH <> PP <> EME MFS

### Action

One of the following command operations:

**DEFINE**

Specify the degrees of freedom (DOFs). This value is the default.

**LIST**

List all previously specified DOFs.

**DELETE**

Delete all previously specified DOFs.

*DOF1, DOF2, DOF3, . . . , DOF10*

The list of DOFs.

## Notes

The **USRDOF** command specifies the degrees of freedom for the user-defined element **USER300**.

Although you can intersperse other commands as necessary for your analysis, issue the **USRDOF** command as part of the following *general sequence* of commands:

1. Issue the **ET** command for element **USER300**, followed by the related **TYPE** command.
2. Issue both the **USRELEM** and **USRDOF** commands (in either order).
3. Define your element using **USER300**.

The DOF list (*DOF1* through *DOF10*) can consist of up to 10 DOFs. Use any valid and appropriate DOF (such as UX, UY, UZ, ROTX, ROTY, ROTZ, AX, AY, AZ, VX, VY, VZ, PRES, WARP, TEMP, VOLT, MAG, ENKE, ENDS, EMF, and CURR).

You can specify a maximum of 10 DOFs per **USRDOF** command. To define additional DOFs, issue the command again.

The maximum number of DOFs for a user-defined element--the number of nodes times the number of DOFs per node--cannot exceed 480.

To learn more about user-defined elements, see [Creating a New Element](#) in the *Programmer's Manual*.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**USRELEM**, *NNODES*, *NDIM*, *KeyShape*, *NREAL*, *NSAVEVARS*, *NRSLTVAR*, *KEYANSMAT*, *NINTPNTS*, *KESTRESS*, *KEYSYM*

**Specifies the characteristics of the user-defined element **USER300**.**

[PREP7: Elements](#)

MP ME ST <> <> <> <> FL EM EH <> PP <> EME MFS

***NNODES***

The number of nodes.

***NDIM***

The number of dimensions (of nodal coordinates). Valid values are 2 or 3.

**KeyShape**

One of the following element shape options:

**ANYSHAPE**

Any shape (that is, no specified shape). This value is the default. (The [ANSYS MeshTool](#) is unavailable.)

**POINT**

Point.

**LINE**

Straight line.

**TRIAN**

Triangle.

**QUAD**

Quadrilateral. This shape can be degenerated to a triangle.

**TET**

Tetrahedron.

**BRICK**

Brick. This shape can be degenerated to a wedge, pyramid, or tetrahedron.

**NREAL**

The number of real constants.

**NSAVEVARS**

The number of saved variables.

**NRSLTVAR**

The number of variables saved in results files.

**KEYANSMAT**

Key for element formulation control:

**0**

Create your own material codes within the element formulation. In this case, the real constants are available to input material properties. You can also input linear material properties via **MP** and **MP-DATA** commands .

**1**

Use ANSYS standard material routines or the `USERMAT` subroutine to form structural material data. ANSYS material properties must be input in the standard way (as you would for non-user-defined elements). This value is invalid when *KeyShape* = ANYSHAPE.

**NINTPNTS**

The maximum number of integration points (used when *KEYANSMAT* = 1).

**KESTRESS**

Key for the element stress state (used when *KEYANSMAT* = 1):

**0**

Plane stress elements.

**1**

Axisymmetric elements.

**2**

Plane strain elements.

- 3** 3-D solid elements.
- 4** 3-D solid-shell elements.
- 5** Generalized plane strain elements.
- 6** Beam elements.
- 7** Link/truss elements.
- 8** 3-D shell elements.
- 9** Axisymmetric shell elements.

**KEYSYM**

Key for specifying whether element stiffness matrices are symmetric or unsymmetric:

- 0** Symmetric.
- 1** Unsymmetric.

**Notes**

The **USRELEM** command specifies the characteristics of the user-defined element [USER300](#).

Although you can intersperse other commands as necessary for your analysis, issue the **USRELEM** command as part of the following *general sequence* of commands:

1. Issue the **ET** command for element [USER300](#), followed by the related **TYPE** command.
2. Issue both the **USRELEM** and **USRDOF** commands (in either order).
3. Define your element using [USER300](#).

The number of real constants (*NREAL*) can refer to geometry quantities, material quantities, or any parameters for element formulation.

ANSYS saves variables in the `.esav` file to preserve element data when you specify a positive *NSAVEVARS* value. When *KEYANSMAT* = 0, all variables of both material and kinematic formulation are saved. When *KEYANSMAT* = 1, only the variables for kinematic formulation (such as deformation gradient tensor) are saved; in this case, the material routine saves all necessary material data automatically.

Element data saved in results files (*NRSLTVAR*) are accessible only as nonsummable miscellaneous data. ANSYS saves stress and total strain data for structural elements in the `.rst` file automatically (as it does for equivalent variables such as thermal gradient and thermal flux in thermal elements); therefore, *NRSLTVAR* does not need to include stress and total strain data.

To learn more about creating user-defined elements, see [Creating a New Element](#) in the *Programmer's Manual*.



---

## Menu Paths

**This command cannot be accessed from a menu.**

---



---

## V Commands

**V**, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *P7*, *P8*

**Defines a volume through keypoints.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**P1**

Keypoint defining starting corner of volume. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

**P2**

Keypoint defining second corner of volume.

**P3**

Keypoint defining third corner of volume.

**P4**

Keypoint defining fourth corner of volume.

**P5**

Keypoint defining fifth corner of volume.

**P6**

Keypoint defining sixth corner of volume.

**P7**

Keypoint defining seventh corner of volume.

**P8**

Keypoint defining eighth corner of volume.

### Notes

Defines a volume (and its corresponding lines and areas) through eight (or fewer) existing keypoints. Keypoints must be input in a continuous order. The order of the keypoints should be around the bottom and then the top. Missing lines are generated "straight" in the active coordinate system and assigned the lowest available numbers [**NUMSTR**]. Missing areas are generated and assigned the lowest available numbers.

Solid modeling in a toroidal coordinate system is not recommended.

Certain faces may be condensed to a line or point by repeating keypoints. For example, use **V**,*P1*,*P2*,*P3*,*P3*,*P5*,*P6*,*P7*,*P7* for a triangular prism or **V**,*P1*,*P2*,*P3*,*P3*,*P5*,*P5*,*P5*,*P5* for a tetrahedron.

Using keypoints to produce partial sections in **CSYS** = 2 can generate anomalies; check the resulting volumes carefully.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Volumes>Arbitrary>Through KPs**

---

**V2DOPT**, *GEOM*, *NDIV*, *HIDOPT*, *NZONE*

**Specifies 2-D/axisymmetric view factor calculation options.**

SOLUTION: Radiosity  
AUX12: Radiosity Solver

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

### **GEOM**

Choice of geometry:

- 0**  
Planar (default).
- 1**  
Axisymmetric

### **NDIV**

Number of divisions for axisymmetric geometry. Defaults to 20.

### **HIDOPT**

Viewing option:

- 0**  
Hidden (default).
- 1**  
Non-hidden

### **NZONE**

Number of zones for view factor calculation. Defaults to 200.

## Command Default

**V2DOPT** allows you to select options for 2-D view factor calculation. The geometry type can be set to either 2-D plane or axisymmetric (defaults to plane). You can also define the number of divisions (defaults to 20) for an axisymmetric geometry. This command also allows you to select either hidden or non-hidden viewing option (defaults to hidden) and the number of zones for view factor calculation (defaults to 200).

## Menu Paths

**Main Menu>Preprocessor>Radiation Opts>View Factor**  
**Main Menu>Radiation Opt>Radiosity Meth>View Factor**  
**Main Menu>Solution>Radiation Opts>View Factor**

---

**VA**, A1, A2, A3, A4, A5, A6, A7, A8, A9, A10

**Generates a volume bounded by existing areas.**

PREP7: Volumes

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**A1, A2, A3, . . . , A10**

List of areas defining volume. The minimum number of areas is 4. If A1 = ALL, use all selected [ASEL] areas and ignore A2 to A10. If A1 = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for A1.

## Notes

This command conveniently allows generating volumes from regions having more than eight keypoints (which is not allowed with the **V** command). Areas may be input in any order. The exterior surface of a **VA** volume must be continuous, but holes may pass completely through it.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Volume by Areas**

**Main Menu>Preprocessor>Modeling>Create>Volumes>Arbitrary>By Areas**

**\*VABS**, KABSR, KAB1, KAB2, KAB3

**Applies the absolute value function to array parameters.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

**KABSR**

Absolute value of results parameter:

**0** --

Do not take absolute value of results parameter (ParR).

**1** --

Take absolute value.

**KAB1**

Absolute value of first parameter:

**0** --

Do not take absolute value of first parameter (Par1 or ParI).

**1** --

Take absolute value.

**KAB2**

Absolute value of second parameter:

**0** --

Do not take absolute value of second parameter (Par2 or ParJ).

**1** --  
Take absolute value.

**KABS3**

Absolute value of third parameter:

**0** --  
Do not take absolute value of third parameter (Par3 or ParK).

**1** --  
Take absolute value.

## Command Default

Do not use absolute values.

## Notes

Applies an absolute value to parameters used in certain **\*Vxx** and **\*Mxx** operations. Typical absolute value applications are of the form:

$$\text{ParR} = |f(|\text{Par1}|)|$$

or

$$\text{ParR} = (| |\text{Par1}| \text{ o } |\text{Par2}| )|$$

The absolute values are applied to each input parameter value before the operation and to the result value after the operation. Absolute values are applied before the scale factors so that negative scale factors may be used. The absolute value settings are *reset* to the default (no absolute value) after each **\*Vxx** or **\*Mxx** operation. Use **\*VSTAT** to list settings.

This command is valid in any processor.

## Menu Paths

**Utility Menu>Parameters>Array Operations>Operation Settings**

---

**VADD**, NV1, NV2, NV3, NV4, NV5, NV6, NV7, NV8, NV9

**Adds separate volumes to create a single volume.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NV1, NV2, NV3, . . . , NV9**

Numbers of volumes to be added. If NV1 = ALL, add all selected volumes and ignore NV2 to NV9. If NV1 = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

A component name may also be substituted for NV1.

## Notes

Adds separate volumes to create a single volume. The original volumes (and their corresponding areas, lines and keypoints) will be deleted by default [**BOPTN**]. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. Concatenated entities are not valid with this command.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Add>Volumes**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Add>Volumes**

## VARDEL, *NVAR*

Deletes a variable (GUI).

POST26: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *NVAR*

The reference number of the variable to be deleted. *NVAR* is as defined by **NSOL**, **ESOL**, etc.

## Notes

Deletes a POST26 solution results variable. This is a command generated by the Graphical User Interface (GUI). It will appear in the log file (`Jobname.LOG`) if a POST26 variable is deleted from the "Defined Time-History Variables" dialog box. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

## Menu Paths

**Main Menu>TimeHist Postpro>Define Variables**

**Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables**

## VARNAM, *IR*, *Name*

Names (or renames) a variable.

POST26: Set Up

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *IR*

Reference number of the variable (2 to NV [**NUMVAR**]).

### *Name*

Thirty-two character name for identifying variable on printouts and displays. Embedded blanks are compressed for output.

## Menu Paths

**Main Menu>TimeHist Postpro>Settings>Graph**

**Main Menu>TimeHist Postpro>Settings>List****VATT**, *MAT*, *REAL*, *TYPE*, *ESYS*, *SECNUM***Associates element attributes with the selected, unmeshed volumes.**

PREP7: Meshing

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

*MAT*, *REAL*, *TYPE*, *ESYS*, *SECNUM*

Material number, real constant set number, type number, coordinate system number, and section number to be associated with selected, unmeshed volumes.

**Notes**

These element attributes will be used when the volumes are meshed. If a volume does not have attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT**, **REAL**, **TYPE**, **ESYS**, and **SECNUM** command settings. Reissue the **VATT** command (before volumes are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association.

If any of the arguments *MAT*, *REAL*, *TYPE*, *ESYS* or *SECNUM* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, ANSYS can proceed with a volume meshing operation even when no logical element type has been assigned via **VATT**, *TYPE* or **TYPE**. For more information, see the discussion on setting element attributes in [Meshing Your Solid Model](#) of the *Modeling and Meshing Guide*.

**Menu Paths****Main Menu>Preprocessor>Meshing>Mesh Attributes>All Volumes****Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked Volumes****VCLEAR**, *NV1*, *NV2*, *NINC***Deletes nodes and volume elements associated with selected volumes.**

PREP7: Meshing

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

*NV1*, *NV2*, *NINC*

Delete mesh for volumes *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and mesh for all selected volumes [**VSEL**] is deleted. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

**Notes**

Deletes *all* nodes and volume elements associated with selected volumes (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed volumes and nodes associated with non-volume elements will not be deleted. Attributes assigned as a result of **VATT** are maintained. In the program's response



to the command, if a volume, area, line, or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

## Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Volumes

---

**\*VCOL**, *NCOL1*, *NCOL2*

Specifies the number of columns in matrix operations.

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *NCOL1*

Number of columns to be used for Par1 with **\*Mxx** operations. Defaults to whatever is needed to fill the result array.

### *NCOL2*

Number of columns to be used for Par2 with **\*Mxx** operations. Defaults to whatever is needed to fill the result array.

## Command Default

Fill all locations of the result array from the specified starting location.

## Notes

Specifies the number of columns to be used in array parameter matrix operations. The size of the submatrix used is determined from the upper left starting array element (defined on the operation command) to the lower right array element (defined by the number of columns on this command and the number of rows on the **\*VLEN** command).

The default *NCOL* is calculated from the maximum number of columns of the result array (the **\*DIM** column dimension) minus the starting location + 1. For example, **\*DIM,R,,1,10** and a starting location of R(1,7) gives a default of 4 columns ( starting with R(1,7), R(1,8), R(1,9), and R(1,10)). Repeat operations automatically terminate at the last column of the result array. Existing values in the rows and columns of the results matrix remain unchanged where not overwritten by the requested input or operation values.

The column control settings are *reset* to the defaults after each **\*Mxx** operation. Use **\*VSTAT** to list settings.

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Array Operations>Operation Settings

---

**/VCONE**, *WN*, *PHI*

**Defines the view cone angle for perspective displays.**

GRAPHICS:Views

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WN**

Window number (or ALL) to which command applies (defaults to 1).

**PHI**

View cone angle (0.0 to 85.°) to define perspective. Use *PHI* = 45.0° for typical perspective. Increase angle for more perspective, decrease angle for less. If the distance [**/DIST**] is not specified, it will be automatically calculated to give full window magnification. If the distance is also specified, *PHI* controls both the perspective and the magnification. The larger the angle, the more the perspective and the less the magnification. Defaults to 0.0 (no perspective).

**Command Default**

0.0 degrees (no perspective (parallel projection)).

**Notes**

Perspective shows the true depth of the object in the display. A variable magnification results since the back plane of the object is further from the observer than the front plane. The largest magnification occurs at the front plane. With perspective, the magnification factor (MAGF) is not only a function of the distance from the object, but also the window shape and the perspective (or view cone) angle  $\Phi$  as follows:

$$\text{MAGF} = \frac{\ell/2}{(d) \text{TAN}\phi}$$

where  $\ell$ , for square windows, is the largest in-plane vertical or horizontal dimension,  $d$  is the distance from the observer to the plane of  $\ell$  (usually the front plane of the object), and  $\Phi$  is the view cone angle (defined with the **/VCONE** command). The bigger the cone angle, the more the perspective. The magnification factor proportionally decreases with increasing  $\Phi$ . The distance can be defined with the **/DIST** or the **/FOCUS** command. Note, the distance input on the **/DIST** command is equal to  $d$  only if the focus point is located on the plane of  $\ell$ . It is recommended that if a general perspective is desired (i.e., not any specific cone angle), use  $\Phi = 45.0$  (since  $\text{TAN}(45.0) = 1.0$ ) and let the  $d$  value be automatically calculated for full window magnification.

Note that any number of **/DIST**, **/FOCUS**, and **/VCONE** combinations can be used to produce the same magnification. Distances less than the object depth will produce views from within the object.

A magnification factor of 1.0 just fills the window. If the automatic scaling option is used [**/AUTO**], the magnification factor is fixed at 0.91 (to allow a 10% margin around the object) and  $d$  is automatically calculated for the given **/VCONE** and **/FOCUS** values. Any value of  $\Phi$  between 0.0 and 85.0 (usually 45.0) may be used to activate the perspective. Views from inside the object are not possible when  $d$  is automatically calculated (use manual scaling [**/USER**] along with **/DIST** specification).

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>View Settings>Perspective View

**VCROSS**, *LabXR*, *LabYR*, *LabZR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

Forms element table items from the cross product of two vectors.

[POST1: Element Table](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*LabXR*, *LabYR*, *LabZR*

Label assigned to X, Y, and Z-component of resultant vector.

*LabX1*, *LabY1*, *LabZ1*

X, Y, and Z-component of first vector label.

*LabX2*, *LabY2*, *LabZ2*

X, Y, and Z-component of second vector label.

## Notes

Forms labeled result items for the selected element from the cross product of two vectors:

$$\{LabXR, LabYR, LabZR\} = \{LabX1, LabY1, LabZ1\} \times \{LabX2, LabY2, LabZ2\}$$

Data must be in a consistent coordinate system. Labels are those associated with the **ETABLE** command.

## Menu Paths

Main Menu>General Postproc>Element Table>Cross Product

\*VCUM, *KEY*

Allows array parameter results to add to existing results.

[APDL: Array Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

**KEY**

Accumulation key:

**0** --

Overwrite results.

**1** --

Add results to the current value of the results parameter.

## Command Default

Overwrite results.

## Notes

Allows results from certain **\*Vxx** and **\*Mxx** operations to overwrite or add to existing results. The cumulative operation is of the form:

$$\text{ParR} = \text{ParR} + \text{ParR}(\text{Previous})$$

The cumulative setting is *reset* to the default (overwrite) after each **\*Vxx** or **\*Mxx** operation. Use **\*VSTAT** to list settings.

This command is valid in any processor.

## Menu Paths

**Utility Menu>Parameters>Array Operations>Operation Settings**

---

### **VDDAM**, *VF, VA, VB, VC*

**Specifies the velocity spectrum computation constants for the analysis of shock resistance of shipboard structures.**

*SOLUTION: Spectrum Options*

MP ME ST <> <> <> <> <> <> <> <> PP <> EME MFS

**VF**  
Direction-dependent velocity coefficient for elastic or elastic-plastic analysis option (Default = 0).

**VA, VB, VC**  
Coefficients for the DDAM velocity spectrum equations. See the *Theory Reference for the Mechanical APDL and Mechanical Applications*. Default for these coefficients is zero.

## Notes

This command specifies velocity coefficients to analyze shock resistance of shipboard equipment. These coefficients are used to compute mode coefficients according to the equations given in the *Theory Reference for the Mechanical APDL and Mechanical Applications*. The form of these equations is based on the Naval NRL Dynamic Design Analysis Method. This command, along with the **ADDAM** and **SED** commands, is used with the spectrum (**ANTYPE,SPECTR**) analysis as a special purpose alternative to the **SV, FREQ**, and **SVTYP** commands. The mass and length units of the model must be in pounds and inches, respectively.

This command is also valid in PREP7.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>DDAM Options**

**Main Menu>Solution>Load Step Opts>Spectrum>DDAM Options**

---

**VDELE**, *NV1*, *NV2*, *NINC*, *KSWP*

**Deletes unmeshed volumes.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NV1*, *NV2*, *NINC***

Delete volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and all selected volumes [**VSEL**] are deleted. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

***KSWP***

Specifies whether keypoints, lines, and areas are also deleted:

**0**

Delete volumes only (default).

**1**

Delete volumes, as well as keypoints, lines, and areas attached to the specified volumes but not shared by other volumes.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Delete>Volume and Below**

**Main Menu>Preprocessor>Modeling>Delete>Volumes Only**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Delete>Volume and Below**

**Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Delete>Volumes Only**

**VDGL**, *NV1*, *NV2*, *NINC*

**Lists keypoints of a volume that lie on a parametric degeneracy.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NV1*, *NV2*, *NINC***

List keypoints that lie on a parametric degeneracy on volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL (default), *NV2* and *NINC* will be ignored and keypoints on all selected volumes [**VSEL**] will be listed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). You may also substitute a component name for *NV1* (ignore *NV2* and *NINC*).

**Notes**

See the [Modeling and Meshing Guide](#) for details about parametric degeneracies.

This command is valid in any processor.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>List Degen Volus**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>List Degen Volus**

---

**VDOT**, *LabR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

**Forms an element table item from the dot product of two vectors.**

[POST1: Element Table](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**LabR**

Label assigned to dot product result.

**LabX1, LabY1, LabZ1**

X, Y, and Z-component of first vector label.

**LabX2, LabY2, LabZ2**

X, Y, and Z-component of second vector label.

## Notes

Forms labeled result items for the selected element from the dot product of two vectors:

$$LabR = \{LabX1, LabY1, LabZ1\} \bullet \{LabX2, LabY2, LabZ2\}$$

Data must be in a consistent coordinate system. Labels are those associated with the **ETABLE** command.

## Menu Paths

**Main Menu>General Postproc>Element Table>Dot Product**

---

**VDRAG**, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NLP1*, *NLP2*, *NLP3*, *NLP4*, *NLP5*, *NLP6*

**Generates volumes by dragging an area pattern along a path.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NA1, NA2, NA3, . . . , NA6**

List of areas in the pattern to be dragged (6 maximum if using keyboard entry). If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NA1* = ALL, all selected areas will be swept along the path. A component name may also be substituted for *NA1*.

**NLP1, NLP2, NLP3, . . . , NLP6**

List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines. To be continuous, adjacent lines must share the connecting keypoint (the end keypoint of one line must also be first keypoint of the next line).

## Notes

Generates volumes (and their corresponding keypoints, lines, and areas) by sweeping a given area pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which the path lines are input (*NLP1*, *NLP2*, etc.). If the drag path is a single line (*NLP1*),

the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given area pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves. Lines are generated with the same shapes as the given pattern and the path lines.

Keypoint, line, area, and volume numbers are automatically assigned (beginning with the lowest available values [NUMSTR]). Adjacent lines use a common keypoint, adjacent areas use a common line, and adjacent volumes use a common area. For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

If element attributes have been associated with the input area via the **AATT** command, the opposite area generated by the **VDRAG** operation will also have those attributes (i.e., the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the input areas are meshed or belong to a meshed volume, the area(s) can be extruded to a 3-D mesh. Note that the *NDIV* argument of the **ESIZE** command should be set before extruding the meshed areas. Alternatively, mesh divisions can be specified directly on the drag line(s) (**LESIZE**). See the *Modeling and Meshing Guide* for more information.

You can use the **VDRAG** command to generate 3-D interface element meshes for elements **INTER194** and **INTER195**. When generating interface element meshes using **VDRAG**, you must specify the line divisions to generate one interface element directly on the drag line using the **LESIZE** command. The source area to be extruded becomes the bottom surface of the interface element. Interface elements must be extruded in what will become the element's local x direction, that is, bottom to top.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>Along Lines**

**\*VEC**, *Vector, Type, Method, Val1, Val2, Val3, Val4*

**Creates a vector.**

APDL: Matrix Operations

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *Vector*

Name used to identify the vector. Must be specified.

### *Type*

Vector type:

**D --**

Double precision real values (default).

**Z --**

Complex double precision values.

**I --**

Integer values.

**Method**

Method used to create the vector:

**ALLOC --**

Allocate space for a vector (default).

**COPY --**

Copy an existing vector.

**IMPORT --**

Import the vector from a file.

**LINK --**

Link to a column of an existing dense **\*DMAT** matrix and use it in subsequent vector calculations. Any changes to the vector are also made to the corresponding matrix column (memory is shared).

**Val1, Val2, Val3, Val4**

Additional input. The meaning of *Val1* through *Val4* will vary depending on the specified *Method*. See details below.

**The following Valx field is used with Method = ALLOC:****Val1**

Number of rows in the vector.

**The following Valx field is used with Method = COPY:****Val1**

Name of the vector to copy.

**The following table describes the Valx fields used with Method = IMPORT.**

<i>Method = IMPORT</i>				
<i>Val1</i>	<i>Val2</i>	<i>Val3</i>	<i>Val4</i>	Description
FULL	File name	RHS - Load vector GVEC - Constraint equation constant terms BACK - nodal mapping vector (internal to user) FORWARD - nodal mapping vector (user to internal)	(not used)	Import a load vector or nodal mapping vector from an existing FULL file.
MODE	File name	Mode number	(not used)	Import a mode from an existing MODE file.
RST	File name	Data set number	(not used)	Import degree of freedom results from an existing RST file.
SUB	File name	RHS - Load vector	(not used)	Import a load vector from an existing SUB file.



EMAT	File name	RHS - Load vector	Element number	Import an element load vector from an existing EMAT file.
MAT	File name	(not used)	(not used)	Restore from a previous <b>*EXPORT</b> ( <i>FORMAT</i> = MAT) command.
AP-DL	Array parameter name	(not used)	(not used)	Import an existing array parameter.
HB-MAT	File name	File format: ASCII BINARY	(not used)	Import a load vector from an existing Harwell-Boeing format file.

The following *Valx* fields are used with *Method* = LINK:

**Val1**

Name of the **\*DMAT** matrix.

**Val2**

Column number of the matrix to link to.

## Notes

Use the **\*DMAT** command to create a matrix.

For more information on the BACK and FORWARD nodal mapping vectors, see [Degree of Freedom Ordering](#) in the *ANSYS Parametric Design Language Guide*.

## Menu Paths

**This command cannot be accessed from a menu.**

### **\*VEDIT**, *Par*

**Allows numerical array parameters to be graphically edited.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

**Par**

Name of the array parameter to be edited.

## Notes

Invokes a graphical editing system that displays array parameter values in matrix form, and allows the use of the mouse to edit individual values. The starting array subscripts must be defined, such as **\*VEDIT**,A(4,6,1), to indicate the section of the array to be edited. The array section starts at the specified array element and continues to the maximum extent of the array parameter. Row and column index values may be set or changed in any plane, and those values will be applied to all planes. The menu system must be on **[MENU]**

when this command is issued. Graphical editing is not available for character array parameters. The **\*VEDIT** command can not be used in a macro or other secondary input file.

This command is not applicable to 4- or 5-D arrays.

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Array Parameters>Define/Edit

**VEORIENT**, *VNUM*, *Option*, *VALUE1*, *VALUE2*

**Specifies brick element orientation for volume mapped (hexahedron) meshing.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **VNUM**

Number identifying volume for which elements are to be oriented (no default).

### **Option**

Option for defining element orientation:

#### **KP**

Orientation is determined by two keypoints on the volume. Input the keypoint numbers (KZ1 and KZ2) in fields *VALUE1* and *VALUE2*, respectively. The element z-axis points from KZ1 toward KZ2. Element x and y directions point away from KZ1 along edges of the volume to make a right-hand triad. (The element x- and y-axes are uniquely determined by this specification.)

#### **LINE**

Orientation is determined by one of the lines defining the volume. Input the line number in field *VALUE1*. The element z direction follows the direction of the line. Input a negative value if the desired z direction is opposite to the direction of the specified line. (The element x- and y-axes are uniquely determined by this specification.) (*VALUE2* is not used.)

#### **AREA**

Orientation is determined by one of the areas defining the volume. The area represents the desired element top surface. Input the area number as *VALUE1*. The shortest line in the volume connected to the area will be used to specify the element z direction. (If more than one shortest line exists, the lowest numbered of those is used.) Element x and y directions are not uniquely specified by this option. (*VALUE2* is not used.)

#### **THIN**

Align the element z normal to the thinnest dimension of the volume. The shortest line in the volume is used to specify the element z direction. (If more than one shortest line exists, the lowest numbered of those is used.) Element x and y directions are not uniquely specified by this option. (*VALUE1* and *VALUE2* are not used.)

#### **DELE**

Delete the previously defined volume orientation for the specified volume (*VNUM*). (*VALUE1* and *VALUE2* are not used.)

**VALUE1, VALUE2**

Parameters required for the element z-axis direction specification. The meaning of *VALUE1* and *VALUE2* will depend on the chosen *Option*. See the description of *Option* above for details.

**Command Default**

Elements are not oriented in any specific manner.

**Notes**

Use **VEORIENT** before the **VMESH** command to specify the desired orientation of brick elements in a mapped mesh. **VEORIENT** has no effect on tetrahedron meshes, extruded meshes (**VROTAT**, **VDRAG**, **VEXT**, etc.), or swept meshes (**VSWEEP**).

Proper brick orientation is essential for certain element types such as **SOLID185** Layered Solid, **SOLID186** Layered Solid, and **SOLSH190**. In such cases, use **VEORIENT** or **EORIENT** to achieve the desired orientation. For other brick element types, you may need to specify element orientation to control orthotropic material property directions without concern for the element connectivity. For those cases, the **ESYS** command is the preferred method of specifying the material property directions.

For *Option* = LINE, AREA, and THIN, the orientation will be internally converted to an equivalent *Option* = KP specification (KP,KZ1,KZ2). Use the **VLIST** command to view the element orientations (in terms of KZ1 and KZ2) associated with each volume.

**Menu Paths**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Delete Specification**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Z Along Line**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Z by 2 Keypoints**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Z in Thin Direction**

**Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Z Normal to Area**

**VEXT, NA1, NA2, NINC, DX, DY, DZ, RX, RY, RZ**

**Generates additional volumes by extruding areas.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**NA1, NA2, NINC**

Set of areas (*NA1* to *NA2* in steps of *NINC*) that defines the pattern to be extruded. *NA2* defaults to *NA1*, *NINC* defaults to 1. If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is defined by all selected areas. If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

**DX, DY, DZ**

Increments to be applied to the X, Y, and Z keypoint coordinates in the active coordinate system (*DR*, *Dθ*, *DZ* for cylindrical; *DR*, *Dθ*, *DΦ* for spherical).

**RX, RY, RZ**

Scale factors to be applied to the X, Y, and Z keypoint coordinates in the active coordinate system (*RR*, *Rθ*, *RZ* for cylindrical; *RR*, *Rθ*, *RΦ* for spherical). Note that the *Rθ* and *RΦ* scale factors are interpreted as

angular *offsets*. For example, if CSYS = 1, *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

## Notes

Generates additional volumes (and their corresponding keypoints, lines, and areas) by extruding and scaling a pattern of areas in the active coordinate system.

If element attributes have been associated with the input area via the **AATT** command, the opposite area generated by the **VEXT** operation will also have those attributes (i.e., the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the areas are meshed or belong to meshed volumes, a 3-D mesh can be extruded with this command. Note that the *NDIV* argument on the **ESIZE** command should be set before extruding the meshed areas.

Scaling of the input areas, if specified, is performed first, followed by the extrusion.

In a non-Cartesian coordinate system, the **VEXT** command locates the end face of the volume based on the active coordinate system. However, the extrusion is made along a straight line between the end faces. Note that solid modeling in a toroidal coordinate system is not recommended.

---

### Caution

Use of the **VEXT** command can produce unexpected results when operating in a non-Cartesian coordinate system. For a detailed description of the possible problems that may occur, see [Solid Modeling](#) in the *Modeling and Meshing Guide*.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>By XYZ Offset

---

**\*VFACT**, *FACTR*, *FACT1*, *FACT2*, *FACT3*

Applies a scale factor to array parameters.

[APDL: Array Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *FACTR*

Scale factor applied to results (ParR) parameter. Defaults to 1.0.

### *FACT1*

Scale factor applied to first parameter (Par1 or ParI). Defaults to 1.0.

### *FACT2*

Scale factor applied to second parameter (Par2 or ParJ). Defaults to 1.0.

**FACT3**

Scale factor applied to third parameter (Par3 or ParK). Defaults to 1.0.

**Command Default**

Use 1.0 for all scale factors.

**Notes**

Applies a scale factor to parameters used in certain **\*Vxx** and **\*Mxx** operations. Typical scale factor applications are of the form:

$$\text{ParR} = \text{FACTR} * f(\text{FACT1} * \text{Par1})$$

or

$$\text{ParR} = \text{FACTR} * ((\text{FACT1} * \text{Par1}) \text{ o } (\text{FACT2} * \text{Par2}))$$

The factors are applied to each input parameter value before the operation and to the result value after the operation. The scale factor settings are *reset* to the default (1.0) after each **\*Vxx** or **\*Mxx** operation. Use **\*VSTAT** to list settings.

This command is valid in any processor.

**Menu Paths**

**Utility Menu>Parameters>Array Operations>Operation Settings**

**\*VFILL**, *ParR*, *Func*, *CON1*, *CON2*, *CON3*, *CON4*, *CON5*, *CON6*, *CON7*, *CON8*, *CON9*, *CON10*

**Fills an array parameter.**

[APDL: Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Argument Descriptions*****ParR***

The name of the resulting numeric array parameter vector. See **\*SET** for name restrictions.

***Func***

Fill function:

**DATA --**

Assign specified values *CON1*, *CON2*, etc. to successive array elements. Up to 10 assignments may be made at a time. Any *CON* values after a blank *CON* value are ignored.

**RAMP --**

Assign ramp function values:  $\text{CON1} + ((n-1) * \text{CON2})$ , where *n* is the loop number [**\*VLEN**]. To specify a constant function (no ramp), set *CON2* to zero.

**RAND --**

Assign random number values based on a uniform distribution:  $\text{RAND}(\text{CON1}, \text{CON2})$ , where *CON1* is the lower bound (defaults to 0.0) and *CON2* is the upper bound (defaults to 1.0).

**GDIS --**

Assign random sample of Gaussian distributions:  $GDIS(CON1,CON2)$ , where  $CON1$  is the mean (defaults to 0.0) and  $CON2$  is the standard deviation (defaults to 1.0).

**TRIA --**

Assigns random number values based on a triangular distribution:  $TRIA(CON1,CON2,CON3)$ , where  $CON1$  is the lower bound (defaults to 0.0),  $CON2$  is the location of the peak value

( $CON1 \leq CON2 \leq CON3$ ;  $CON2$  defaults to 0 if  $CON1 \leq 0 \leq CON3$ ,  $CON1$  if  $0 \leq CON1$ , or  $CON3$  if  $CON3 \leq 0$ ), and  $CON3$  is the upper bound (defaults to  $1.0 + CON1$  if  $CON1 \geq 0$  or 0.0 if  $CON1 \leq 0$ ).

**BETA --**

Assigns random number values based on a beta distribution:  $BETA(CON1,CON2,CON3,CON4)$ , where  $CON1$  is the lower bound (defaults to 0.0),  $CON2$  is the upper bound (defaults to  $1.0+CON1$  if  $CON1 \geq 0$  or 0.0 if  $CON1 \leq 0$ ), and  $CON3$  and  $CON4$  are the alpha and beta parameters, respectively, of the beta function. Alpha and beta must both be positive; they default to 1.0.

**GAMM --**

Assigns random number values based on a gamma distribution:  $GAMM(CON1,CON2,CON3)$ , where  $CON1$  is the lower bound (defaults to 0.0), and  $CON2$  and  $CON3$  are the alpha and beta parameters, respectively, of the gamma function. Alpha and beta must both be positive; they default to 1.0.

$CON1, CON2, CON3, \dots, CON10$

Constants used with above functions.

**Notes**

Operates on input data and produces one output array parameter vector according to:

$ParR = f(CON1, CON2, \dots)$

where the functions (f) are described above. Operations use successive array elements [**\*VLEN**, **\*VMASK**] with the default being all successive elements. For example, **\*VFILL**,A,RAMP,1,10 assigns  $A(1) = 1.0$ ,  $A(2) = 11.0$ ,  $A(3) = 21.0$ , etc. **\*VFILL**,B(5,1),DATA,1.5,3.0 assigns  $B(5,1) = 1.5$  and  $B(6,1) = 3.0$ . Absolute values and scale factors may be applied to the result parameter [**\*VABS**, **\*VFACT**]. Results may be cumulative [**\*VCUM**]. See the **\*VOPER** command for details.

This command is valid in any processor.

**Menu Paths**

Utility Menu>Parameters>Array Parameters>Fill

**VFOPT**, *Opt*, *Filename*, *Ext*, *Dir*, *Format*

**Specifies options for view factor file.**

SOLUTION: Radiosity  
AUX12: Radiosity Solver

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

*Opt*

View factor option:

**NEW**

Compute view factors and write them to a file.

**OFF**

Do not recompute view factors if they already exist in the database, otherwise compute them (default).

**READ**

Read view factors from a binary file. For subsequent **SOLVE** commands, switch to the default option (OFF).

**NONE**

Do not write view factors to a file.

***Fname***

File name for view factor matrix. Defaults to Jobname.

***Ext***

Filename extension for view factor matrix (default = vf).

***Dir***

Directory path for view factor matrix. If you do not specify a directory path, it will default to your working directory.

***Format***

File format for view factor file. Defaults to a Binary format.

**BINA**

Binary file (default).

**ASCI**

ASCII file.

**Notes**

This command allows you to deactivate the view factor computation (*Opt* = OFF) if the view factors already exist in the database. OFF is the default upon encountering the second and subsequent **SOLVE** commands in the solution processor. When using the READ option, only a previously-calculated view factor binary file is supported.

**Menu Paths**

**Main Menu>Preprocessor>Radiation Opts>View Factor**  
**Main Menu>Radiation Opt>Radiosity Meth>Compute**  
**Main Menu>Radiation Opt>Radiosity Meth>View Factor**  
**Main Menu>Solution>Radiation Opts>View Factor**

**VFQUERY, SRCELEM, TARELEM**

**Queries and prints element Hemicube view factors and average view factor.**

[AUX12: Radiosity Solver](#)

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

***SRCELEM***

Elements representing the source radiating surfaces used to query the view factor at the target element(s). If *SRCELEM* = P, graphical picking is enabled (valid only in the GUI). If *SRCELEM* = ALL, all selected ele-

ments will have their view factors queried. A component name may also be substituted for *SRCELEM*. Selected elements must be flagged for surface to surface radiation in order to query view factors (**SF**, **SFA**, or **SFE** with Lab = RDSF). The view factors must have been previously computed.

#### **TARELEM**

Element for view factor query. If *TARELEM* = P, graphical picking is enabled (valid only in the GUI). If *TARELEM* = ALL, all selected elements will have their view factors queried. A component name may also be substituted for *TARELEM*. Selected elements must be flagged for surface to surface radiation in order to query view factors (**SF**, **SFA**, or **SFE** with Lab = RDSF). The view factors must have been previously computed.

## Notes

View factors for each target element will be printed.

An average view factor for all target elements will be computed. (Use **\*GET** to retrieve the average value).

When resuming a database, issue the command **VFOPT,READ** before issuing the **VFQUERY** command.

## Menu Paths

**Main Menu>Radiation Opt>Radiosity Meth>Query**

### **VFSM**, *Action*, *ENCL*, *Scale*

**Scales view factor matrix to yield row sum values equal to one.**

[AUX12: Radiosity Solver](#)

MP ME <> PR <> <> <> <> <> <> <> PP <> EME MFS

#### **Action**

Action to be performed:

#### **Define**

Define a view factor summation (default)

#### **Clear**

Resets the scaling method to 0 for all enclosures. All subsequent arguments are ignored.

#### **Status**

Outputs scaling method for each enclosure in the model.

#### **ENCL**

Enclosure number for the view factor scaling.

#### **Scale**

Scaling factor key:

**0**

The view factor matrix is not changed and no scaling is applied (default).

**1**

Use scaling.



## Notes

For a perfect enclosure the view factor row sum should be 1.0. Computational errors may yield a row sum < 1.0, requiring a space mode temperature specification for the enclosure, and yielding an unrealistic temperature distribution. Use the **VFSM** command to correct this problem.

## Menu Paths

**This command cannot be accessed from a menu.**

**\*VFUN**, *ParR*, *Func*, *Par1*, *CON1*, *CON2*, *CON3*

**Performs a function on a single array parameter.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *ParR*

The name of the resulting numeric array parameter vector. See **\*SET** for name restrictions.

### *Func*

Function to be performed:

#### **ACOS** --

Arccosine:  $ACOS(Par1)$ .

#### **ASIN** --

Arcsine:  $ASIN(Par1)$ .

#### **ASORT** --

*Par1* is sorted in ascending order. **\*VCOL**, **\*VMASK**, **\*VCUM**, and **\*VLEN**,,NINC do not apply.

**\*VLEN**,NROW does apply.

#### **ATAN** --

Arctangent:  $ATAN(Par1)$ .

#### **COMP** --

Compress: Selectively compresses data set. "True" (**\*VMASK**) values of *Par1* (or row positions to be considered according to the *NINC* value on the **\*VLEN** command) are written in compressed form to *ParR*, starting at the specified position.

#### **COPY** --

Copy: *Par1* copied to *ParR*.

#### **COS** --

Cosine:  $COS(Par1)$ .

#### **COSH** --

Hyperbolic cosine:  $COSH(Par1)$ .

#### **DIRCOS** --

Direction cosines of the principal stresses ( $n \times 9$ ). *Par1* contains the  $n \times 6$  component stresses for the *n* locations of the calculations.

**DSORT --**

*Par1* is sorted in descending order. **\*VCOL**, **\*VMASK**, **\*VCUM**, and **\*VLEN**,,NINC do not apply. **\*VLEN**,NROW does apply.

**EULER --**

Euler angles of the principal stresses (*nX3*). *Par1* contains the *nX6* component stresses for the *n* locations of the calculations.

**EXP --**

Exponential: EXP(*Par1*).

**EXPA --**

Expand: Reverse of the COMP function. All elements of *Par1* (starting at the position specified) are written in expanded form to corresponding "true" (**\*VMASK**) positions (or row positions to be considered according to the *NINC* value on the **\*VLEN** command) of *ParR*.

**LOG --**

Natural logarithm: LOG(*Par1*).

**LOG10 --**

Common logarithm: LOG10(*Par1*).

**NINT --**

Nearest integer: 2.783 becomes 3.0, -1.75 becomes -2.0.

**NOT --**

Logical complement: values  $\leq 0.0$  (false) become 1.0 (true). Values  $> 0.0$  (true) become 0.0 (false).

**PRIN --**

Principal stresses (*nX5*). *Par1* contains the *nX6* component stresses for the *n* locations of the calculations.

**PWR --**

Power function: *Par1*\*\**CON1*. Exponentiation of any negative number in the vector *Par1* to a non-integer power is performed by exponentiating the positive number and prepending the minus sign. For example, -4\*\*2.3 is -(4\*\*2.3).

**SIN --**

Sine: SIN(*Par1*).

**SINH --**

Hyperbolic sine: SINH(*Par1*).

**SQRT --**

Square root: SQRT(*Par1*).

**TAN --**

Tangent: TAN(*Par1*).

**TANH --**

Hyperbolic tangent: TANH(*Par1*).

**TANG --**

Tangent to a path at a point: the slope at a point is determined by linear interpolation half way between the previous and next points. Points are assumed to be in the global Cartesian coordinate system. Path points are specified in array *Par1* (having 3 consecutive columns of data, with the columns containing the x, y, and z coordinate locations, respectively, of the points). Only the starting row index and the column index for the x coordinates are specified, such as A(1,1). The y and z coordinates of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and

A(1,3). The tangent result, *ParR*, must also have 3 consecutive columns of data and will contain the tangent direction vector (normalized to 1.0); such as 1,0,0 for an x-direction vector.

#### **NORM --**

Normal to a path and an input vector at a point: determined from the cross-product of the calculated tangent vector (see TANG) and the input direction vector (with the i, j, and k components input as *CON1*, *CON2*, and *CON3*). Points are assumed to be in the global Cartesian coordinate system. Path points are specified in array *Par1* (having 3 consecutive columns of data, with the columns containing the x, y, and z coordinate locations, respectively, of the points). Only the starting row index and the column index for the x coordinates are specified, such as A(1,1). The y and z coordinates of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3). The normal result, *ParR*, must also have 3 consecutive columns of data and will contain the normal direction vector (normalized to 1.0); such as 1,0,0 for an x-direction vector.

#### **LOCAL --**

Transforms global Cartesian coordinates of a point to the coordinates of a specified system: points to be transformed are specified in array *Par1* (having 3 consecutive columns of data, with the columns containing the x, y, and z global Cartesian coordinate locations, respectively, of the points). Only the starting row index and the column index for the x coordinates are specified, such as A(1,1). The y and z coordinates of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3). Results are transformed to coordinate system *CON1* (which may be any valid coordinate system number, such as 1,2,11,12, etc.). The transformed result, *ParR*, must also have 3 consecutive columns of data and will contain the corresponding transformed coordinate locations.

#### **GLOBAL --**

Transforms specified coordinates of a point to global Cartesian coordinates: points to be transformed are specified in array *Par1* (having 3 consecutive columns of data, with the columns containing the local coordinate locations (x, y, z or r,  $\theta$ , z or etc.) of the points). Only the starting row index and the column index for the x coordinates are specified, such as A(1,1). The y and z coordinates (or  $\theta$  and z, or etc.) of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3). Local coordinate locations are assumed to be in coordinate system *CON1* (which may be any valid coordinate system number, such as 1,2,11,12, etc.). The transformed result, *ParR*, must also have 3 consecutive columns of data, with the columns containing the global Cartesian x, y, and z coordinate locations, respectively.

#### ***Par1***

Array parameter vector in the operation.

#### ***CON1, CON2, CON3***

Constants (used only with the PWR, NORM, LOCAL, and GLOBAL functions).

## **Notes**

Operates on one input array parameter vector and produces one output array parameter vector according to:

$$ParR = f(Par1)$$

where the functions (f) are described below. Functions are based on the standard FORTRAN definitions where possible. Out-of-range function results (or results with exponents whose magnitudes are approximately greater than 32 or less than -32) produce a zero value. Input and output for angular functions may be radians (default) or degrees [**\*AFUN**]. *ParR* may be the same as *Par1*. Starting array element numbers must be defined for each array parameter vector if it does not start at the first location. For example, **\*VFUN,A,SQRT,B(5)** takes the square root of the fifth element of B and stores the result in the first element of A. Operations

continue on successive array elements [**\*VLEN**, **\*VMASK**] with the default being all successive elements. Absolute values and scale factors may be applied to all parameters [**\*VABS**, **\*VFACT**]. Results may be cumulative [**\*VCUM**]. Skipping array elements via **\*VMASK** or **\*VLEN** for the TANG and NORM functions skips only the writing of the results (skipped array element data are used in all calculations). See the **\*VOPER** command for details.

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Array Operations>Vector Functions

**VGEN**, *ITIME*, *NV1*, *NV2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

**Generates additional volumes from a pattern of volumes.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***ITIME***

Do this generation operation a total of *ITIME*s, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be > 1 for generation to occur.

### ***NV1*, *NV2*, *NINC***

Generate volumes from pattern beginning with *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is all selected volumes [**VSEL**]. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

### ***DX*, *DY*, *DZ***

Keypoint location increments in the active coordinate system (--, D $\theta$ , DZ for cylindrical, --, D $\theta$ , -- for spherical).

### ***KINC***

Keypoint increment between generated sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

### ***NOELEM***

Specifies if elements and nodes are also to be generated:

**0**

Generate nodes and elements associated with the original volumes, if they exist.

**1**

Do not generate nodes and elements.

### ***IMOVE***

Specifies whether to redefine the existing volumes:

**0**

Generate additional volumes as requested with the *ITIME* argument.

**1**

Move original volumes to new position retaining the same keypoint line, and area numbers (*ITIME*, *KINC*, and *NOELEM* are ignored). Corresponding meshed items are also moved if not needed at their original position.

## Notes

Generates additional volumes (and their corresponding keypoints, lines, areas and mesh) from a given volume pattern. The MAT, TYPE, REAL, and ESYS attributes are based upon the volumes in the pattern and not upon the current settings of the pointers. End slopes of the generated lines remain the same (in the active coordinate system) as those of the given pattern. For example, radial slopes remain radial, etc. Generations which produce volumes of a size or shape different from the pattern (i.e., radial generations in cylindrical systems, radial and phi generations in spherical systems, and theta generations in elliptical systems) are not allowed. Note that solid modeling in a toroidal coordinate system is not recommended. Volume, area, and line numbers are automatically assigned (beginning with the lowest available values [NUMSTR]).

## Menu Paths

**Main Menu>Preprocessor>Modeling>Copy>Volumes**  
**Main Menu>Preprocessor>Modeling>Move / Modify>Volumes**

**\*VGET**, *ParR*, *Entity*, *ENTNUM*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*, *KLOOP*

Retrieves values and stores them into an array parameter.

[APDL: Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *ParR*

The name of the resulting vector array parameter. See **\*SET** for name restrictions.

### *Entity*

Entity keyword. Valid keywords are NODE, ELEM, KP, LINE, AREA, VOLU, etc. as shown for *Entity* = in the table below.

### *ENTNUM*

The number of the entity (as shown for *ENTNUM* = in the table below).

### *Item1*

The name of a particular item for the given entity. Valid items are as shown in the *Item1* columns of the table below.

### *IT1NUM*

The number (or label) for the specified *Item1* (if any). Valid *IT1NUM* values are as shown in the *IT1NUM* columns of the table below. Some *Item1* labels do not require an *IT1NUM* value.

### *Item2, IT2NUM*

A second set of item labels and numbers to further qualify the item for which data is to be retrieved. Most items do not require this level of information.

### *KLOOP*

Field to be looped on:

#### **0 or 2 --**

Loop on the *ENTNUM* field (default).

#### **3 --**

Loop on the *Item1* field.

- 4** -- Loop on the *IT1NUM* field. Successive items are as shown with *IT1NUM*.
- 5** -- Loop on the *Item2* field.
- 6** -- Loop on the *IT2NUM* field. Successive items are as shown with *IT2NUM*.

## Notes

Retrieves values for specified items and stores the values in an output vector of a user-named array parameter according to:

$$ParR = f(Entity, ENTNUM, Item1, IT1NUM, Item2, IT2NUM)$$

where (f) is the **\*GET** function; *Entity*, *Item1*, and *Item2* are keywords; and *ENTNUM*, *IT1NUM*, and *IT2NUM* are numbers or labels corresponding to the keywords. Looping continues over successive entity numbers (*ENTNUM*) for the *KLOOP* default. For example, **\*VGET**,A(1),ELEM,5,CENT,X returns the centroid x-location of element 5 and stores the result in the first location of A. Retrieving continues with element 6, 7, 8, etc., regardless of whether the element exists or is selected, until successive array locations are filled. Use **\*VLEN** or **\*VMASK** to skip locations. Absolute values and scale factors may be applied to the result parameter [**\*VABS**, **\*VFACT**]. Results may be cumulative [**\*VCUM**]. See the **\*VOPER** command for general details. Results can be put back into an analysis by writing a file of the desired input commands with the **\*VWRITE** command. See also the **\*VPUT** command.

Both **\*GET** and **\*VGET** retrieve information from the active data stored in memory. The database is often the source, and sometimes the information is retrieved from common memory blocks that ANSYS uses to manipulate information. Although POST1 and POST26 operations use a \*.rst file, GET data is accessed from the database or from the common blocks. Get operations do not access the \*.rst file directly.

The **\*VGET** command retrieves both the unprocessed real and the imaginary parts (original and duplicate sector nodes and elements) of a cyclic symmetry solution.

This command is valid in any processor.

**Table 260 \*VGET - PREP7 Items**

PREP7 Items		
<i>Entity</i> = NODE, <i>ENTNUM</i> = <i>n</i> (node number)		
Item1	IT1NUM	Description
LOC	X, Y, Z	X, Y, or Z location in the active coordinate system.
ANG	XY, YZ, ZX	THXY, THYZ, THZX rotation angle.
NSEL		Select status of node <i>n</i> (-1 - unselected, 0 - undefined, 1 - selected).
NLIST		Returns the list of selected nodes ( <i>ENTNUM</i> is ignored).
<i>Entity</i> = ELEM, <i>ENTNUM</i> = <i>n</i> (element number)		
Item1	IT1NUM	Description
NODE	1,2,--20	Node number at position 1,2,--20 of element <i>n</i> .

### PREP7 Items

CENT	X, Y, Z	Centroid X, Y, or Z location (based on shape function) in the active coordinate system.
ADJ	1, 2, -- 6	Number of element adjacent to face 1, 2, -- 6.
ATTR	name	Number assigned to attribute <i>name</i> = MAT, TYPE, REAL, ESYS, ENAM, or SECN).
GEOM		Characteristic element geometry. Length of line element (straight line between ends), area of area element, or volume of volume element. Issuing <b>*VGET</b> for an element returns a signed value. To always get a positive value, issue <b>*VABS,1</b> just prior to issuing <b>*VGET,par(n),ELEM,x,GEOM</b> .
ESEL		Select status of element <i>n</i> (-1 - unselected, 0 - undefined, 1 - selected).
SHPAR	<i>Test</i>	Element shape test result for selected element <i>n</i> , where <i>Test</i> =ANGD ( <b>SHELL28</b> corner angle deviation), ASPE (aspect ratio), JACR (Jacobian ratio), MAXA (maximum corner angle), PARA (deviation from parallelism of opposite edges), or WARP (warping factor).
ELIST		Returns the list of selected elements ( <i>ENTNUM</i> is ignored). <i>Entity</i> = KP, <i>ENTNUM</i> = <i>n</i> (keypoint number)
	Item1      IT1NUM	Description
LOC	X, Y, Z	X, Y, or Z location in the active coordinate system.
ATTR	name	Number assigned to attribute ( <i>name</i> = MAT, TYPE, REAL, ESYS, NODE or ELEM).
DIV		Divisions (element size setting) from <b>KESIZE</b> command.
KSEL		Select status of keypoint <i>n</i> (-1 - unselected, 0 - undefined, 1 - selected).
KLIST		Returns the list of selected keypoints ( <i>ENTNUM</i> is ignored). <i>Entity</i> = LINE, <i>ENTNUM</i> = <i>n</i> (line number)
	Item1      IT1NUM	Description
KP	1,2	Keypoint number at position 1 or 2.
ATTR	name	Number assigned to attribute ( <i>name</i> = MAT, TYPE, REAL, ESYS, NNOD, NELM, or NDIV). NNOD = number of nodes, NELM = number of elements, NDIV = number of divisions.
LENG		Length.
LSEL		Select status of line <i>n</i> (-1 - unselected, 0 - undefined, 1 - selected).
LLIST		Returns the list of selected lines ( <i>ENTNUM</i> is ignored). <i>Entity</i> = AREA, <i>ENTNUM</i> = <i>n</i> (area number)
	Item1      IT1NUM	Description
LOOP	1,2,-- <i>I</i>	Loop number. Must be input if LINE number is to be retrieved.
	Item2      IT2NUM	Description

**PREP7 Items**

LINE	1, 2, -- <i>p</i>	Line number at position 1, 2, --- <i>p</i> .
ATTR	name	Number assigned to attribute ( <i>name</i> = MAT, TYPE, REAL, ESYS, SECN, NNOD, or NELM). NNOD = number of nodes, NELM = number of elements.
AREA		Area (after last <b>ASUM</b> ).
ASEL		Select status of area <i>n</i> (-1 - unselected, 0 - undefined, 1 - selected).
ALIST		Returns the list of selected areas ( <i>ENTNUM</i> is ignored).
<i>Entity</i> = VOLU, <i>ENTNUM</i> = <i>n</i> (volume number)		
Item1	IT1NUM	Description
SHELL	1,2,--1	Shell number. Must be input if AREA number is to be retrieved.
Item2	IT2NUM	Description
AREA	1, 2, -- <i>p</i>	Area number at position 1, 2, --- <i>p</i> .
ATTR	name	Number assigned to attribute ( <i>name</i> = MAT, TYPE, REAL, ESYS, NNOD, or NELM). NNOD = number of nodes, NELM = number of elements.
VOLU		Volume (after last <b>VSUM</b> ).
VSEL		Select status of volume <i>n</i> (-1 - unselected, 0 - undefined, 1 - selected).
VLIST		Returns the list of selected volumes ( <i>ENTNUM</i> is ignored).
<i>Entity</i> = CDSY, <i>ENTNUM</i> = <i>n</i> (coordinate system number)		
Item1	IT1NUM	Description
LOC	X, Y, Z	X, Y, or Z origin location (global Cartesian coordinate).
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle (°) relative to the global Cartesian coordinate system.
ATTR	name	Number assigned to attribute ( <i>name</i> = KCS, KTHET, KPHI, PAR1, or PAR2). A -1.0 is returned for KCS if coordinate system is undefined).
<i>Entity</i> = RCON, <i>ENTNUM</i> = <i>n</i> (real constant set number)		
Item1	IT1NUM	Description
CONST	1,2,-- <i>m</i>	Real constant value for constant 1,2,--- <i>m</i> .
<i>Entity</i> = TLAB, <i>ENTNUM</i> = <i>n</i> ( <i>Tlab</i> is the data table label: BKIN, MKIN, MISO, etc. as described on the <b>TB</b> command. <i>n</i> is the material number.)		
Item1	IT1NUM	Description
TEMP	val	Temperature value (if any) at which to retrieve table data.
Item2	IT2NUM	Description
CONST	<i>num</i>	Constant number whose value is to be retrieved (see <a href="#">Data Tables - Implicit Analysis</a> in the <a href="#">Element Reference</a> ). For constants input



**PREP7 Items**

as X, Y points, the constant numbers are consecutive with the X constants being the odd numbers, beginning with one.

**Table 261 \*VGET - POST1 Items**

*Entity* = NODE, *ENTNUM* = *n* (node number)

Vector items are in the active results coordinate system unless otherwise specified.

Item1	IT1NUM	Description
Valid labels for nodal degree of freedom results are:		
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
TEMP		Temperature. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use TBOT, TE2, TE3, . . . , TTOP instead of TEMP. Alternative get functions: TEMP(N), TBOT(N), TE2(N), etc.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity.
A	X, Y, Z	X, Y, or Z magnetic vector potential.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy (FLOTRAN).
ENDS		Turbulent energy dissipation (FLOTRAN).

Valid labels for element nodal results are:

Item1	IT1NUM	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1,2,3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.

*Entity* = NODE, *ENTNUM* = *n* (node number)

Vector items are in the active results coordinate system unless otherwise specified.

Item1	IT1NUM	Description
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
HS	X, Y, Z	Component magnetic field intensity from current sources (in the global Cartesian coordinate system).
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution.
TG	X, Y, Z, SUM	Component thermal gradient and sum.
TF	X, Y, Z, SUM	Component thermal flux and sum.
PG	X, Y, Z, SUM	Component pressure gradient and sum.
EF	X, Y, Z, SUM	Component electric field and sum.
D	X, Y, Z, SUM	Component electric flux density and sum.
H	X, Y, Z, SUM	Component magnetic field intensity and sum.
B	X, Y, Z, SUM	Component magnetic flux density and sum.
FMAG	X, Y, Z, SUM	Component electromagnetic force and sum.

### Note

Element nodal results are the average nodal value of the selected elements.

Valid labels for FLOTRAN nodal results are:

Item1	IT1NUM	Description
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.

*Entity* = NODE, *ENTNUM* = *n* (node number)

Vector items are in the active results coordinate system unless otherwise specified.

Item1	IT1NUM	Description
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.

*Entity* = ELEM, *ENTNUM* = *n* (element number)

Valid labels for element results are:

ETAB	Lab	Any user-defined element table label (see <b>ETABLE</b> command).
------	-----	---

## Menu Paths

Utility Menu>Parameters>Get Array Data

**VGET**, *Par*, *IR*, *TSTRT*, *KCPLX*

Moves a variable into an array parameter vector.

POST26: Special Purpose

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Par**

Array parameter vector in the operation.

**IR**

Reference number of the variable (1 to NV [**NUMVAR**]).

**TSTRT**

Time (or frequency) corresponding to start of *IR* data. If between values, the nearer value is used.

**KCPLX**

Complex number key:

**0**

Use the real part of the *IR* data.

**1**

Use the imaginary part of the *IR* data.

## Notes

Moves a variable into an array parameter vector. The starting array element number must be defined. For example, **VGET**,A(1),2 moves variable 2 (starting at time 0.0) to array parameter A. Looping continues from array element A(1) with the index number incremented by one until the variable is filled. The number of loops may be controlled with the **\*VLEN** command (except that loop skipping (*NINCL*) is not allowed). For multi-dimensioned array parameters, only the first (row) subscript is incremented.

## Menu Paths

**Main Menu>TimeHist Postpro>Table Operations>Variable to Par**

---

**VGLUE**, NV1, NV2, NV3, NV4, NV5, NV6, NV7, NV8, NV9

**Generates new volumes by "gluing" volumes.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NV1, NV2, NV3, . . . , NV9*

Numbers of the volumes to be glued. If *NV1* = ALL, all selected volumes will be glued (*NV2* to *NV9* will be ignored). If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

## Notes

Use of the **VGLUE** command generates new volumes by "gluing" input volumes. The glue operation redefines the input volumes so that they share areas along their common boundaries. The new volumes encompass the same geometry as the original volumes. This operation is only valid if the intersections of the input volumes are areas along the boundaries of those volumes. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

The **VGLUE** command results in the merging of areas, lines, and keypoints at the common volume boundaries. The areas, lines, and keypoints of the lower numbered volume will be kept. This means one must be aware of volume numbering when multiple **VGLUE** commands are applied to avoid any "ungluing" of geometry.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Glue>Volumes**

---

**/VIEW**, *WN, XV, YV, ZV*

**Defines the viewing direction for the display.**

GRAPHICS:Views

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WN**

Window number (or ALL) to which command applies (defaults to 1).

**XV, YV, ZV**

The object is viewed along the line from point *XV,YV,ZV* (in the global coordinate system) to the global coordinate system origin. For section displays, the cutting plane is assumed to be perpendicular to this line. If *XV = WP*, modify view to be normal to the currently defined working plane. Defaults to (0,0,1).

## Command Default

0,0,1 view. The default reference orientation is X-axis horizontal to the right, Y-axis vertical upward, and Z-axis out from the screen (normal). See **/VUP** command to change reference orientation.

## Notes

The view line is always normal to the screen. The view is selected by defining a point (in the global Cartesian coordinate system) representing a point along the viewing line. This point, and the global Cartesian coordinate system origin, define the line along which the object is viewed while looking toward the origin. Any point along the view line may be used, i.e., (1,1,1) and (2,2,2) give the same view. The display orientation may be changed as desired [**/ANGLE**]. The display coordinate system type may be changed (from Cartesian to cylindrical, spherical, toroidal, etc.) with the **DSYS** command.

This command is valid in any processor.

## Menu Paths

**Main Menu>General Postproc>Path Operations>Define Path>On Working Plane**

**Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane**

**Utility Menu>PlotCtrls>Pan, Zoom, Rotate**

**Utility Menu>PlotCtrls>View Settings>Viewing Direction**

**VIMP**, *VOL, CHGBND, IMPLEVEL*

**Improves the quality of the tetrahedral elements in the selected volume(s).**

PREP7:Meshing

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**VOL**

Number of the volume containing the tetrahedral elements to be improved. If *VOL = ALL* (default), improve the tetrahedral elements in all selected volumes. If *VOL = P*, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *VOL*.

**CHGBND**

Specifies whether to allow boundary modification. Boundary modification includes such things as changes in the connectivity of the element faces on the boundary and the addition of boundary nodes. (Also see "Notes" (p. 1826) below for important usage information for *CHGBND*.)

**0**

Do not allow boundary modification.

**1**

Allow boundary modification (default).

**IMPLEVEL**

Identifies the level of improvement to be performed on the elements. (Improvement occurs primarily through the use of face swapping and node smoothing techniques.)

**0**

Perform the least amount of swapping/smoothing.

**1**

Perform an intermediate amount of swapping/smoothing.

**2**

Perform the greatest amount of swapping/smoothing.

**3**

Perform the greatest amount of swapping/smoothing, plus additional improvement techniques (default).

## Notes

**VIMP** is useful for further improving a volume mesh created in ANSYS [**VMESH**], especially quadratic tetrahedral element meshes.

The **VIMP** command enables you to improve a given tetrahedral mesh by reducing the number of poorly-shaped tetrahedral elements (in particular, the number of sliver tetrahedral elements)--as well as the overall number of elements--in the mesh. It also improves the overall quality of the mesh.

Regardless of the value of the *CHGBND* argument, boundary mid-nodes can be moved as long as you are not using p-method analysis. When *CHGBND* = 0 and you are using p-method analysis, boundary mid-nodes cannot be moved. (ANSYS issues an error message if it would be necessary to move boundary mid-nodes in order to generate valid quadratic elements.)

When loads or constraints have been placed on boundary nodes or mid-nodes, and boundary mid-nodes are later moved, ANSYS issues a warning message to let you know that it will not update the loads or constraints.

Even when *CHGBND* = 1, no boundary modification is performed on areas and lines that are not modifiable (for example, areas that are adjacent to other volumes or that contain shell elements, or lines that are not incident on modifiable areas, contain beam elements, or have line divisions specified for them [**LESIZE**]).

## Menu Paths

**Main Menu>Preprocessor>Meshing>Modify Mesh>Improve Tets>Volumes**

---

**VINP**, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

**Finds the pairwise intersection of volumes.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NV1*, *NV2*, *NV3*, . . . , *NV9*

Numbers of volumes to be intersected pairwise. If *NV1* = ALL, *NV2* to *NV9* are ignored and the pairwise intersection of all selected volumes is found. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

## Notes

Finds the pairwise intersection of volumes. The pairwise intersection is defined as all regions shared by any two or more volumes listed on this command. New volumes will be generated where the original volumes intersect pairwise. If the regions of pairwise intersection are only areas, new areas will be generated. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Pairwise>Volumes**

---

**VINV**, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

**Finds the intersection of volumes.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NV1*, *NV2*, *NV3*, . . . , *NV9*

Numbers of volumes to be intersected. If *NV1* = ALL, *NV2* to *NV9* are ignored, and the intersection of all selected volumes is found. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

## Notes

Finds the common (not pairwise) intersection of volumes. The common intersection is defined as the regions shared (in common) by **all** volumes listed on this command. New volumes will be generated where the original volumes intersect. If the regions of intersection are only areas, new areas will be generated instead. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Common>Volumes**

---

**\*VITRP**, *ParR*, *ParT*, *ParI*, *ParJ*, *ParK*

**Forms an array parameter by interpolation of a table.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *ParR*

The name of the resulting array parameter. See **\*SET** for name restrictions.

### *ParT*

The name of the TABLE array parameter. The parameter must exist as a dimensioned array of type TABLE [**\*DIM**].

### *ParI*

Array parameter vector of I (row) index values for interpolation in *ParT*.

### *ParJ*

Array parameter vector of J (column) index values for interpolation in *ParT* (which must be at least 2-D).

### *ParK*

Array parameter vector of K (depth) index values for interpolation in *ParT* (which must be 3-D).

## Notes

Forms an array parameter (of type ARRAY) by interpolating values of an array parameter (of type TABLE) at specified table index locations according to:

$$ParR = f(ParT, ParI, ParJ, ParK)$$

where *ParT* is the type TABLE array parameter, and *ParI*, *ParJ*, *ParK* are the type ARRAY array parameter vectors of index values for interpolation in *ParT*. See the **\*DIM** command for TABLE and ARRAY declaration types. Linear interpolation is used. The starting array element number for the TABLE array (*ParT*) is not used (but a value must be input). Starting array element numbers must be defined for each array parameter vector if it does not start at the first location. For example, **\*VITRP**,R(5),TAB(1,1),X(2),Y(4) uses the second element of X and the fourth element of Y as index values (row and column) for a 2-D interpolation in TAB and stores the result in the fifth element of R. Operations continue on successive array elements [**\*VLEN**, **\*VMASK**] with the default being all successive elements. Absolute values and scale factors may be applied to the result parameter [**\*VABS**, **\*VFACT**]. Results may be cumulative [**\*VCUM**]. See the **\*VOPER** command for details.

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Array Operations>Vector Interpolate



**\*VLEN**, *NROW*, *NINC*

**Specifies the number of rows to be used in array parameter operations.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *NROW*

Number of rows to be used with the **\*Vxx** or **\*Mxx** operations. Defaults to the number of rows needed to fill the result array.

### *NINC*

Perform the operation on every *NINC* row (defaults to 1).

## Command Default

Fill all locations of the result array from the specified starting location.

## Notes

Specifies the number of rows to be used in array parameter operations. The size of the submatrix used is determined from the upper left starting array element (defined on the operation command) to the lower right array element (defined by the number of rows on this command and the number of columns on the **\*VCOL** command). *NINC* allows skipping row operations for some operation commands. Skipped rows are included in the row count. The starting row number must be defined on the operation command for each parameter read and for the result written.

The default *NROW* is calculated from the maximum number of rows of the result array (the **\*DIM** row dimension) minus the starting location + 1. For example, **\*DIM,R,,10** and a starting location of R(7) gives a default of 4 loops (filling R(7), R(8), R(9), and R(10)). Repeat operations automatically terminate at the last row of the result array. Existing values in the rows and columns of the results matrix remain unchanged where not overwritten by the requested input or operation values.

The stride (*NINC*) allows operations to be performed at regular intervals. It has no effect on the total number of row operations. Skipped operations retain the previous result. For example, **\*DIM,R,,6**, with a starting location of R(1), *NROW* = 10, and *NINC* = 2 calculates values for locations R(1), R(3), and R(5) and retains values for locations R(2), R(4), and R(6). A more general skip control may be done by masking [**\*VMASK**]. The row control settings are *reset* to the defaults after each **\*Vxx** or **\*Mxx** operation. Use **\*VSTAT** to list settings.

This command is valid in any processor.

## Menu Paths

**Utility Menu>Parameters>Array Operations>Operation Settings**

**VLIST**, *NV1*, *NV2*, *NINC***Lists the defined volumes.**[PREP7: Volumes](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***NV1*, *NV2*, *NINC***

List volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL (default), *NV2* and *NINC* are ignored and all selected volumes [**VSEL**] are listed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

**Notes**

An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **VATT** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer [TYPE, MAT, REAL, or ESYS] that was active during meshing (and will be reset to zero if the mesh is cleared). A "-1" in the "nodes" column indicates that the volume has been meshed but there are no interior nodes. The volume size is listed only if a **VSUM** command has been performed on the volume. Volume orientation attributes (KZ1 and KZ2) are listed only if a **VEORIENT** command was previously used to define an orientation for the volume.

This command is valid in any processor.

**Menu Paths**

Utility Menu&gt;List&gt;Volumes

**VLSCALE**, *NV1*, *NV2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE***Generates a scaled set of volumes from a pattern of volumes.**[PREP7: Volumes](#)

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***NV1*, *NV2*, *NINC***

Set of volumes (*NV1* to *NV2* in steps of *NINC*) that defines the pattern to be scaled. *NV2* defaults to *NV1*, *NINC* defaults to 1. If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is defined by all selected volumes. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

***RX*, *RY*, *RZ***

Scale factors to be applied to the X, Y, and Z keypoint coordinates in active coordinate system (*RR*, *R $\theta$* , *RZ* for cylindrical; *RR*, *R $\theta$* , *R $\Phi$*  for spherical). Note that the *R $\theta$*  and *R $\Phi$*  scale factors are interpreted as angular offsets. For example, if CSYS = 1, *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

**KINC**

Increment to be applied to keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned [**NUMSTR**].

**NOELEM**

Specifies whether nodes and elements are also to be generated:

**0**

Nodes and elements associated with the original volumes will be generated (scaled) if they exist.

**1**

Nodes and elements will *not* be generated.

**IMOVE**

Specifies whether volumes will be moved or newly defined:

**0**

Additional volumes will be generated.

**1**

Original volumes will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old volumes are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

## Notes

Generates a scaled set of volumes (and their corresponding keypoints, lines, areas, and mesh) from a pattern of volumes. The *MAT*, *TYPE*, *REAL*, and *ESYS* attributes are based on the volumes in the pattern and not the current settings. Scaling is done in the active coordinate system. Volumes in the pattern could have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Volumes

### \*VMASK, *Par*

Specifies an array parameter as a masking vector.

[APDL: Array Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

***Par***

Name of the mask parameter. The starting subscript must also be specified.

## Command Default

No mask parameter specified (use true for all operations).

## Notes

Specifies the name of the parameter whose values are to be checked for each resulting row operation. The mask vector usually contains only 0 (for false) and 1 (for true) values. For each row operation the corresponding mask vector value is checked. A true value allows the operation to be done. A false value skips the operation (and retains the previous results). A mask vector can be created from direct input, such as `M(1) = 1,0,0,1,1,0,1`; or from the DATA function of the **\*VFILL** command. The NOT function of the **\*VFUN** command can be used to reverse the logical sense of the mask vector. The logical compare operations (LT, LE, EQ, NE, GE, and GT) of the **\*VOPER** command also produce a mask vector by operating on two other vectors. Any numeric vector can be used as a mask vector since the actual interpretation assumes values less than 0.0 are 0.0 (false) and values greater than 0.0 are 1.0 (true). If the mask vector is not specified (or has fewer values than the result vector), true (1.0) values are assumed for the unspecified values. Another skip control may be input with `NINC` on the **\*VLEN** command. If both are present, operations occur only when both are true. The mask setting is *reset* to the default (no mask) after each **\*Vxx** or **\*Mxx** operation. Use **\*VSTAT** to list settings.

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Array Operations>Operation Settings

**VMESH**, *NV1*, *NV2*, *NINC*

**Generates nodes and volume elements within volumes.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NV1*, *NV2*, *NINC*

Mesh volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and all selected volumes [**VSEL**] are meshed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

## Notes

Missing nodes required for the generated elements are created and assigned the lowest available numbers [**NUMSTR**]. During a batch run and if elements already exist, a mesh abort will write an alternative database file (`File.DBE`) for possible recovery.

Tetrahedral mesh expansion [**MOPT**,`TETEXPND,Value`] is supported for both the **VMESH** and **FVMESH** commands.

## Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Free

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided

**VOFFST**, *NAREA*, *DIST*, *KINC*

**Generates a volume, offset from a given area.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NAREA***

Area from which generated volume is to be offset. If *NAREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

***DIST***

Distance normal to given area at which keypoints for generated volume are to be located. Positive normal is determined from the right-hand rule keypoint order.

***KINC***

Increment to be applied to the keypoint numbers between sets. If zero, keypoint numbers will be automatically assigned beginning with the lowest available value [**NUMSTR**].

**Notes**

Generates a volume (and its corresponding keypoints, lines, and areas) by offsetting from an area. The direction of the offset varies with the given area normal. End slopes of the generated lines remain the same as those of the given pattern.

If element attributes have been associated with the input area via the **AATT** command, the opposite area generated by the **VOFFST** operation will also have those attributes (i.e., the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the areas are meshed or belong to meshed volumes, a 3-D mesh can be extruded with this command. Note that the *NDIV* argument on the **ESIZE** command should be set before extruding the meshed areas.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>Along Normal**

**VOLUMES**

**Specifies "Volumes" as the subsequent status topic.**

[PREP7: Status](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and appear in the log file (*Jobname*.LOG) if status is requested for some items by choosing **Utility Menu>List>Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

## Menu Paths

This command cannot be accessed from a menu.

---

**\*VOPER**, *ParR*, *Par1*, *Oper*, *Par2*, *CON1*, *CON2*

Operates on two array parameters.

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *ParR*

The name of the resulting array parameter vector. See **\*SET** for name restrictions.

### *Par1*

First array parameter vector in the operation. May also be a scalar parameter or a literal constant.

### *Oper*

Operations:

#### **ADD--**

Addition:  $Par1+Par2$ .

#### **SUB --**

Subtraction:  $Par1-Par2$ .

#### **MULT --**

Multiplication:  $Par1*Par2$ .

#### **DIV --**

Division:  $Par1/Par2$  (a divide by zero results in a value of zero).

#### **MIN --**

Minimum: minimum of  $Par1$  and  $Par2$ .

#### **MAX --**

Maximum: maximum of  $Par1$  and  $Par2$ .

#### **LT --**

Less than comparison:  $Par1<Par2$  gives 1.0 if true, 0.0 if false.

#### **LE --**

Less than or equal comparison:  $Par1 \leq Par2$  gives 1.0 if true, 0.0 if false.

#### **EQ --**

Equal comparison:  $Par1 = Par2$  gives 1.0 if true, 0.0 if false.

#### **NE --**

Not equal comparison:  $Par1 \neq Par2$  gives 1.0 if true, 0.0 if false.

#### **GE --**

Greater than or equal comparison:  $Par1 \geq Par2$  gives 1.0 if true, 0.0 if false.

#### **GT --**

Greater than comparison:  $Par1>Par2$  gives 1.0 if true, 0.0 if false.

**DER1 --**

First derivative:  $d(\text{Par1})/d(\text{Par2})$ . The derivative at a point is determined over points half way between the previous and next points (by linear interpolation). *Par1* must be a function (a unique *Par1* value for each *Par2* value) and *Par2* must be in ascending order.

**DER2 --**

Second derivative:  $d^2(\text{Par1})/d(\text{Par2})^2$ . See also DER1.

**INT1 --**

Single integral:  $\int \text{Par1} d(\text{Par2})$ , where *CON1* is the integration constant. The integral at a point is determined by using the single integration procedure described in the [Theory Reference for the Mechanical APDL and Mechanical Applications](#).

**INT2 --**

Double integral:  $\iint \text{Par1} d(\text{Par2})$ , where *CON1* is the integration constant of the first integral and *CON2* is the integration constant of the second integral. If *Par1* contains acceleration data, *CON1* is the initial velocity and *CON2* is the initial displacement. See also INT1.

**DOT --**

Dot product:  $\text{Par1} \cdot \text{Par2}$ . *Par1* and *Par2* must each have three consecutive columns of data, with the columns containing the i, j, and k vector components, respectively. Only the starting row index and the column index for the i components are specified for *Par1* and *Par2*, such as A(1,1). The j and k components of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3).

**CROSS --**

Cross product:  $\text{Par1} \times \text{Par2}$ . *Par1*, *Par2*, and *ParR* must each have 3 components, respectively. Only the starting row index and the column index for the i components are specified for *Par1*, *Par2*, and *ParR*, such as A(1,1). The j and k components of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3).

**GATH --**

Gather: For a vector of position numbers, *Par2*, copy the value of *Par1* at each position number to *ParR*. Example: for *Par1* = 10,20,30,40 and *Par2* = 2,4,1; *ParR* = 20,40,10.

**SCAT --**

Scatter: Opposite of GATH operation. For a vector of position numbers, *Par2*, copy the value of *Par1* to that position number in *ParR*. Example: for *Par1* = 10,20,30,40,50 and *Par2* = 2,1,0,5,3; *ParR* = 20,10,50,0,40.

**ATN2 --**

Arctangent: arctangent of  $\text{Par1}/\text{Par2}$  with the sign of each component considered.

**LOCAL --**

Transform the data in *Par1* from the global Cartesian coordinate system to the local coordinate system given in *CON1*. *Par1* must be an  $N \times 3$  (i.e., vector) or an  $N \times 6$  (i.e., stress or strain tensor) array. If the local coordinate system is a cylindrical, spherical, or toroidal system, then you must provide the global Cartesian coordinates in *Par2* as an  $N \times 3$  array. Set *CON2* = 1 if the data is strain data.

**GLOBAL --**

Transform the data in *Par1* from the local coordinate system given in *CON1* to the global Cartesian coordinate system. *Par1* must be an  $N \times 3$  (i.e., vector) or an  $N \times 6$  (i.e., stress or strain tensor) array. If the local coordinate system is a cylindrical, spherical, or toroidal system, then you must provide the global Cartesian coordinates in *Par2* as an  $N \times 3$  array. Set *CON2* = 1 if the data is strain data.

**Par2**

Second array parameter vector in the operation. May also be a scalar parameter or a literal constant.

**CON1**

First constant (used only with the INT1 and INT2 operations).

**CON2**

Second constant (used only with the INT2 operation).

## Notes

Operates on two input array parameter vectors and produces one output array parameter vector according to:

$$ParR = Par1 \text{ o } Par2$$

where the operations (o) are described below. *ParR* may be the same as *Par1* or *Par2*. Absolute values and scale factors may be applied to all parameters [**\*VABS**, **\*VFACT**]. Results may be cumulative [**\*VCUM**]. Starting array element numbers must be defined for each array parameter vector if it does not start at the first location, such as **\*VOPER**,A,B(5),ADD,C(3) which adds the third element of C to the fifth element of B and stores the result in the first element of A. Operations continue on successive array elements [**\*VLEN**, **\*VMASK**] with the default being all successive elements. Skipping array elements via **\*VMASK** or **\*VLEN** for the DER\_ and INT\_ functions skips only the writing of the results (skipped array element data are used in all calculations).

Parameter functions and operations are available to operate on a scalar parameter or a single element of an array parameter, such as SQRT(B) or SQRT(A(4)). See the **\*SET** command for details. Operations on a sequence of array elements can be done by repeating the desired function or operation in a do-loop [**\*DO**]. The vector operations within the ANSYS program (**\*VXX** commands) are internally programmed do-loops that conveniently perform the indicated operation over a sequence of array elements. If the array is multidimensional, only the first subscript is incremented in the do-loop, that is, the operation repeats in column vector fashion "down" the array. For example, for A(1,5), A(2,5), A(3,5), etc. The starting location of the row index must be defined for each parameter read and for the result written.

The default number of loops is from the starting result location to the last result location and can be altered with the **\*VLEN** command. A logical mask vector may be defined to control at which locations the operations are to be skipped [**\*VMASK**]. The default is to skip no locations. Repeat operations automatically terminate at the last array element of the result array column if the number of loops is undefined or if it exceeds the last result array element. Zeroes are used in operations for values read beyond the last array element of an input array column. Existing values in the rows and columns of the results matrix remain unchanged where not changed by the requested operation values. The result array column may be the same as the input array column since results in progress are stored in a temporary array until being moved to the results array at the end of the operation. Results may be overwritten or accumulated with the existing results [**\*VCUM**]. The default is to overwrite results. The absolute value may be used for each parameter read or written [**\*VABS**]. A scale factor (defaulting to 1.0) is also applied to each parameter read and written [**\*VFACT**].

This command is valid in any processor.

## Menu Paths

**Utility Menu>Parameters>Array Operations>Vector Operations**

---



**VOVLAP**, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

**Overlaps volumes.**

PREP7: Booleans

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NV1*, *NV2*, *NV3*, . . . , *NV9*

Numbers of volumes to be operated on. If *NV1* = ALL, *NV2* to *NV9* are ignored and all selected volumes are used. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

## Notes

Overlaps volumes. Generates new volumes which encompass the geometry of all the input volumes. The new volumes are defined by the regions of intersection of the input volumes, and by the complementary (non-intersecting) regions. See the [Modeling and Meshing Guide](#) for an illustration. This operation is only valid when the region of intersection is a volume. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Overlap>Volumes

**\*VPLOT**, *ParX*, *ParY*, *Y2*, *Y3*, *Y4*, *Y5*, *Y6*, *Y7*, *Y8*

**Graphs columns (vectors) of array parameters.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

*ParX*

Name of the array parameter whose column vector values will be the abscissa of the graph. If blank, row subscript numbers are used instead. *ParX* is not sorted by the program.

*ParY*

Name of the array parameter whose column vector values will be graphed against the *ParX* values.

*Y2*, *Y3*, *Y4*, . . . , *Y8*

Additional column subscript of the *ParY* array parameter whose values are to be graphed against the *ParX* values.

## Notes

The column to be graphed and the starting row for each array parameter must be specified as subscripts. Additional columns of the *ParY* array parameter may be graphed by specifying column numbers for *Y2*, *Y3*, . . . , *Y8*. For example, **\*VPLOT**, TIME (4,6), DISP (8,1),2,3 specifies that the 1st, 2nd, and 3rd columns of array parameter DISP (all starting at row 8) are to be graphed against the 6th column of array parameter TIME (starting at row 4). The columns will be graphed from the starting row to their maximum extent. See the

**\*VLEN** and **\*VMASK** commands to limit or skip data to be graphed. The array parameters specified on the **\*VPLOT** command must be of the same type (type ARRAY or TABLE; [**\*DIM**]). Arrays of type TABLE will be graphed as continuous curves. Arrays of type ARRAY will be displayed in bar chart fashion.

The normal curve labeling scheme for **\*VPLOT** is to label curve 1 "COL 1", curve 2 "COL 2" and so on. You can use the **/GCOLUMN** command to apply user-specified labels (8 characters maximum) to your curves. See [Modifying Curve Labels](#) in the *ANSYS Parametric Design Language Guide* for more information on using **/GCOLUMN**.

When a graph plot reaches minimum or maximum y-axis limits, ANSYS indicates the condition by clipping the graph. The clip appears as a horizontal magenta line. ANSYS calculates y-axis limits automatically; however, you can modify the (YMIN and YMAX) limits via the **/YRANGE** command.

This command is valid in any processor.

## Menu Paths

Utility Menu>Plot>Array Parameters

---

**VPLOT**, *NV1*, *NV2*, *NINC*, *DEGEN*, *SCALE*

**Displays the selected volumes.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NV1*, *NV2*, *NINC***

Display volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL (default), *NV2* and *NINC* are ignored and all selected volumes [**VSEL**] are displayed.

***DEGEN***

Degeneracy marker:

**(blank)**

No degeneracy marker is used (default).

**DEGE**

A red star is placed on keypoints at degeneracies (see the [Modeling and Meshing Guide](#)). Not available if **/FACET**,**WIRE** is set.

***SCALE***

Scale factor for the size of the degeneracy-marker star. The scale is the size in window space (-1 to 1 in both directions) (defaults to .075).

## Notes

Displays selected volumes. (Only volumes having areas within the selected area set [**ASEL**] will be plotted.) With PowerGraphics on [**/GRAPHICS**,**POWER**], **VPLOT** will display only the currently selected areas. This command is also a utility command, valid anywhere. The degree of tessellation used to plot the volumes is set through the **/FACET** command.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>Plot Degen Volus**  
**Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>Plot Degen Volus**  
**Utility Menu>Plot>Specified Entities>Volumes**  
**Utility Menu>Plot>Volumes**

**VPTN**, *NV1, NV2, NV3, NV4, NV5, NV6, NV7, NV8, NV9*

**Partitions volumes.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*NV1, NV2, NV3, . . . , NV9*

Numbers of volumes to be operated on. If *NV1* = ALL, *NV2* to *NV9* are ignored and all selected volumes are used. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

## Notes

Partitions volumes. Generates new volumes which encompass the geometry of all the input volumes. The new volumes are defined by the regions of intersection of the input volumes, and by the complementary (non-intersecting) regions. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Partition>Volumes**

**\*VPUT**, *ParR, Entity, ENTNUM, Item1, IT1NUM, Item2, IT2NUM, KLOOP*

**Restores array parameter values into the ANSYS database.**

[APDL: Array Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### *ParR*

The name of the input vector array parameter. See **\*SET** for name restrictions. The parameter must exist as a dimensioned array [**\*DIM**] with data input.

### *Entity*

Entity keyword. Valid keywords are shown for *Entity* = in the table below.

### *ENTNUM*

The number of the entity (as shown for *ENTNUM*= in the table below).

**Item1**

The name of a particular item for the given entity. Valid items are as shown in the *Item1* columns of the table below.

**IT1NUM**

The number (or label) for the specified *Item1* (if any). Valid *IT1NUM* values are as shown in the *IT1NUM* columns of the table below. Some *Item1* labels do not require an *IT1NUM* value.

**Item2, IT2NUM**

A second set of item labels and numbers to further qualify the item for which data is to be stored. Most items do not require this level of information.

**KLOOP**

Field to be looped on:

**0 or 2 --**

Loop on the *ENTNUM* field (default).

**3 --**

Loop on the *Item1* field.

**4 --**

Loop on the *IT1NUM* field. Successive items are as shown with *IT1NUM*.

**5 --**

Loop on the *Item2* field.

**6 --**

Loop on the *IT2NUM* field. Successive items are as shown with *IT2NUM*.

## Notes

The **\*VPUT** command is not supported for PowerGraphics displays. Inconsistent results may be obtained if this command is not used in **/GRAPHICS, FULL**.

Plot and print operations entered via the GUI (**Utility Menu> Pltctrls, Utility Menu> Plot**) incorporate the **AVPRIN** command. This means that the principal and equivalent values are recalculated. If you use **\*VPUT** to put data back into the database, issue the plot commands from the command line to preserve your data.

This operation is basically the inverse of the **\*VGET** operation. Vector items are put directly (without any coordinate system transformation) into the ANSYS database. Items can only replace existing items of the database and not create new items. Degree of freedom results that are replaced in the database are available for all subsequent postprocessing operations. Other results are changed temporarily and are available mainly for the immediately following print and display operations. The vector specification **\*VCUM** does not apply to this command. The valid labels for the location fields (*Entity*, *ENTNUM*, *Item1*, and *IT1NUM*) are listed below. *Item2* and *IT2NUM* are not currently used. Not all items from the **\*VGET** list are allowed on **\*VPUT** since putting values into some locations could cause the database to be inconsistent.

This command is valid in any processor.

### Table 262 \*VPUT - POST1 Items

*Entity* = **NODE**, *ENTNUM* = *n* (node number)

Item1	IT1NUM	Description
-------	--------	-------------

Valid labels for nodal degree of freedom results are:

**Entity = NODE, ENTNUM = n (node number)**

Item1	IT1NUM	Description
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
TEMP		Temperature. For <a href="#">SHELL131</a> and <a href="#">SHELL132</a> elements with KEYOPT(3) = 0 or 1, use TBOT, TE2, TE3, . . . , TTOP instead of TEMP. Alternative get functions: TEMP(N), TBOT(N), TE2(N), etc.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity.
A	X, Y, Z	X, Y, or Z magnetic vector potential.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy (FLOTRAN).
ENDS		Turbulent energy dissipation (FLOTRAN).

Valid labels for element nodal results are:

Item1	IT1NUM	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1,2,3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1,2,3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.

**Entity = NODE, ENTNUM = n (node number)**

Item1	IT1NUM	Description
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
TG	X, Y, Z	Component thermal gradient.
TF	X, Y, Z	Component thermal flux.
PG	X, Y, Z	Component pressure gradient.
EF	X, Y, Z	Component electric field.
D	X, Y, Z	Component electric flux density.
H	X, Y, Z	Component magnetic field intensity.
B	X, Y, Z	Component magnetic flux density.
FMAG	X, Y, Z	Component electromagnetic force.

Valid labels for FLOTRAN nodal results are:

Item1	IT1NUM	Description
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.

**Entity = ELEM, ENTNUM = n (element number)**

Valid labels for element results are:

*Entity* = **NODE**, *ENTNUM* = *n* (node number)

Item1	IT1NUM	Description
ETAB	Lab	Any user-defined element table label (see <b>ETABLE</b> command).

## Menu Paths

Utility Menu>Parameters>Array Operations>Put Array Data

**VPUT**, *Par*, *IR*, *TSTRT*, *KCPLX*, *Name*

Moves an array parameter vector into a variable.

POST26: Special Purpose

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*Par*

Array parameter vector in the operation.

*IR*

Arbitrary reference number assigned to this variable (1 to *NV* [**NUMVAR**]). Overwrites any existing results for this variable.

*TSTRT*

Time (or frequency) corresponding to start of *IR* data. If between values, the nearer value is used.

*KCPLX*

Complex number key:

**0**

Use the real part of the *IR* data.

**1**

Use the imaginary part of the *IR* data.

*Name*

Thirty-two character name identifying the item on printouts and displays. Defaults to the label formed by concatenating VPUT with the reference number *IR*.

## Notes

At least one variable should be defined (**NSOL**, **ESOL**, **RFORCE**, etc.) before using this command. The starting array element number must be defined. For example, **VPUT**,A(1),2 moves array parameter A to variable 2 starting at time 0.0. Looping continues from array element A(1) with the index number incremented by one until the variable is filled. Unfilled variable locations are assigned a zero value. The number of loops may be controlled with the **\*VLEN** command (except that loop skipping (NINC) is not allowed). For multi-dimensional array parameters, only the first (row) subscript is incremented.

## Menu Paths

Main Menu>TimeHist Postpro>Table Operations>Parameter to Var

**\*VREAD**, *ParR*, *Fname*, *Ext*, *--*, *Label*, *n1*, *n2*, *n3*, *NSKIP*

**Reads data and produces an array parameter vector or matrix.**

[APDL: Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

### ***ParR***

The name of the resulting array parameter vector. See **\*SET** for name restrictions. The parameter must exist as a dimensioned array [**\*DIM**]. String arrays are limited to a maximum of 8 characters.

### ***Fname***

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

If the *Fname* field is left blank, reading continues from the current input device, such as the terminal.

### ***Ext***

Filename extension (8 character maximum).

**--**

Unused field.

### ***Label***

Can take a value of IJK, IKJ, JIK, JKI, KIJ, KJI, or blank (IJK).

### ***n1*, *n2*, *n3***

Read as  $((ParR(i,j,k), k = 1, n1), i = 1, n2), j = 1, n3)$  for *Label* = IJK. *n2* and *n3* default to 1.

### ***NSKIP***

Number of lines at the beginning of the file being read that will be skipped during the reading. Default = 0.

## Notes

Reads data from a file and fills in an array parameter vector or matrix. Data are read from a formatted file or, if the menu is off [**/MENU,OFF**] and *Fname* is blank, from the next input lines. The format of the data to be read must be input immediately following the **\*VREAD** command. The format specifies the number of fields to be read per record, the field width, and the placement of the decimal point (if none specified in the value). The read operation follows the available FORTRAN FORMAT conventions of the system (see your system FORTRAN manual). Any standard FORTRAN *real* format (such as (4F6.0), (E10.3,2X,D8.2), etc.) or alphanumeric format (A) may be used. Alphanumeric strings are limited to a maximum of 8 characters for any field (A8). For storage of string arrays greater than 8 characters, the **\*SREAD** command can be used. Integer (I) and list-directed (\*) descriptors may *not* be used. *The parentheses must be included in the format* and the format must not exceed 80 characters (including parentheses). The input line length is limited to 128 characters.

A starting array element number must be defined for the result array parameter vector (numeric or character). For example, entering these two lines:

```
*VREAD,A(1),ARRAYVAL  
(2F6.0)
```



will read two values from each line of file ARRAYVAL and assign the values to A(1), A(2), A(3), etc. Reading continues until successive row elements [**\*VLEN**, **\*VMASK**, **\*DIM**] are filled.

For an array parameter matrix, a starting array element row and column number must be defined. For example, entering these two lines:

```
VREAD,A(1,1),ARRAYVAL,,,IJK,10,2
(2F6.0)
```

will read two values from each line of file ARRAYVAL and assign the values to A(1,1), A(2,1), A(3,1), etc. Reading continues until *n1* (10) successive row elements are filled. Once the maximum row number is reached, subsequent data will be read into the next column (e.g., A(1,2), A(2,2), A(3,2), etc.)

For numerical parameters, absolute values and scale factors may be applied to the result parameter [**\*VABS**, **\*VFACT**]. Results may be cumulative [**\*VCUM**]. See the **\*VOPER** command for details. If you are in the GUI the **\*VREAD** command must be contained in an externally prepared file read into the ANSYS program (i.e., **\*USE**, **/INPUT**, etc.).

This command is not applicable to 4- or 5-D arrays.

This command is valid in any processor.

## Menu Paths

Utility Menu>Parameters>Array Parameters>Read from File

**VROTAT**, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *PAX1*, *PAX2*, *ARC*, *NSEG*

**Generates cylindrical volumes by rotating an area pattern about an axis.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NA1*, *NA2*, *NA3*, . . . , *NA6***

List of areas in the pattern to be rotated (6 maximum if using keyboard entry). Areas must lie to one side of, and in the plane of, the axis of rotation. If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NA1* = ALL, all selected areas will define the pattern to be rotated. A component name may also be substituted for *NA1*.

***PAX1*, *PAX2***

Keypoints defining the axis about which the area pattern is to be rotated.

***ARC***

Arc length (in degrees). Positive follows right-hand rule about *PAX1*-*PAX2* vector. Defaults to 360.

***NSEG***

Number of volumes (8 maximum) around circumference. Defaults to minimum required for 90° (maximum) arcs, i.e., 4 for 360°, 3 for 270°, etc.

## Notes

Generates cylindrical volumes (and their corresponding keypoints, lines, and areas) by rotating an area pattern (and its associated line and keypoint patterns) about an axis. Keypoint patterns are generated at regular angular locations (based on a maximum spacing of 90°). Line patterns are generated at the keypoint patterns.

Arc lines are also generated to connect the keypoints circumferentially. Keypoint, line, area, and volume numbers are automatically assigned (beginning with the lowest available values). Adjacent lines use a common keypoint, adjacent areas use a common line, and adjacent volumes use a common area.

To generate a single volume with an arc greater than  $180^\circ$ , *NSEG* must be greater than or equal to 2.

If element attributes have been associated with the input area via the **AATT** command, the opposite area generated by the **VROTAT** operation will also have those attributes (i.e., the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the given areas are meshed or belong to meshed volumes, the 2-D mesh can be rotated (extruded) to a 3-D mesh. See the [Modeling and Meshing Guide](#) for more information. Note that the *NDIV* argument on the **ESIZE** command should be set before extruding the meshed areas.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>About Axis**

---

**VSBA**, *NV*, *NA*, *SEPO*, *KEEPV*, *KEEPA*

**Subtracts areas from volumes.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***NV***

Volume (or volumes, if picking is used) to be subtracted from. If ALL, use all selected volumes. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NV*.

### ***NA***

Area (or areas, if picking is used) to subtract. If ALL, use all selected areas. A component name may also be substituted for *NA*.

### ***SEPO***

Behavior of the touching boundary:

#### **(blank)**

The resulting volumes will share area(s) where they touch.

#### **SEPO**

The resulting volumes will have separate, but coincident area(s) where they touch.

### ***KEEPV***

Specifies whether *NV* volumes are to be deleted:

#### **(blank)**

Use the setting of KEEP on the **BOPTN** command.

#### **DELETE**

Delete *NV* volumes after **VSBA** operation (override **BOPTN** command settings).

#### **KEEP**

Keep *NV* volumes after **VSBA** operation (override **BOPTN** command settings).

**KEEPA**

Specifies whether *NA* areas are to be deleted:

**(blank)**

Use the setting of KEEP on the **BOPTN** command.

**DELETE**

Delete *NA* areas after **VSBA** operation (override **BOPTN** command settings).

**KEEP**

Keep *NA* areas after **VSBA** operation (override **BOPTN** command settings).

## Notes

Generates new volumes by subtracting the regions common to both the volumes and areas (the intersection) from the *NV* volumes. The intersection will be an area(s). If *SEPO* is blank, the volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEPO* is set to SEPO, the volume is divided into two unconnected volumes with separate areas where they touch. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Volume by Area**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Volume by Area**

**VSBV**, *NV1*, *NV2*, *SEPO*, *KEEP1*, *KEEP2*

**Subtracts volumes from volumes.**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NV1***

Volume (or volumes, if picking is used) to be subtracted from. If ALL, use all selected volumes. Volumes specified in set *NV2* are removed from set *NV1*. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NV1*.

***NV2***

Volume (or volumes, if picking is used) to subtract. If ALL, use all selected volumes (except those included in the *NV1* argument). A component name may also be substituted for *NV2*.

***SEPO***

Behavior if the intersection of the *NV1* volumes and the *NV2* volumes is an area or areas:

**(blank)**

The resulting volumes will share area(s) where they touch.

**SEPO**

The resulting volumes will have separate, but coincident area(s) where they touch.

***KEEP1***

Specifies whether *NV1* volumes are to be deleted:

**(blank)**

Use the setting of *KEEP* on the **BOPTN** command.

**DELETE**

Delete *NV1* volumes after **VSBV** operation (override **BOPTN** command settings).

**KEEP**

Keep *NV1* volumes after **VSBV** operation (override **BOPTN** command settings).

**KEEP2**

Specifies whether *NV2* volumes are to be deleted:

**(blank)**

Use the setting of *KEEP* on the **BOPTN** command.

**DELETE**

Delete *NV2* volumes after **VSBV** operation (override **BOPTN** command settings).

**KEEP**

Keep *NV2* volumes after **VSBV** operation (override **BOPTN** command settings).

## Notes

Generates new volumes by subtracting the regions common to both *NV1* and *NV2* volumes (the intersection) from the *NV1* volumes. The intersection can be an volume(s) or area(s). If the intersection is an area and *SEPO* is blank, the *NV1* volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEPO* is set to *SEPO*, *NV1* is divided into two unconnected volumes with separate areas where they touch. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **VSBV,ALL,ALL** will have no effect because all the volumes in set *NV1* will have been moved to set *NV2*.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Volumes**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Volumes**

**Main Menu>Preprocessor>Modeling>Operate>Subtract>Volumes**

---

## **VSBW**, *NV*, *SEPO*, *KEEP*

**Subtracts intersection of the working plane from volumes (divides volumes).**

[PREP7: Booleans](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***NV***

Volume (or volumes, if picking is used) to be subtracted from. If *NV* = ALL, use all selected volumes. If *NV* = P, graphical picking is enabled (valid only in the GUI). A component name may also be input for *NV*.

***SEPO***

Behavior of the created boundary.

**(blank)**

The resulting volumes will share area(s) where they touch.

**SEPO**

The resulting volumes will have separate, but coincident area(s).

**KEEP**

Specifies whether *NV* volumes are to be deleted.

**(blank)**

Use the setting of *KEEP* on the **BOPTN** command.

**DELETE**

Delete *NV* volumes after **VSBW** operation (override **BOPTN** command settings).

**KEEP**

Keep *NV* volumes after **VSBW** operation (override **BOPTN** command settings).

## Notes

Generates new volumes by subtracting the intersection of the working plane from the *NV* volumes. The intersection will be an area(s). If *SEPO* is blank, the volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEPO* is set to SEPO, the volume is divided into two unconnected volumes with separate areas. The SEPO option may cause unintended consequences if any keypoints exist along the cut plane. See the [Modeling and Meshing Guide](#) for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Issuing the **VSBW** command under certain conditions may generate a topological degeneracy error. Do not issue the command if:

- A sphere or cylinder has been scaled. (A cylinder must be scaled unevenly in the XY plane.)
- A sphere or cylinder has not been scaled but the work plane has been rotated.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Volu by WrkPlane**

**Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Volu by WrkPlane**

**Main Menu>Preprocessor>Modeling>Operate>Divide>Volu by WrkPlane**

## /VSCALE, *WN*, *VRATIO*, *KEY*

Scales the length of displayed vectors.

GRAPHICS: Scaling

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***WN***

Window number (or ALL) to which command applies (defaults to 1).

***VRATIO***

Ratio value applied to the automatically calculated scale factor (defaults to 1.0, i.e., use scale factor as automatically calculated).

**KEY**

Relative scaling key:

**0**

Use relative length scaling among vectors based on magnitudes.

**1**

Use uniform length scaling for all vector lengths.

## Command Default

Automatic vector scaling.

## Notes

Allows scaling of the vector length displayed with the **PLVECT** command of POST1 and the **/PBC** and **/PSF** commands. Also allows the scaling of the element (i.e., **/PSYMB,ESYS**) and the nodal (i.e., **/PSYMB,NDIR**) coordinate system symbols.

This command is valid in any processor.

## Menu Paths

**Main Menu>General Postproc>Plot Results>Vector Plot>Predefined**  
**Utility Menu>Plot>Results>Vector Plot**  
**Utility Menu>PlotCtrls>Multi-Plot Contrls**  
**Utility Menu>PlotCtrls>Style>Vector Arrow Scaling**

---

**\*VSCFUN**, *ParR*, *Func*, *Par1*

**Determines properties of an array parameter.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

**ParR**

The name of the resulting scalar parameter. See **\*SET** for name restrictions.

**Func**

Functions:

**MAX --**

Maximum: the maximum *Par1* array element value.

**MIN --**

Minimum: the minimum *Par1* array element value.

**LMAX --**

Index location of the maximum *Par1* array element value. Array *Par1* is searched starting from its specified index.

**LMIN --**

Index location of the minimum *Par1* array element value. Array *Par1* is searched starting from its specified index.

**FIRST --**

Index location of the first nonzero value in array *Par1*. Array *Par1* is searched starting from its specified index.

**LAST --**

Index location of the last nonzero value in array *Par1*. Array *Par1* is searched starting from its specified index.

**SUM --**

Sum: *Par1* (the summation of the *Par1* array element values).

**MEDI --**

Median: value of *Par1* at which there are an equal number of values above and below.

**MEAN --**

Mean:  $(\sigma \text{ Par1})/\text{NUM}$ , where NUM is the number of summed values.

**VARI --**

Variance:  $(\sigma ((\text{Par1}-\text{MEAN})^{**2}))/\text{NUM}$ .

**STDV --**

Standard deviation: square root of VARI.

**RMS --**

Root-mean-square: square root of  $(\sigma (\text{Par1}^{**2}))/\text{NUM}$ .

**NUM --**

Number: the number of summed values (masked values are not counted).

***Par1***

Array parameter vector in the operation.

## Notes

Operates on one input array parameter vector and produces one output scalar parameter according to:

$$\text{ParR} = f(\text{Par1})$$

where the functions (f) are described below. The starting array element number must be defined for the array parameter vector. For example, \*VSCFUN,MU,MEAN,A(1) finds the mean of the A vector values, starting from the first value and stores the result as parameter MU. Operations use successive array elements [\*VLEN, \*VMASK] with the default being all successive array elements. Absolute values and scale factors may be applied to all parameters [\*VABS, \*VFACT]. Results may be cumulative [\*VCUM]. See the \*VOPER command for details.

This command is valid in any processor.

## Menu Paths

**Utility Menu>Parameters>Array Operations>Vector-Scalar Func**

**VSEL**, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP***Selects a subset of volumes.**

DATABASE: Selecting

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**Type**

Label identifying the type of volume select:

**S**

Select a new set (default).

**R**

Reselect a set from the current set.

**A**

Additionally select a set and extend the current set.

**U**

Unselect a set from the current set.

**ALL**

Restore the full set.

**NONE**

Unselect the full set.

**INVE**

Invert the current set (selected becomes unselected and vice versa).

**STAT**

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:**Item**Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to VOLU.**Comp**

Component of the item (if required). Valid component labels are shown in the table below.

**VMIN**Minimum value of item range. Ranges are volume numbers, coordinate values, attribute numbers, etc., as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, or ESYS and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **VLIST** command for a discussion of signed attributes.**VMAX**Maximum value of item range. *VMAX* defaults to *VMIN*.**VINC**Value increment within range. Used only with integer ranges (such as for volume numbers). Defaults to 1. *VINC* cannot be negative.**KSWP**

Specifies whether only volumes are to be selected:



**0**

Select volumes only.

**1**Select volumes, as well as keypoints, lines, areas, nodes, and elements associated with selected volumes. Valid only with *Type* = S.

## Command Default

All volumes are selected.

## Notes

Selects volumes based on values of a labeled item and component. For example, to select a new set of volumes based on volume numbers 1 through 7, use **VSEL,S,VOLU,,1,7**. The subset is used when the ALL label is entered (or implied) on other commands, such as **VLIST,ALL**. Only data identified by volume number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

This command is valid in any processor.

For Selects based on non-integer numbers (coordinates, results, etc.), items that are within the range  $VMIN - Toler$  and  $VMAX + Toler$  are selected. The default tolerance *Toler* is based on the relative values of VMIN and VMAX as follows:

- If  $VMIN = VMAX$ ,  $Toler = 0.005 \times VMIN$ .
- If  $VMIN = VMAX = 0.0$ ,  $Toler = 1.0E-6$ .
- If  $VMAX \neq VMIN$ ,  $Toler = 1.0E-8 \times (VMAX - VMIN)$ .

Use the **SELTOL** command to override this default and specify *Toler* explicitly.

**Table 263 VSEL - Valid Item and Component Labels**

VSEL <i>Type, Item, Comp, VMIN, VMAX, VINC, KABS</i>		
Item	Comp	Description
VOLU		Volume number.
LOC	X, Y, Z	X, Y, or Z center (picking "hot spot" location in the active coordinate system).
MAT		Material number associated with the volume.
TYPE		Element type number associated with the volume.
REAL		Real constant set number associated with the volume.
ESYS		Element coordinate system associated with the volume.

## Menu Paths

**Utility Menu>Select>Entities**

**VSLA**, *Type*, *VLKEY*

Selects those volumes containing the selected areas.

[DATABASE: Selecting](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Type**

Label identifying the type of volume select:

- S**  
Select a new set (default).
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

**VLKEY**

Specifies whether all contained volume areas must be selected [[ASEL](#)]:

- 0**  
Select volume if any of its areas are in the selected area set.
- 1**  
Select volume only if all of its areas are in the selected area set.

**Notes**

This command is valid in any processor.

**Menu Paths**

Utility Menu>Select>Entities

---

**\*VSTAT**

Lists the current specifications for the array parameters.

[APDL: Array Parameters](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**Notes**

Lists the current specifications for the **\*VABS**, **\*VCOL**, **\*VCUM**, **\*VFACT**, **\*VLEN**, and **\*VMASK** commands.

This command is valid in any processor.

**Menu Paths**

This command cannot be accessed from a menu.

---

**VSUM**, *LAB*

**Calculates and prints geometry statistics of the selected volumes.**

[PREP7: Volumes](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**LAB**

Controls the degree of tessellation used in the calculation of area properties. If *LAB* = DEFAULT, area calculations will use the degree of tessellation set through the **/FACET** command. If *LAB* = FINE, area calculations are based on a finer tessellation.

**Notes**

Calculates and prints geometry statistics (volume, centroid location, moments of inertia, etc.) associated with the selected volumes. Geometry items are reported in the global Cartesian coordinate system. A unit density is assumed unless the volumes have a material association via the **VATT** command. Items calculated by **VSUM** and later retrieved by a **\*GET** or **\*VGET** command are valid only if the model is not modified after the **VSUM** command is issued.

Setting a finer degree of tessellation will provide area calculations with greater accuracy, especially for thin, hollow models. However, using a finer degree of tessellation requires longer processing.

For very thin volumes, such that the ratio of the minimum to the maximum dimension is less than 0.01, the **VSUM** command can provide erroneous volume information. To ensure that such calculations are accurate, make certain that you subdivide such volumes so that the ratio of the minimum to the maximum is at least 0.05.

**Menu Paths**

**Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Volumes**

**VSWEEP**, *VNUM*, *SRCA*, *TRGA*, *LSMO*

**Fills an existing unmeshed volume with elements by sweeping the mesh from an adjacent area through the volume.**

[PREP7: Meshing](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**VNUM**

Number identifying the volume that is to be meshed by **VSWEEP**. If *VNUM* = P, graphical picking is enabled, you will be prompted to choose the volume or volumes based on the setting of **EXTOPT**, **VSWE**, **AUTO**. This argument is required.

"ALL" is a valid input value that when selected sends all the selected volumes to the sweeper. If *VNUM* = ALL, each volume that can be swept will be and those not able to be swept will be unmeshed or meshed with tets depending upon the setting of **EXTOPT**, **VSWE**, **TETS**.

A component name is a valid input value. All volumes that are part of the specified component will be sent to the sweeper.

**SRCA**

Number identifying the source area. This is the area whose mesh will provide the pattern for the volume elements. (If you do not mesh the source area prior to volume sweeping, ANSYS meshes it internally when you initiate volume sweeping.) ANSYS sweeps the pattern of the area elements through the volume to create the mesh of volume elements. You cannot substitute a component name for *SRCA*.

This argument is optional. If *VNUM* = ALL or is a component containing more than one volume, *SRCA* is ignored. If *SRCA* is not provided or if it is ignored, **VSWEEP** attempts to automatically determine which area should be the target area.

**TRGA**

Number identifying the target area. This is the area that is opposite the source area specified by *SRCA*. You cannot substitute a component name for *TRGA*.

This argument is optional. If *VNUM* = ALL or component containing more than one volume, *TRGA* is ignored. If *TRGA* is not provided or if it is ignored, **VSWEEP** attempts to automatically determine which area should be the target area.

**LSMO**

Value specifying whether ANSYS should perform line smoothing during volume sweeping. (The value of this argument controls line smoothing for the **VSWEEP** command only; it has no effect on the setting of the **MOPT** command's LSMO option.) This argument is optional.

**0**

Do not perform line smoothing. This is the default.

**1**

Always perform line smoothing. This setting is not recommended for large models due to speed considerations.

## Notes

If the source mesh consists of quadrilateral elements, ANSYS fills the volume with hexahedral elements. If the source mesh consists of triangles, ANSYS fills the volume with wedges. If the source mesh consists of a combination of quadrilaterals and triangles, ANSYS fills the volume with a combination of hexahedral and wedge elements.

In the past, you may have used the **VROTAT**, **VEXT**, **VOFFST**, and/or **VDRAG** commands to extrude a meshed area into a meshed volume. However, those commands create the volume and the volume mesh simultaneously. In contrast, the **VSWEEP** command is intended for use in an existing unmeshed volume. This makes **VSWEEP** particularly useful when you have imported a solid model that was created in another program, and you want to mesh it in ANSYS.

For related information, see the description of the **EXTOPT** command (although **EXTOPT** sets volume sweeping options, it does not affect element spacing). Also see the detailed discussion of volume sweeping in [Meshing Your Solid Model](#) of the *Modeling and Meshing Guide*.

## Menu Paths

**Main Menu>Preprocessor>Meshing>Mesh>Volume Sweep>Sweep**

---

**VSYMM**, *Ncomp*, *NV1*, *NV2*, *NINC*, *KINC*, *NOELEM*, *IMOVE***Generates volumes from a volume pattern by symmetry reflection.**

PREP7: Volumes

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

***Ncomp***

Symmetry key:

**X**

X symmetry (default).

**Y**

Y symmetry.

**Z**

Z symmetry.

***NV1*, *NV2*, *NINC***

Reflect volumes from pattern beginning with *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is all selected volumes [VSEL]. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

***KINC***

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [NUMSTR].

***NOELEM***

Specifies whether nodes and elements are also to be generated:

**0**

Generate nodes and elements associated with the original volumes, if they exist.

**1**

Do not generate nodes and elements.

***IMOVE***

Specifies whether volumes will be moved or newly defined:

**0**

Generate additional volumes.

**1**

Move original volumes to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Corresponding meshed items are also moved if not needed at their original position.

**Notes**

Generates a reflected set of volumes (and their corresponding keypoints, lines, areas and mesh) from a given volume pattern by a symmetry reflection (see analogous node symmetry command, **NSYM**). The MAT, TYPE, REAL, and ESYS attributes are based upon the volumes in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. The active coordinate system must be a Cartesian system. Volumes in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Volumes are generated as described in the **VGEN** command.

See the **ESYM** command for additional information about symmetry elements.

## Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Volumes

---

## /VT

Enters the Variational Technology preprocessor.

SESSION: Processor Entry  
 VARIATIONAL TECHNOLOGY: DesignXplorer  
 <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

## Notes

Enters the Variational Technology processor.

## Menu Paths

Main Menu>DesignXplorer>Solution>Solve

---

## VTCLR, *Type*

Clears the Variational Technology database.

VARIATIONAL TECHNOLOGY: DesignXplorer  
 <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

### *Type*

Specifies the part of the Variational Technology database to be cleared. Valid labels are:

#### **ALL**

Clears the entire Variational Technology database. Both the preprocessing part (settings for input variables, result parameters, etc.) and results part (results for the derivatives) are cleared.

#### **RSLT**

Clear only the results part of the Variational Technology database. This is necessary if you want to modify the Variational Technology model (for example, adding a new input variable and so on) after a solution has already been performed.

## Notes

Clear the database of the Variational Technology module. The settings are reset to their default values and the memory is cleared. Remember that the result files containing the results of Variational Technology (see **VTRFIL** command) are never deleted unless you intentionally delete them. We recommend that you use this command before switching to a new solution, which, for example, includes different input variables or different element or node components for input variables. See the various Variational Technology commands for changes that are possible after the solution has been performed.

If the entire database is cleared with **VTCLR,ALL** then also all definitions for input variables, result parameters, solution method, result file storage and so on are either deleted and/or reset to their default values.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>DesignXplorer>Other>Clear Database

---

**VTDISC**, *Name*, *ElComp*

Defines an element component as a discrete input variable for the DesignXplorer.

VARIATIONAL TECHNOLOGY:DesignXplorer  
<> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

### *Name*

Name for the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks are allowed in the string.

### *ElComp*

The name of either of an element component or the number of a single element.

## Command Default

None

## Notes

Defines a component containing a group of elements as a discrete input variable of the DesignXplorer. In a discrete optimization the DesignXplorer will evaluate the dependency of the result parameter(s) (see the **VT<sub>RS</sub>LT** command) as a function of the discrete case of either including the group of elements in the finite-element model or excluding the group of elements from the model.

The *ElComp* field defines a group of elements that are considered as either being included or excluded from the finite-element model during a discrete optimization.

If a solution has already been performed with Variational Technology using **STA<sub>OPT</sub>**,VT and the **SOLVE** command, then the Variational Technology database is locked. In this case no input variables can be added to the model. You cannot change the specific definitions of an existing input variable. If you want to modify the Variational Technology model by adding a discrete variable, you first have to unlock the model with the **VT<sub>CLR</sub>** command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>DesignXplorer>Setup>Discrete Variable

---

**VTEVAL**, --, *Mode*

Triggers evaluation of generated results based on input variables specified via the **VTVMOD** command.

VARIATIONAL TECHNOLOGY: DesignXplorer  
<> <> <> <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

--

Unused field.

**Mode**

The mode number for normal modes. By default, driven by the application.

**Command Default**

Defaults as outlined above.

**Notes**

Used in conjunction with the **VTVMOD** command, **VTEVAL** evaluates the designated variables and loads the results into the ANSYS database. You can employ the ANSYS postprocessing tools to view the results, or perform an optimization or PDS loop.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

Main Menu> DesignXplorer> Postprocessing> Evaluate Results

**VTFREQ**, *Name, MIN, MAX, INC, RedOpt*

Defines the frequency as input variable for the harmonic sweep functionality of VT Accelerator.

VARIATIONAL TECHNOLOGY: DesignXplorer  
VARIATIONAL TECHNOLOGY: VT Harmonic Sweep  
<> <> <> <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

**Name**

Name of the variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be included in the string.

**MIN**

Minimum value of the input variable.

**MAX**

Maximum value of the input variable.

**INC**

Number of increments within the range of the input variable as defined by the *MIN* and *MAX* fields.

**RedOpt**

Reduction option that specifies what to do if the DesignXplorer needs to reduce the variation range from *MIN* to *MAX*. Valid labels are:



**CONT**

Continue with the calculation of the derivatives. This is the default.

**STOP**

Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

## Command Default

There are no default values for the command fields.

## Notes

Defines the frequency as an input variable. Variational Technology will evaluate a frequency sweep from the specified minimum to the maximum frequency values given in the *MIN* and *MAX* fields. The frequency sweep will include as many frequency increments as specified in the *INC* field.

If a solution has already been performed with Variational Technology using **STAOPT**, **VT** and the **SOLVE** command, then the Variational Technology database is locked. In this case no input variables can be added to the model. You cannot change the specific definitions of an existing input variable. If you want to modify the Variational Technology model by adding a discrete variable, you first have to unlock the model with the **VTCLR** command.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>DesignXplorer>Setup>Frequency

**VTGEOM**, *Name*, *MIN*, *MAX*, *RedOpt*, *Order*

**Defines a geometry parameter created with ANSYS MeshMorpher as a DesignXplorer input variable.**

VARIATIONAL TECHNOLOGY:DesignXplorer

<> <> <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

**Name**

Name of an input variable. This is a string of up to 256 characters used for postprocessing purposes. It can have blanks included in the string.

**MIN**

Minimum value of the input variable. By default, driven by the application.

**MAX**

Maximum value of the input variable. By default, driven by the application.

**RedOpt**

Reduction option that specifies what to do if the DesignXplorer needs to reduce the variation range from *MIN* to *MAX*. Valid labels are;

**CONT**

Continue with the calculation of the derivatives. This is the default.

**STOP**

Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

**Order**

Deviation Order. By default, driven by the application.

## Command Default

Defaults as outlined above.

## Notes

If you do not use ANSYS MeshMorpher, you can ignore the **VTGEOM** command.

The **VTGEOM** command allows you to define a geometry parameter created with ANSYS MeshMorpher as an input variable of the DesignXplorer.

The filename of the ANSYS MeshMorpher database has to be defined using the **VTRFIL** command. DesignXplorer uses the same RSX file to store its results. If the RSX file does not exist or does not contain the expected geometry parameters and associated morphed mesh, then the **VTGEOM** command is ignored.

In the general case, the fields *MIN* and *MAX* should not be used because a range of variation is already defined in ANSYS MeshMorpher. Nevertheless, the *MIN* and *MAX* fields allow to define a smaller range of variation to compute the VT results. Although it is not recommended, it is also possible to define a larger range of variation. In this case, a warning message is always generated.

The *MIN* and *MAX* field must always be specified or left blank together. If only one field is specified, then the **VTGEOM** command is ignored.

If the Order is not specified, it will be increased until the accuracy requirements of the **VTRSLT** command are satisfied. When you set a value for *Order*, no further adjustments for accuracy will be performed. Order should only be used when an input parameter has an influence with a limited order.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**This command cannot be accessed from a menu.**

---

**VTIN**, *Name*, *MIN*, *MAX*, *RedOpt*, *Lab*, *Comp*, *--*, *VarType*, *Order*

**Defines an inertial load as an input variable for DesignXplorer.**

VARIATIONAL TECHNOLOGY: DesignXplorer

<> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

**Name**

Name for the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be included in the string.

**MIN**

Minimum value of the input variable.

**MAX**

Maximum value of the input variable.

**RedOpt**

Reduction option that specifies what to do if DesignXplorer needs to reduce the variation range from *MIN* to *MAX*. Valid labels are:

**CONT**

Continue with the calculation of the derivatives. This is the default.

**STOP**

Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

**Lab**

Valid inertial load label. This label must be defined for the inertial load using the ANSYS command of the same name prior to issuing this command. Applicable labels are:

**ACEL**

Linear acceleration.

**CGOMGA**

Rotational velocity of the global origin.

**DCGOMG**

Rotational acceleration of the global origin.

**DOMEGA**

Rotational acceleration of the structure.

**OMEGA**

Rotational velocity.

--

Unused field.

**Comp**

X, Y, Z or ALL

If *Comp* = ALL, the *VarType* must be FACT.

**VarType**

Label indicating how the variability of vectors should be handled, when *COMP* = ALL is used. Valid labels are:

**FACT**

The variability of vector is described by a scaling factor that is applied to all 3 vectors components. This means that the *MIN* field is the value of the minimum factor to be applied and the *MAX* field is the maximum factor to be applied.

**ADD**

The variability of vector components is described by an add-on that is applied to vector components. This means that the *MIN* field is the minimum value that is added to the vector components and the *MAX* field contains the maximum value that is added to the vector components. As such the values specified for *MIN* and *MAX* have the same unit as the vector itself.

**VAL**

The variability of the vectors is described by the variability of the vector components itself. This is only applicable if ALL has not been used. This means that the *MIN* field is the minimum value that

the vector component itself can take and the *MAX* field contains the maximum value, which the vector component can take. This is the default.

**Order**

Derivation order. By default, driven by the application.

**Notes**

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**This command cannot be accessed from a menu.**

---

**VTMETH**, *SoluType*, *ApprType*, *ModeTrack*

**Defines the solution options for the DesignXplorer.**

VARIATIONAL TECHNOLOGY:DesignXplorer  
<> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

**SoluType**

Specifies the type of solution that needs to be evaluated during the solution run of the DesignXplorer. Valid labels are:

**INDP**

Builds the solution approximation assuming that the input variables are independent. Therefore, mixed input variables interactions will be neglected.

**FULL**

Builds the full solution approximation, addressing the interactions between the input variables (if necessary). This option is the default.

**ApprType**

Specifies the type of approximation function that is used to build the DesignXplorer. Valid labels are:

**ROMS**

Uses a Reduced Order Model Sweep approximation. This method is the more accurate, it supports multiple input variables per element, continuous as well as discrete input variables. However it is limited to linear static analysis (structural and thermal), and cannot accept more than 10 continuous input variables or 20 discrete input variables. Continuous and Discrete input variables cannot be mixed.

**PADE**

Uses a series expansion based on Pade approximations using rational functions. This method is more accurate than TAYLOR, however it is limited to a single input variable per element and does not support geometry input variables.

**TAYL**

Uses a Taylor series expansion as the approximation function. This method supports multiple input variables per element, i.e. Young's Modulus and thickness, geometry input variables, linear static and normal modes analysis. However, Discrete input variables is not supported.

**AUTO**

This option is the default and allows DesignXplorer to determine automatically the most appropriate method to use. This is the recommended option.

**ModeTrack**

Specifies whether or not mode tracking should be in a modal solution of the DesignXplorer. Valid labels are:

**YES**

Mode Tracking will be used in a modal solution. This option is the default.

**NO**

Mode Tracking will not be used in a modal solution.

## Command Default

Defaults as outlined above.

## Notes

Determines which approximation function of the result parameters with respect to the input variables shall be calculated during the DesignXplorer solution run.

The default ApprType=AUTO is recommended. It selects the ROMS method each time it is possible, or the most appropriate method in case ROMS does not apply to the model.

If Variational Technology is not used in connection with a modal analysis, then the ModeTrack option is ignored.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>DesignXplorer>Solution>Solution Method**

**VTMP**, *Name*, *MIN*, *MAX*, *RedOpt*, *Lab*, *MAT*, *ElComp*, *VarType*, *Order*

**Defines a material property as an input variable for DesignXplorer.**

VARIATIONAL TECHNOLOGY: DesignXplorer

<> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

**Name**

Name for the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be included in the string.

**MIN**

Minimum value of the input variable.

**MAX**

Maximum value of the input variable.

**RedOpt**

Reduction option that specifies what to do if DesignXplorer needs to reduce the variation range from *MIN* to *MAX*. Valid labels are:

**CONT**

Continue with the calculation of the derivatives. This is the default.

**STOP**

Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

**Lab**

Valid material property label. This label must be defined for the material number *MAT* using the **MP** or **MPDATA** command prior to issuing this command. Applicable labels are:

**EX**

Elastic moduli (also EY, EZ).

**NUXY**

Minor Poisson's ratios (also NUYZ, NUXZ).

**GXY**

Shear moduli (also GYZ, GXZ).

**DENS**

Material density

**KXX**

Conductivity (only in thermal analysis), also KYY, KZZ

**ALPX**

Thermal Expansion, also ALPY, ALPZ

**MAT**

Material identification number

**ElComp**

Name of an element component or number of a single individual element. If *ElComp* = ALL, then all elements are considered for the **VTMP** command.

**VarType**

Label indicating how the variability of temperature dependent material properties should be handled. This is particularly useful to describe how temperature dependent material properties are handled. At this release, only one label (the default) is valid:

**FACT**

The variability of the material property is described by a scale factor that is applied to the material property values. In the case of temperature dependent material properties, a scale factor will be applied to all defined material property values. This means that the *MIN* field is the value of the minimum factor to be applied and the *MAX* field is the maximum factor to be applied. As such the values specified for *MIN* and *MAX* are unitless quantities. *VarType* = FACT is the default for temperature dependent material properties.

**ADD**

The variability of the material property is described by an add-on that is applied to the material property values. In the case of a temperature dependent material property, the defined property values are shifted all at the same time. This means that the *MIN* field is the minimum value that is added to the material property values and the *MAX* field contains the maximum value that is added to the material property values. Hence, the values specified for *MIN* and *MAX* have the same unit as the material property itself.

**VAL**

The variability of the material property is described by the variability of the material property value itself. This is only applicable if the material property is *not* temperature dependent. This means that the *MIN* field is the minimum value that the material property itself can take and the *MAX* field contains the maximum value which the material property values can take. This is the default if the material properties are *not* temperature dependent.

**Order**

Derivation order. By default, driven by the application.

**Command Default**

As described above.

**Notes**

Defines a material property of a single element or a set of elements as an input variable of the DesignXplorer. The DesignXplorer will evaluate the derivatives of the result parameter(s) (see **VTRSLT** command) with respect to the material property defined here. The derivatives are used to determine an approximation function for the result parameter(s) as a function of the material property in the range specified by the *MIN* and *MAX* fields.

The type of the material property is defined by the Lab field, which must not be left blank. The material property must exist or be specified using the **MP** or **MPDATA** command prior to issuing the **VTMP** command.

Using either the *MAT* field or the *ELComp* field a set of element is selected for which the material property is varied. The user must define one of them and leave the respective other one blank. All selected elements must be associated with the same material number. This is of course always given in the *MAT* field. If selected by the *ELComp* field, an error will be issued if an element is found that is associated with a material number that is different from the one associated with any other element in the element component.

If the material properties are temperature dependent, then the temperature dependency is defined by either several data points at different temperatures (**MP**) or by coefficients defining the temperature dependent polynomial (**MPDATA**). To describe a variation of the material property with one single parameter is only possible by using a scalar addition to the temperature dependency curve (*VarType* = ADD) or by using a scalar multiplication factor for the entire temperature dependency curve (*VarType* = FACT). Varying the material property value itself using *VarType* = VAL is not possible, if the material property is temperature dependent.

An error message is generated and the **VTMP** command is ignored when the current value is outside the variation range.

If the Order is not specified, it will be increased until the accuracy requirements of the **VTRSLT** command are satisfied. When you set a value for Order, no further adjustments for accuracy will be performed. Order should only be used when an input parameter has an influence with a limited order.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>DesignXplorer>Setup>Material Property**

**VTOP**, *Oper*, *Name*, *Value*

**Defines options value for the DesignXplorer.**

VARIATIONAL TECHNOLOGY: DesignXplorer

<> <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

**Oper**

Label for the type of operation that is performed. Valid labels are:

**SET**

Add an option (or if it already exists, modify its value).

**DELETE**

Delete an option.

**RESET**

Reset an option to its default value. If the option name is not specified, this applies to all options.

**Name**

Name of the option to define or modify. If this field is left blank, the command applies to all options. Valid labels are listed in the table below.

**Value**

If *Oper*=SET, this is the value assigned to the option. Valid labels and values for each option are listed in the table below.

**Notes**

Table of valid *Name* and *Value* labels.

Name	Option Description	Value	Value Description
VERBOSITY	Controls the level of details of the output messages.	PUBLIC	Normal (default)
		DETAILED	Detailed messages
		VERBOSE	Maximum verbosity
OPT_FAST_PerformInterpol	Activates an additional interpolation phase at the end of the ROMS Variational solve in order to provide fast post-processing evaluations.	0	Interpolation deactivated (default)
		1	Interpolation activated
OUTOFCORE	Activates/deactivates the out-of-core postprocessing mode to evaluate large results databases using less memory. Valid for TAYLOR and ROMS approximation types.	0	Out-of-core is deactivated (default).
		1	Out-of-core is activated.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**This command cannot be accessed from a menu.**



## VTPOST

Launches the DesignXplorer postprocessing application.

VARIATIONAL TECHNOLOGY: DesignXplorer  
<> <> <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

### Notes

The postprocessing application provides a viewer for Variational Technology results files. You can only launch this application if the ANSYS GUI is active; that is, it cannot be launched from batch mode.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Menu Paths

Main Menu>DesignXplorer>Postprocessing>SolutionViewer

**VTRAN**, *KCNTO*, *NV1*, *NV2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Transfers a pattern of volumes to another coordinate system.

PREP7: Volumes

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

#### *KCNTO*

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system. The coordinate system type and parameters of *KCNTO* must be the same as the active system.

#### *NV1*, *NV2*, *NINC*

Transfer volumes from pattern beginning with *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is all selected volumes [VSEL]. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

#### *KINC*

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [NUMSTR].

#### *NOELEM*

Specifies whether elements and nodes are also to be generated:

0

Generate nodes and elements associated with the original volumes, if they exist.

1

Do not generate nodes and elements.

#### *IMOVE*

Specifies whether to redefine the existing volumes:

0

Generate additional volumes.

1

Move original volumes to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Corresponding meshed items are also moved if not needed at their original position.

## Notes

Transfers a pattern of volumes (and their corresponding keypoints, lines, areas and mesh) from one coordinate system to another (see analogous node transfer command, **TRANSFER**). The *MAT*, *TYPE*, *REAL*, and *ESYS* attributes are based upon the volumes in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Coordinate and slope values are interpreted in the active coordinate system and are transferred directly. Volumes are generated as described in the **VGEN** command.

## Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Volumes

**VTREAL**, *Name*, *MIN*, *MAX*, *RedOpt*, *Lab*, *NSET*, *EIComp*, *VarType*, *Order*

Defines a real constant property as an input variable for the DesignXplorer.

VARIATIONAL TECHNOLOGY: DesignXplorer

<> <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

### Name

Name of the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks are allowed in the string.

### MIN

Minimum value of the input variable.

### MAX

Maximum value of the input variable.

### RedOpt

Reduction option that specifies what to do if the DesignXplorer needs to reduce the variation range from *MIN* to *MAX*. Valid labels are:

#### CONT

Continue with the calculation of the derivatives. This is the default.

#### STOP

Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

### Lab

Valid real constant label. This label must refer to a valid property of the real constant set as defined in the **R** command. Applicable labels are:

#### TK

Shell thickness. This applies only to shell elements of type **SHELL181**, where the element thickness has been defined with the **R** command.

**STIF**

Spring stiffness. This applies only to combination elements of type **COMBIN14**, where the spring stiffness has been defined with the **R** command.

**MASS**

Lumped mass. This applies only to structural mass elements of type **MASS21** with **KEYOPT(3) = 2**, where the lumped mass has been defined with the **R** command.

**NSET**

Real constant set identification number as defined by the **R** command.

**ELComp**

Name of an element component or number of a single individual element. If *ELComp* = **ALL**, then all elements are considered for the **VTREAL** command.

**VarType**

Label indicating how the variability of a spatially dependent element thickness should be handled. This label is ignored if the shell element thickness is constant for all elements selected by either the **NSET** or the *ELComp* field. Valid labels are:

**FACT**

The variability of the section property is described by a scale factor that is applied to the section property values. In case the section properties are different from element to element, a scale factor will be applied to all section property values. This means that the *MIN* field is the value of the minimum factor to be applied and the *MAX* field is the maximum factor to be applied. As such the values specified for *MIN* and *MAX* are unitless quantities. *VarType* = **FACT** is the default if the section property is different from element to element.

**ADD**

The variability of the section property is described by an add-on that is applied to the section property values. In case the section properties are different from element to element, then the section property values are shifted at the same time by this add-on. This means that the *MIN* field is the minimum value that is added to the section property values and the *MAX* field contains the maximum value that is added to the property values. Hence, the values specified for *MIN* and *MAX* have the physical unit of a length.

**VAL**

The variability of the section property is described by the modification of the section property value itself. This is only applicable if the section property value is identical and constant for all selected elements. This means that the *MIN* field is the minimum value that the section property value itself can take and the *MAX* field contains the maximum value, which the section property value can take. This is the default if the section property is identical and constant from element to element.

**Order**

Derivation order. By default, driven by the application.

**Notes**

Defines the real constant property of a single element or a set of elements as an input variable of the DesignXplorer. The real constant property must be specified with the **R** command prior to the **VTREAL** command. The DesignXplorer will evaluate the derivatives of the result parameter(s) (see **VTRSLT** command) with respect to the section property defined here. The derivatives are used to determine an approximation function for the result parameter(s) as a function of the section property in the range specified by the *MIN* and *MAX* fields.

Using either the *NSET* field or the *ElComp* field a set of elements is selected for which the real constant property is varied. The user must define one of these two fields and leave the respective other one blank.

If the real constant property is different from element to element, then each element has different values for the real constant properties at the individual nodes. To describe a variation of the real constant property with one single parameter is only possible by using a scalar addition to all real constant property values (*VarType* = ADD) or by using a scalar multiplication factor for all real constant property values (*VarType* = FACT). Varying the real constant property by its value itself using *VarType* = VAL is only possible if the element thickness is constant and identical for all elements.

An error message is generated and the **VTREAL** command is ignored when the current value is outside the variation range.

If the Order is not specified, it will be increased until the accuracy requirements of the **VTRSLT** command are satisfied. When you set a value for Order, no further adjustments for accuracy will be performed. Order should only be used when an input parameter has an influence with a limited order.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>DesignXplorer>Setup>Real Constant

**VTRFIL**, *Fname*, *Ext*, *Dir*

Specifies the file to which DesignXplorer results are written.

VARIATIONAL TECHNOLOGY: DesignXplorer  
<> <> <> <> <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

### *Fname*

File name (32 characters maximum). Defaults to Jobname .

### *Ext*

File name extension (8 characters maximum). Defaults to *rsx* if *Fname* is blank; otherwise, no default.

### *Dir*

Directory name (64 characters maximum). Defaults to the current working directory.

## Command Default

*Fname* = Jobname, *Ext* = *rsx*, the default directory is the current working directory.

## Notes

Specifies which file is used to store the results of DesignXplorer. The file contains the description of the higher order derivatives of the result parameters (selected by the **VTRSLT** command) with respect to the input variables (selected by commands **VTREQ**, **VTMP**, **VTSEC**,... ).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>DesignXplorer>Solution>Results File

**VTRSLT**, *Name*, *Entity*, *Type*, *Comp*, *ACC*, *CompName*

**Defines a result quantity for the DesignXplorer.**

VARIATIONAL TECHNOLOGY:DesignXplorer

<> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

### **Name**

Name of the result parameter. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be included.

### **Entity**

Entity keyword. Valid keywords are listed in the tables below.

### **Type**

Type of result parameter. Valid labels are listed in the table below depending on the *Entity* field.

### **Comp**

Component of the result parameter type. Valid labels are listed in the table below depending on the *Type* field.

### **ACC**

Required accuracy for the requested result parameter. Default values depend on the individual result parameter. The default value is 0.02 (2%).

### **CompName**

Name of an element or node component, depending on the *Entity* field. If *CompName* = ALL then all entities as specified in the *Entity* field are considered for the **VTRSLT** command. *CompName* = ALL is the default.

## Command Default

*ACC* = 2%, *CompName* = ALL.

## Notes

Defines the result quantity, for which an approximation function with respect to the input variables will be calculated.

In static analysis, displacement and temperature do not need to be specified, as they are automatically computed for all nodes of the model and for all components.

The accuracy provided with the *ACC* field should not be confused with the error due to meshing of the finite-element model. The accuracy is the maximum difference you can expect between an evaluation of the results of DesignXplorer at any point of the input space of the defined input variables and a rerun of a full ANSYS model at the same location of the input space.

The accuracy for all result parameters will be used to derive a global accuracy measure using the minimum value. This means that if there are several result parameters defined with the **VTRSLT** command and some

have a requested accuracy of  $ACC = 0.05$  and one has a requested accuracy of  $ACC = 0.01$ , then an accuracy of 0.01 (1%) will be adopted for all result parameters.

If the derivation order has been specified for one parameter, then the accuracy is not guaranteed for any variation involving this parameter.

For a normal modes analysis the result type MASS cannot be selected.

**Table 264 Table of Valid Labels for the *Type* and *Comp* Fields for *Entity* = ELEM**

Type	Comp	Description
MASS		Structural mass based on the elements.
S	ALL	Component nodal stress of the element(s) (not averaged at nodes).
TG	ALL	Component nodal temperature gradient of the element(s) (not averaged at nodes).
TF	ALL	Component nodal heat flux of the element(s) (not averaged at nodes).

### Note

For *Entity* = ELEM an existing component name including a set of elements must be specified in the *CompName* field.

**Table 265 Table of valid labels for the *Type* and *Comp* fields for *Entity* = Node**

Type	Comp	Description
U	ALL	Displacement
TEMP	ALL	Temperature
RF	FX, FY, FZ, MX, MY, MZ, ALL	Nodal reaction forces in the nodal coordinate system

### Note

For *Entity* = NODE an existing component name including a set of nodes must be specified in the *CompName* field.

**Table 266 Table of valid labels for the *Type* and *Comp* fields for *Entity* = Mode**

Type	Comp	Description
MODE		All mode shapes and the corresponding eigenfrequencies that are evaluated as part of the solution.
FREQ		All eigenfrequencies that are evaluated as part of the solution. Compared to the <i>Type</i> = MODE option, the <i>Type</i> = FREQ option does not provide the solutions for the mode shapes. The option will save memory compared to the <i>Type</i> = MODE option.

## Note

For *Entity* = MODE, all modes are always taken into consideration, that is, the only valid user input is *CompName* = ALL.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>DesignXplorer>Setup>Result Quantity

**VTSEC**, *Name*, *MIN*, *MAX*, *RedOpt*, *Lab*, *SECID*, *LAYERID*, *EIComp*, *VarType*, *Order*

Defines a section property as an input variable for DesignXplorer.

VARIATIONAL TECHNOLOGY: DesignXplorer  
<> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

### *Name*

Name for the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be included.

### *MIN*

Minimum value of the input variable.

### *MAX*

Maximum value of the input variable.

### *RedOpt*

Reduction option that specifies what to do if DesignXplorer must reduce the variation range from *MIN* to *MAX*. Valid labels are:

#### CONT

Continue with the calculation of the derivatives. This is the default.

#### STOP

Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

### *Lab*

Valid section property label. This label must refer to a valid property of the section as defined by the **SECTYPE** and **SECDATA** commands. The applicable label is:

#### TK

Shell thickness. This applies only if the section property has been defined with *Type* = SHELL of the **SECTYPE** command.

#### A

Area of section.

#### Iyy

Moment of inertia about the y axis.

#### Iyz

Product of inertia.

**Izz**

Moment of inertia about the z axis.

**Iw**

Wrapping constant.

**J**

Torsional constant.

**SECID**

Section identification number as defined by the **SECTYPE** command and further specified by the **SECDATA** command.

**LAYERID**

Layer identification number as defined by the **SECDATA** command.

**ELComp**

Name of an element component or number of a single individual element. If  $ELComp = ALL$ , then all elements are considered for the **VTSEC** command.

**VarType**

Label indicating how the variability of a spatially dependent element thickness should be handled. This label is ignored if the shell element thickness is constant for all elements selected by either the **SECID** or the **ELComp** field. Valid labels are:

**FACT**

The variability of the section property is described by a scale factor that is applied to the section property values. In case the section properties are different from element to element, a scale factor will be applied to all section property values. This means that the **MIN** field is the value of the minimum factor to be applied and the **MAX** field is the maximum factor to be applied. As such the values specified for **MIN** and **MAX** are unitless quantities.  $VarType = FACT$  is the default if the section property is different from element to element.

**ADD**

The variability of the section property is described by an add-on that is applied to the section property values. In case the section properties are different from element to element, then the section property values are shifted at the same time by this add-on. This means that the **MIN** field is the minimum value that is added to the section property values and the **MAX** field contains the maximum value that is added to the section property values. Hence, the values specified for **MIN** and **MAX** have the physical unit of a length.

**VAL**

The variability of the section property is described by the modification of the section property value itself. This is only applicable if the section property value is identical and constant for all selected elements. This means that the **MIN** field is the minimum value that the section property value itself can take and the **MAX** field contains the maximum value, which the section property value can take. This is the default if the section property is identical and constant from element to element.

**Order**

Derivation order. By default, driven by the application.

## Notes

Defines the section property of a single element or a set of elements as an input variable of DesignXplorer. The section property must be specified with the **SECTYPE** and **SECDATA** commands prior to the **VTSEC** command. The DesignXplorer will evaluate the derivatives of the result parameter(s) (see **VTRSLT** command) with respect to the section property defined here. The derivatives are used to determine an approximation





**Lab**

Valid surface load label. This label must be defined using the **SFE** or **SF** command prior to issuing this command. Applicable labels are:

**PRES**

Pressure, Force

**HFILM**

Convection surface loads (film coefficient)

**TBULK**

Convection surface loads (bulk temperature)

**HFLUX**

Heat flux

**LKEY**

Load key associated with surface load (not applicable when *Lab* is different from *PRES*). Valid load keys are: 1,2, 3, 4, or 5 for element **SURF154** and 1,2, 3, or 4 for element **SURF153**.

**ElComp**

Name of an element component or number of a single individual element. If *ElComp* = *ALL*, then all elements are considered for the **VTSFE** command.

**VarType**

Label indicating how the variability of surface load should be handled. Valid labels are:

**FACT**

The variability of the surface load is described by a scale factor that is applied to the load values. This means that the *MIN* field is the value of the minimum factor to be applied and the *MAX* field is the maximum factor to be applied. As such the values specified for *MIN* and *MAX* are unitless quantities. *VarType* = *FACT* is the default for surface load parameters.

**ADD**

The variability of the surface load is described by an add-on that is applied to the load values. This means that the *MIN* field is the minimum value that is added to the load values and the *MAX* field contains the maximum value that is added to the surface load values. Hence, the values specified for *MIN* and *MAX* have the same unit as the surface load itself.

**Order**

Derivation order. By default, driven by the application.

## Notes

Defines the surface load of a single element or a set of elements as an input variable of the DesignXplorer. The DesignXplorer will evaluate the derivatives of the result parameter(s) (see **VTRSLT** command) with respect to the surface load defined here. The derivatives are used to determine an approximation function for the result parameter(s) as a function of the surface load in the range specified by the *MIN* and *MAX* fields.

The type of the surface load is defined by the *Lab* field, which must not be left blank. The surface load must exist or be specified using the **SFE** command prior to issuing the **VTSFE** command.

Using the *ElComp* field a set of elements is selected for which the surface load is varied.

If the *Order* is not specified, it will be increased until the accuracy requirements of the **VTRSLT** command are satisfied. When you set a value for *Order*, no further adjustments for accuracy will be performed. *Order* should only be used when an input parameter has an influence with a limited order.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

This command cannot be accessed from a menu.

**VTSL**, *Type*, *Varname*

Selects a subset of elements associated with an VT input variable.

VARIATIONAL TECHNOLOGY: DesignXplorer  
 <> <> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

### *Type*

Label identifying the type of select:

- S**  
Select a new set. This is the default.
- R**  
Reselect a set from the current set.
- A**  
Additionally select a set and extend the current set.
- U**  
Unselect a set from the current set.

### *Varname*

Name of an VT input variable, defined with the **VTMP**, **VTREAL**, **VTSEC** or **VTDISC** commands.

## Command Default

There are no default values for the command fields.

## Notes

This command does not apply to output variables or **VTGEOM** input variables.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu> DesignXplorer> Setup> Frequency

**VTSTAT**, *EntyLis*

Print the status of the DesignXplorer definitions and settings into a separate window.

VARIATIONAL TECHNOLOGY: DesignXplorer  
<> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

**EntyLis**

Option that specifies whether the nodes or element lists should be printed for those Variational Technology commands where nodes or element lists have been specified. Valid labels are:

**NO**

Do not print out the list of nodes or elements (default).

**YES**

Print out the list of nodes or elements.

**Notes**

Prints the status the user definitions and settings of the DesignXplorer to a separate window.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>DesignXplorer>Other>Status**

---

**VTTEMP**, *Name, MIN, MAX, RedOpt, --, VarType, Order*

Defines the temperature as input variable for the DesignXplorer.

VARIATIONAL TECHNOLOGY: DesignXplorer  
<> <> <> <> <> <> <> <> <> <> <> <> VT <> <>

**Name**

Name for the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks are allowed in the string.

**MIN**

Minimum value of the input variable.

**MAX**

Maximum value of the input variable.

**RedOpt**

Reduction option that specifies what to do if the DesignXplorer needs to reduce the variation range from *MIN* to *MAX*. Valid labels are:

**CONT**

Continue with the calculation of the derivatives. This is the default.

**STOP**

Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

--

Unused field.

### ***VarType***

Label indicating how the variability of temperature should be handled. Valid labels are:

#### **FACT**

The variability of the temperature is described by an scaling factor that is applied to the temperature. This means that the *MIN* field is the value of the minimum factor to be applied and the *MAX* field is the maximum factor to be applied. Hence, the values specified for *MIN* and *MAX* are unitless quantities. *VarType* = FACT is the default for temperature parameters.

#### **ADD**

The variability of the temperature is described by an add-on that is applied to the temperature values. This means that the *MIN* field is the minimum value that is added to the temperature values and the *MAX* field contains the maximum value that is added to the temperature values. Hence, the values specified for *MIN* and *MAX* have the same unit as the temperature itself.

### ***Order***

Derivation Order. By default, driven by the application.

## **Command Default**

Defaults as outlined above.

## **Notes**

Defines the temperature as an input variable.

If the Order is not specified, it will be increased until the accuracy requirements of the **VTRSLT** command are satisfied. When you set a value for Order, no further adjustments for accuracy will be performed.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## **Menu Paths**

**Main Menu>DesignXplorer>Setup>Temperature Load**

**VTVMOD**, *Name*, *Oper*, *Value*

**Modifies the status or current value of an input variable for the DesignXplorer.**

VARIATIONAL TECHNOLOGY: DesignXplorer  
<> <> <> <> <> <> <> <> <> <> <> VT <> <>

### ***Name***

Name for an input or results variable of the DesignXplorer. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be used. An input variable with this name must be previously defined using one of the following commands: **VTFREQ**, **VTMP**, **VTRSLT**, ...

### ***Oper***

Label for the type of operation that is performed on the input variable Name. Valid labels are:

**ACT**

Activates an input variable in the DesignXplorer. This operation is ignored, if the variable specified is already activated.

**DEACT**

Deactivates an input variable of the DesignXplorer. This operation is ignored, if the variable specified is not activated.

**DEL**

Deletes an input or results variable of the DesignXplorer.

**SET**

Sets the value of an input variable of the DesignXplorer.

**Value**

Is the value of the input variable for the **VTEVAL** command.

## Notes

Modifies the status of an input variable for the DesignXplorer that have been previously defined, by activating, deactivating or deleting it.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>DesignXplorer>Setup>Modify**

---

### **VTYPE**, *NOHID*, *NZONE*

**Specifies the viewing procedure used to determine the form factors for the Radiation Matrix method.**

[AUX12: Radiation Matrix Method](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

**NOHID**

Type of viewing procedure:

**0**

Hidden procedure.

**1**

Non-hidden (faster, but less general) procedure.

**NZONE**

Number of sampling zones for the hidden procedure (100 maximum for 3-D, 1000 maximum for 2-D). Defaults to 20 for 3-D, 200 for 2-D. Number of points is  $2^{*NZONE}$  for 2-D and  $2^{*NZONE}*(NZONE+1)$  for 3-D.

## Command Default

Hidden procedure using 20 zones for 3-D, 200 zones for 2-D.

## Menu Paths

Main Menu>Radiation Opt>Matrix Method>Write Matrix

**/VUP**, *WN*, *Label*

Specifies the global Cartesian coordinate system reference orientation.

GRAPHICS:Views

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *WN*

Window number (or ALL) to which command applies (defaults to 1).

### *Label*

Orientation:

#### **Y**

Y vertical upward, X horizontal to the right, Z out from the screen (default).

#### **-Y**

Y vertical downward, X horizontal to the left, Z out from the screen.

#### **X**

X vertical upward, Y horizontal to the left, Z out from the screen.

#### **-X**

X vertical downward, Y horizontal to the right, Z out from the screen.

#### **Z**

Z vertical upward, Y horizontal to the right, X out from the screen. With this choice, you should use a view other than the **/VIEW** default of (0,0,1).

#### **-Z**

Z vertical downward, Y horizontal to the left, X out from the screen. With this choice, you should use a view other than the **/VIEW** default of (0,0,1).

## Command Default

Y vertical upward, X horizontal to the right, Z out from the screen.

## Notes

Specifies the global Cartesian coordinate system reference orientation. The **/VIEW** and **/ANGLE** commands may be used to reorient the view and are relative to this reference orientation. All coordinate systems are right-handed.

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>View Settings>Viewing Direction

**\*VWRITE**, *Par1, Par2, Par3, Par4, Par5, Par6, Par7, Par8, Par9, Par10, Par11, Par12, Par13, Par14, Par15, Par16, Par17, Par18, Par19*

**Writes data to a file in a formatted sequence.**

APDL: Array Parameters

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

## Argument Descriptions

*Par1, Par2, Par3, . . . , Par19*

You can write up to 19 parameters (or constants) at a time. Any *Par* values after a blank *Par* value are ignored. If you leave them all blank, one line will be written (to write a title or a blank line). If you input the keyword SEQU, a sequence of numbers (starting from 1) will be written for that item.

## Notes

You use **\*VWRITE** to write data to a file in a formatted sequence. Data items (*Par1, Par2*, etc.) may be array parameters, scalar parameters, character parameters (scalar or array), or constants. You must evaluate expressions and functions in the data item fields before using the **\*VWRITE** command, since initially they will be evaluated to a constant and remain constant throughout the operation. Unless a file is defined with the **\*CFOPEN** command, data is written to the standard output file. Data written to the standard output file may be diverted to a different file by first switching the current output file with the **/OUTPUT** command. You can also use the **\*MWRITE** command to write data to a specified file. Both commands contain format descriptors on the line immediately following the command. The format descriptors can be in either Fortran or C format.

You must enclose Fortran format descriptors in parentheses. They must immediately follow the **\*VWRITE** command on a separate line of the same input file. Do not include the word FORMAT. The format must specify the number of fields to be written per line, the field width, the placement of the decimal point, etc. You should use one field descriptor for each data item written. The write operation uses your system's available FORTRAN FORMAT conventions (see your system FORTRAN manual). You can use any standard FORTRAN *real* format (such as (4F6.0), (E10.3,2X,D8.2), etc.) and alphanumeric format (A). Alphanumeric strings are limited to a maximum of 8 characters for any field (A8) using the Fortran format. Use the "C" format for string arrays larger than 8 characters. Integer (I) and list-directed (\*) descriptors may *not* be used. You can include text in the format as a quoted string. *The parentheses must be included in the format* and the format must not exceed 80 characters (including parentheses). The output line length is limited to 128 characters.

The "C" format descriptors are used if the first character of the format descriptor line is not a left parenthesis. "C" format descriptors are up to 80 characters long, consisting of text strings and predefined "data descriptors" between the strings where numeric or alphanumeric character data will be inserted. The normal descriptors are %I for integer data, %G for double precision data, %C for alphanumeric character data, and %/ for a line break. There must be one data descriptor for each specified value (8 maximum) in the order of the specified values. The enhanced formats described in **\*MSG** may also be used.

For array parameter items, you must define the starting array element number. Looping continues (incrementing the vector index number of each array parameter by one) each time you output a line, until the maximum array vector element is written. For example, **\*VWRITE**,A(1) followed by (F6.0) will write one value per output line, i.e., A(1), A(2), A(3), A(4), etc. You write constants and scalar parameters with the same values for each loop. You can also control the number of loops and loop skipping with the **\*VLEN** and **\*VMASK** commands. The vector specifications **\*VABS**, **\*VFACT**, and **\*VCUM** do not apply to this command. If looping



continues beyond the supplied data array's length, zeros will be output for numeric array parameters and blanks for character array parameters. For multi-dimensioned array parameters, only the first (row) subscript is incremented. See the **\*VOPER** command for details. If you are in the GUI, the **\*VWRITE** command must be contained in an externally prepared file and read into ANSYS (i.e., **\*USE, /INPUT**, etc.).

This command is valid in any processor.

## Menu Paths

**Utility Menu>Parameters>Array Parameters>Write to File**

---



---

## W Commands

**/WAIT**, *DTIME*

Causes a delay before the reading of the next command.

APDL: Process Controls

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### Argument Descriptions

*DTIME*

Time delay (in seconds). Maximum time delay is 59 seconds.

### Notes

The command following the **/WAIT** will not be processed until the specified wait time increment has elapsed. Useful when reading from a prepared input file to cause a pause, for example, after a display command so that the display can be reviewed for a period of time. Another "wait" feature is available via the **\*ASK** command.

This command is valid in any processor.

### Menu Paths

**This command cannot be accessed from a menu.**

---

**WAVES**, *Wopt*, *OLDMAX*, *OLDRMS*

Initiates reordering.

PREP7: Element Reordering

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

*Wopt*

Option for comparison:

**MAX**

Use maximum wavefront value for comparison (default).

**RMS**

Use RMS wavefront value for comparison.

*OLDMAX*, *OLDRMS*

Maximum and RMS wavefront values of model to be used in place of the old values. *OLDRMS* defaults to *OLDMAX* (and vice versa). If neither is specified, each defaults to its calculated old value.

## Notes

Initiates the element reordering based upon the previously defined starting wave lists (if any). Reordering affects only the element order for the solution phase and not the element numbers (input referring to element numbers, such as element pressures, is unaffected by reordering).

---

### Note

The new order is retained only if new the new maximum or RMS wavefront values are lower than the old values, as described below. See the **WSORT** command for another reordering procedure. The resulting element ordering can be shown by listing the wavefront history [**WFRONT**,1] or by displaying elements with their element location numbers [**/PNUM**].

The **WAVES** reordering procedure is based upon a list of nodes defining where the element reordering is to start (either input on the **WSTART** command, or automatically determined if not input). If a list is input, additional starting wave lists may be defined to allow the user to block or guide the wave. An input starting list usually consists of one node for a line element model, a line of nodes for an area element model, or a plane of nodes for a volume element model. Elements are ordered from the first starting wave list in a direction that attempts to minimize the wavefront. Nodes are deleted and added to the total wave list set as reordering progresses through the model. Elements attached to nodes on succeeding starting lists, if any, are ignored at this time, thereby temporarily blocking that path. Whenever no more elements can be re-ordered, the procedure is repeated starting from the nodes defined on the second starting list (if any). The same node number should not appear on more than one list. If no starting list is input, a starting node is automatically selected from an element weighting procedure (see [Element Reordering](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications*).

Reordering may be made from the previously reordered model or from the initial model (by issuing the **SAVE** command before reordering and then restoring that model back to PREP7 with the **RESUME** command after reordering). The **WFRONT** command is useful for calculating and printing the current wavefront statistics at any time without causing any reordering of the elements. If a model is to be used for both a thermal and a structural analysis, the reordered statistics should be based upon the structural model (because of the higher number of degrees of freedom per node).

The reordering procedure treats separate portions of the model (i.e., not connected by elements) as discontinuous. Reordering automatically continues across a discontinuity as best as possible.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Numbering Ctrl>Element Reorder>Reorder by List**

---

## WERASE

Erases all reordering wave lists.

[PREP7: Element Reordering](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

## Notes

The **REORDER** then **STAT** commands will display the current wave lists.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

Main Menu>Preprocessor>Numbering Ctrl>Element Reorder>Erase Wave List

## WFRONT, *KPRNT*, *KCALC*

Estimates wavefront statistics.

[PREP7: Element Reordering](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

### *KPRNT*

Wavefront history print key:

**0**

Print current wavefront statistics.

**1**

Print current wavefront statistics but also print wavefront history (wavefront at each element). Elements are listed in the reordered sequence.

### *KCALC*

Calculation options:

**0**

Wavefront estimate assumes maximum model DOF set at each node and does not include the effects of master degrees of freedom and specified displacement constraints.

**1**

Wavefront estimate uses the actual DOF set at each node and does not include the effects of master degrees of freedom and specified displacement constraints. More time consuming than estimated wavefront. *KPRNT* = 1 is not available with this option.

## Notes

Estimates wavefront statistics of the model as currently ordered.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>NumberingCtrls>Element Reorder>Est. Wavefront**

---

**/WINDOW**, *WN*, *XMIN*, *XMAX*, *YMIN*, *YMAX*, *NCOPY*

**Defines the window size on the screen.**

[GRAPHICS: Set Up](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### ***WN***

Window reference number (1 to 5). Defaults to 1. This number, or ALL (for all active windows), may be used on other commands.

### ***XMIN*, *XMAX*, *YMIN*, *YMAX***

Screen coordinates defining window size. Screen coordinates are measured as -1.0 to 1.67 with the origin at the screen center. For example, (-1,1.67,-1,1) is full screen, (-1,0,-1,0) is the left bottom quadrant. If *XMIN* = OFF, deactivate this previously defined window; if ON, reactivate this previously defined window. If FULL, LEFT, RIGH, TOP, BOT, LTOP, LBOT, RTOP, RBOT, form full, half, or quarter window. If SQUA, form largest square window within the current graphics area. If DELE, delete this window (cannot be reactivated with ON).

### ***NCOPY***

Copies the current specifications from window *NCOPY* (1 to 5) to this window. If *NCOPY* = 0 (or blank), no specifications are copied.

## Command Default

One window at full screen.

## Notes

Defines the window size on the screen. Windows may occupy a separate section of the screen or they may overlap. Requested displays are formed in all windows according to the selected window specifications.

This command is valid in any processor.

## Menu Paths

**Utility Menu>PlotCtrls>Window Controls>Copy Window Specs**

**Utility Menu>PlotCtrls>Window Controls>Delete Window**

**Utility Menu>PlotCtrls>Window Controls>Window Layout**

**Utility Menu>PlotCtrls>Window Controls>Window On or Off**

---

**WMID**, *Key*

Specifies reordering options for the **WAVES** command.

[PREP7: Element Reordering](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

**Key**

Determines whether midside nodes are considered when reordering.

**NO**

Do not consider midside nodes when reordering (default).

**YES**

Consider midside nodes when reordering. This option is useful for models where line elements are only attached to midside nodes of solid elements.

**Notes**

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**This command cannot be accessed from a menu.**

**WMORE**, *NODE1, NODE2, NINC, ITIME, INC*

Adds more nodes to the starting wave list.

[PREP7: Element Reordering](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

***NODE1, NODE2, NINC***

Add another node set to the previous starting wave list. Set is *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* is negative, delete (instead of add) this node set from previous starting wave list.

***ITIME, INC***

Add other node sets to the same starting wave list by repeating the previous node set with *NODE1* and *NODE2* incremented by *INC* (defaults to 1) each time after the first. *ITIME* is the total number of sets (defaults to 1) defined with this command.

**Notes**

Adds more nodes to (or modifies) the previous starting wave list (if any) [**WSTART**]. Repeat **WMORE** command to add more nodes to the previous starting wave list. Up to 10,000 nodes may be defined (total, for all starting waves).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

**Menu Paths**

**Main Menu>Preprocessor>Numbering Ctrl>Element Reorder>Extend Wave List**

**WPAVE**, *X1, Y1, Z1, X2, Y2, Z2, X3, Y3, Z3*

**Moves the working plane origin to the average of specified points.**

DATABASE: Working Plane

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***X1, Y1, Z1***

Coordinates (in the active coordinate system) of the first point. If  $X1 = P$ , graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

***X2, Y2, Z2***

Coordinates (in the active coordinate system) of the second point.

***X3, Y3, Z3***

Coordinates (in the active coordinate system) of the third point.

**Notes**

Moves the origin of the working plane to the average of the specified points. A point is considered specified only if at least one of its coordinates is non-blank, and at least one point (1, 2, or 3) must be specified. Blank coordinates of a specified point are assumed to be zero. Averaging is based on the active coordinate system.

This command is valid in any processor.

**Menu Paths**

Utility Menu>WorkPlane>Offset WP to>Global Origin  
Utility Menu>WorkPlane>Offset WP to>Origin of Active CS  
Utility Menu>WorkPlane>Offset WP to>XYZ Locations

**WPCSYS**, *WN, KCN*

**Defines the working plane location based on a coordinate system.**

DATABASE: Working Plane

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***WN***

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If  $WN$  is a negative value, the viewing direction will not be modified.

***KCN***

Coordinate system number.  $KCN$  may be 0,1,2 or any previously defined local coordinate system number (defaults to the active system).

**Notes**

Defines a working plane location and orientation based on an existing coordinate system. If a Cartesian system is used as the basis ( $KCN$ ) for the working plane, the working plane will also be Cartesian, in the X-Y plane of the base system. If a cylindrical, spherical, or toroidal base system is used, the working plane will be a polar system in the R- $\theta$  plane of the base system.



If working plane tracking has been activated (**CSYS**,WP or **CSYS**,4), the updated active coordinate system will be of a similar type, except that a toroidal system will be updated to a cylindrical system. See the [Modeling and Meshing Guide](#) for more information on working plane tracking.

This command is valid in any processor.

Some primitive generation commands will not honor R-theta transformations for non-cartesian coordinate systems. Refer to the [primitive commands table](#) for more information.

## Menu Paths

**Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node**  
**Utility Menu>WorkPlane>Align WP with>Active Coord Sys**  
**Utility Menu>WorkPlane>Align WP with>Global Cartesian**  
**Utility Menu>WorkPlane>Align WP with>Specified Coord Sys**

---

**WPLANE**, *WN*, *XORIG*, *YORIG*, *ZORIG*, *XXAX*, *YXAX*, *ZXAX*, *XPLAN*, *YPLAN*, *ZPLAN*

**Defines a working plane to assist in picking operations.**

DATABASE: Working Plane

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### *WN*

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified. If fewer than three points are used, the viewing direction of window *WN* will be used instead to define the normal to the working plane.

### *XORIG*, *YORIG*, *ZORIG*

Global Cartesian coordinates of the origin of the working plane coordinate system.

### *XXAX*, *YXAX*, *ZXAX*

Global Cartesian coordinates of a point defining the x-axis orientation. The x-axis aligns with the projection of the line from this orientation point to the origin.

### *XPLAN*, *YPLAN*, *ZPLAN*

Global Cartesian coordinates of the third point defining the working plane. This point will also define the location of the positive XY-sector of the working plane coordinate system.

## Command Default

Working plane parallel to the global X-Y plane at Z = 0.0.

## Notes

Defines a working plane to assist in picking operations using the coordinates of three noncolinear points. The three points also define the working plane coordinate system. A minimum of one point (the working plane origin) is required. Immediate mode may also be active. See **WPSTYL** command to set the style of working plane display.

This command is valid in any processor.

## Menu Paths

Utility Menu>WorkPlane>Align WP with>XYZ Locations

---

**WPOFFS**, *XOFF*, *YOFF*, *ZOFF*

**Offsets the working plane.**

DATABASE: Working Plane

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*XOFF*, *YOFF*, *ZOFF*

Offset increments defined in the working plane coordinate system. If only *ZOFF* is used, the working plane will be redefined parallel to the present plane and offset by *ZOFF*.

## Notes

Changes the origin of the working plane by translating the working plane along its coordinate system axes.

This command is valid in any processor.

## Menu Paths

Utility Menu>WorkPlane>Offset WP by Increments

---

**WPROTA**, *THXY*, *THYZ*, *THZX*

**Rotates the working plane.**

DATABASE: Working Plane

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*THXY*

First rotation about the working plane Z axis (positive X toward Y).

*THYZ*

Second rotation about working plane X axis (positive Y toward Z).

*THZX*

Third rotation about working plane Y axis (positive Z toward X).

## Notes

The specified angles (in degrees) are relative to the orientation of the working plane.

This command is valid in any processor.

## Menu Paths

Utility Menu>WorkPlane>Offset WP by Increments

---

**WPSTYL**, *SNAP*, *GRSPAC*, *GRMIN*, *GRMAX*, *WPTOL*, *WPCTYP*, *GRTYPE*, *WPVIS*, *SNAPANG***Controls the display and style of the working plane.**

DATABASE: Working Plane

MP ME ST PR PRN &lt;&gt; &lt;&gt; FL EM EH DY PP &lt;&gt; EME MFS

**SNAP**

Snap increment for a locational pick (1E-6 minimum). If -1, turn off snap capability. For example, a picked location of 1.2456 with a snap of 0.1 gives 1.2, with 0.01 gives 1.25, with 0.001 gives 1.246, and with 0.025 gives 1.250 (defaults to 0.05).

**GRSPAC**

Graphical spacing between grid points. For graphical representation only and not related to snap points (defaults to 0.1).

**GRMIN, GRMAX**

Defines the size of a square grid (if *WPCTYP* = 0) to be displayed over a portion of the working plane. The opposite corners of the grid will be located at grid points nearest the working plane coordinates of (*GRMIN*,*GRMIN*) and (*GRMAX*,*GRMAX*). If a polar system (*WPCTYP* = 1), *GRMAX* is the outside radius of grid and *GRMIN* is ignored. If *GRMIN* = *GRMAX*, no grid will be displayed (defaults to -1.0 and 1.0 for *GRMIN* and *GRMAX* respectively).

**WPTOL**

The tolerance that an entity's location can deviate from the specified working plane, while still being considered on the plane. Used only for locational picking of vertices for polygons and prisms (defaults to 0.003).

**WPCTYP**

Working plane coordinate system type:

**0**

Cartesian (default). If working plane tracking is on [**CSYS**,4], the updated active coordinate system will also be Cartesian.

**1**

Polar. If working plane tracking is on, the updated active coordinate system will be cylindrical.

**2**

Polar. If working plane tracking is on, the updated active coordinate system will be spherical.

**GRTYPE**

Grid type:

**0**

Grid and WP triad.

**1**

Grid only.

**2**

WP triad only (default).

**WPVIS**

Grid visibility:

**0**

Do not show *GRTYPE* entities (grid and/or triad) (default).

1

Show *GRTYPE* entities. Cartesian working planes will be displayed with a Cartesian grid, polar with a polar grid.

**SNAPANG**

Snap angle (0--180) in degrees. Used only if *WPCTYP* = 1 or 2. Defaults to 5 degrees.

## Command Default

Snap capability on with an increment of .05, spacing between grid points of 0.1, Cartesian coordinate system, WP triad displayed, and a tolerance of .003.

## Notes

Use **WPSTYL,DEFA** to reset the working plane to its default location and style. Use **WPSTYL,STAT** to list the status of the working plane. Blank fields will keep present settings.

It is possible to specify *SNAP* and *WPTOL* values that will cause conflicts during picking operations. Check your values carefully, and if problems are noted, revert to the default values.

**WPSTYL** with no arguments will toggle the grid on and off. The working plane can be displayed in the non-GUI interactive mode only after issuing a **/PLOPTS,WP,1** command. See the *Modeling and Meshing Guide* for more information on working plane tracking. See **/PLOPTS** command for control of hidden line working plane.

This command is valid in any processor.

## Menu Paths

**Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>ROM>ElecStruc**

**Main Menu>Preprocessor>Modeling>Create>Circuit>Set Grid**

**Utility Menu>List>Status>Working Plane**

**Utility Menu>WorkPlane>Display Working Plane**

**Utility Menu>WorkPlane>Offset WP by Increments**

**Utility Menu>WorkPlane>Show WP Status**

**Utility Menu>WorkPlane>WP settings**

---

## WRFULL, *Ldstep*

**Stops solution after assembling global matrices.**

[SOLUTION: Load Step Options](#)

MP ME ST PR PRN <> <> <> EM EH <> PP <> EME MFS

***Ldstep***

Specify action to take:

**OFF or 0**

Turn off feature (default)

***N***

Turn on feature and set it to stop after assembling the global matrices and writing the *.FULL* file for load step *N*.

## Command Default

By default the WRFULL command is turned OFF.

## Notes

This command is used in conjunction with the **SOLVE** command to generate the assembled matrix file (.FULL file) only. The element matrices are assembled into the relevant global matrices for the particular analysis being performed and the .FULL file is written. Equation solution and the output of data to the results file are skipped. To dump the matrices written on the .FULL file into Harwell-Boeing format, use the **HBMAT** command in /AUX2. To copy the matrices to a postscript format that can be viewed graphically, use the **PSMAT** command.

To use the **LSSOLVE** macro with this command, you may need to modify the **LSSOLVE** macro to properly stop at the load step of interest.

This command only valid for linear static, full harmonic, and full transient analyses when the sparse direct solver is selected. This command is also valid for buckling or modal analyses with any mode extraction method. This command is not valid for nonlinear analyses.

In general, the assembled matrix file .FULL contains stiffness, mass, and damping matrices. However, the availability of the matrices depends on the analysis type chosen when the file is written.

## Menu Paths

**Main Menu>Preprocessor>Loads>Load Step Opts>Stop Solution**

**Main Menu>Solution>Load Step Opts>Stop Solution**

---

## WRITE, *Fname*

**Writes the radiation matrix file.**

[AUX12: Radiation Matrix Method](#)

MP ME ST PR PRN <> <> <> <> <> <> PP <> EME MFS

### *Fname*

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name Defaults to Jobname.

## Notes

Writes radiation matrix file (File.SUB) for input to the substructure thermal "use" pass. Subsequent **WRITE** operations to the same file overwrite the file.

## Menu Paths

**Main Menu>Radiation Opt>Matrix Method>Write Matrix**

---

**WSORT**, *Lab*, *KORD*, *--*, *Wopt*, *OLDMAX*, *OLDRMS*

**Initiates element reordering based upon a geometric sort.**

[PREP7: Element Reordering](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

### **Lab**

Coordinate (in the active system) along which element centroid locations are sorted. Valid labels are: X, Y, Z, ALL. If ALL (default), all three directions will be used, and the order corresponding to the lowest MAX or RMS wavefront value will be retained.

### **KORD**

Sort order:

**0**

Sort according to ascending coordinate values.

**1**

Sort according to descending coordinate values.

**--**

Unused field.

### **Wopt**

Option for comparison:

**MAX**

Use maximum wavefront value for comparison (default).

**RMS**

Use RMS wavefront value.

### **OLDMAX, OLDRMS**

MAX and RMS wavefront values of model to be used in place of the old values. *OLDRMS* defaults to *OLDMAX* (and vice versa). If neither is specified, each defaults to its calculated old value.

## **Notes**

Initiates element reordering based upon a geometric sort of the element centroid locations. Wave lists, if any [**WSTART**], are ignored. Reordering affects only the element order for the solution phase and not the element numbers (input referring to element numbers, such as element pressures, is unaffected by reordering).

---

### **Note**

The new order is retained only if new the new maximum or RMS wavefront values are lower than the old values, as described below. See the **WAVES** command for another reordering procedure and for more details on reordering. The resulting element ordering can be shown by listing the wavefront history [**WFRONT**,1] or by displaying elements with their element location numbers [**/PNUM**].

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## **Menu Paths**

**Main Menu>Preprocessor>Numbering Ctrl>Element Reorder>Reorder by XYZ**

## WSPRINGS

Creates weak springs on corner nodes of a bounding box of the currently selected elements.

[SOLUTION: Misc Loads](#)

MP ME ST PR PRN DS DSS <> <> <> <> PP <> EME MFS

### Notes

**WSPRINGS** invokes a predefined ANSYS macro that is used during the import of loads from the ADAMS program into the ANSYS program. **WSPRINGS** creates weak springs on the corner nodes of the bounding box of the currently selected elements. The six nodes of the bounding box are attached to ground using [COMBIN14](#) elements. The stiffness is chosen as a small number and can be changed by changing the real constants of the [COMBIN14](#) elements. This command works only for models that have a geometric extension in two or three dimensions. One dimensional problems (pure beam in one axis) are not supported.

For more information on how **WSPRINGS** is used during the transfer of loads from the ADAMS program to ANSYS, see [Import Loads into ANSYS](#) in the *Advanced Analysis Techniques Guide*.

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

### Menu Paths

Main Menu>Solution>ADAMS Connection>Import fr ADAMS

---

**WSTART**, *NODE1*, *NODE2*, *NINC*, *ITIME*, *INC*

Defines a starting wave list.

[PREP7: Element Reordering](#)

MP ME ST PR PRN <> <> <> EM <> <> PP <> EME MFS

***NODE1*, *NODE2*, *NINC***

Define a set of nodes in the starting wave list from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, ignore remaining fields and use all selected nodes [[NSEL](#)].

***ITIME*, *INC***

Add more node sets to the same starting wave list by repeating the previous node set with *NODE1* and *NODE2* incremented by *INC* (defaults to 1) each time after the first. *ITIME* is the total number of sets (defaults to 1) defined with this command.

### Command Default

If no starting waves, a starting node will be automatically defined when the [WAVES](#) command is issued.

### Notes

Defines a starting wave list (optional) for reordering with the [WAVES](#) command. Repeat **WSTART** command to define *other* starting wave lists (20 maximum).

**Distributed ANSYS Restriction** This command is not supported in Distributed ANSYS.

## Menu Paths

**Main Menu>Preprocessor>Numbering Ctrl>Element Reorder>Define Wave List**

---



---

## X Commands

**/XFRM**, *LAB*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

**Controls the centroid or the axis of dynamic rotation.**

GRAPHICS: Views

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

### **LAB**

The location or entity (centroid) used to define the center or axis of rotation.

### **NODE**

If NODE is chosen for the center of rotation, the node number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two nodes between which a line is drawn to determine the axis. The remaining arguments are ignored.

### **ELEMENT**

If ELEMENT is chosen for the center of rotation, the element number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two elements between which a line is drawn to determine the axis. The remaining arguments are ignored.

### **KP**

If KP is chosen for the center of rotation, the keypoint number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two keypoints between which a line is drawn to determine the axis. The remaining arguments are ignored.

### **LINE**

If LINE is chosen for the center of rotation, the line number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two lines between which a line is drawn to determine the axis. The remaining arguments are ignored.

### **AREA**

If AREA is chosen for the center of rotation, the area number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two areas between which a line is drawn to determine the axis. The remaining arguments are ignored.

### **VOLUME**

If VOLUME is chosen for the center of rotation, the volume number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two volumes between which a line is drawn to determine the axis. The remaining arguments are ignored.

### **XYZ**

If XYZ is chosen for the center of rotation, the location of that center is determined by the coordinates *X1*, *Y1*, *Z1*. If values are specified for *X2*, *Y2*, *Z2*, then the axis of rotation will be about the line between those two points.

### **OFF**

If *LAB* = OFF, DEFAULT, FOCUS or if no value is specified, then the center of rotation is set at the FOCUS point, as defined by the **/FOCUS** command.

**X1**

The entity number or X coordinate for the center of rotation.

**Y1**

The entity number or Y coordinate for the center of rotation.

**Z1**

The Z coordinate for the center of rotation.

**X2**

The X coordinate for the axis of rotation.

**Y2**

The Y coordinate for the axis of rotation.

**Z2**

The Z coordinate for the axis of rotation.

### **Command Default**

Issuing **/XFRM**, with no *LAB* defined sets the center of rotation at the focal point specified by the **/FOCUS** command.

### **Notes**

The **/XFRM** command is active only when the cumulative rotation key is specified ON for the **/ANGLE** command (*KINCR* = 1). This command affects dynamic manipulations only.

For center rotation, the middle mouse button will rotate the model about the screen Z axis and the right mouse button will rotate the model about the screen X and Y axis.

For rotation about an axis, the middle mouse button will rotate the model about the defined axis of rotation and the right mouse button will be deactivated.

This command is valid in any processor.

### **Menu Paths**

**Utility Menu>PlotCtrls>ViewSettings>Rotational Center>ByPick**

---

### **/XRANGE**, *XMIN*, *XMAX*

**Specifies a linear abscissa (X) scale range.**

[GRAPHICS: Graphs](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**XMIN**

Minimum abscissa scale value.

**XMAX**

Maximum abscissa scale value.

## Command Default

Automatically select X-range scale to include all data being displayed.

## Notes

Specifies a linear abscissa (X) scale range for the line graph display. Use **/XRANGE,DEFAULT** to return to automatic scaling.

Automatic scaling will often yield inappropriate range values for logarithmic scales (**/GROPT, LOGX**).

This command is valid in any processor.

## Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

## XVAR, *N*

Specifies the X variable to be displayed.

POST26:Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

*N*

X variable number:

**0 or 1**

Display **PLVAR** values vs. time (or frequency).

*n*

Display **PLVAR** values vs. variable *n* (2 to *NV* [**NUMVAR**]).

**1**

Interchange time and **PLVAR** variable numbers with time as the curve parameter. **PLVAR** variable numbers are displayed uniformly spaced along X-axis from position 1 to 10.

## Command Default

Use time or frequency.

## Notes

Defines the X variable (displayed along the abscissa) against which the Y variable(s) [**PLVAR**] are to be displayed.

## Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

**XVAROPT**, *Lab*

**Specifies the parameter to be used as the X-axis variable.**

OPTIMIZATION: Display

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

***Lab***

Parameter to be used as the X variable. Defaults to set numbers.

**Command Default**

Use design set numbers as the X variable.

**Notes**

Specifies the parameter to be used as the X-axis variable for graphical displays produced by the **PLVAROPT** command, and for the first column of the tabular listings produced by the **PRVAROPT** command. Design sets are automatically sorted in a sequence corresponding to an ascending order of the specified parameter.

**Menu Paths**

**Main Menu>Design Opt>Design Sets>Graphs/Tables**

---

---

## Y Commands

**/YRANGE**, *YMIN*, *YMAX*, *NUM*

Specifies a linear ordinate (Y) scale range.

[GRAPHICS: Graphs](#)

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**YMIN**

Minimum ordinate scale value.

**YMAX**

Maximum ordinate scale value.

**NUM**

Y-axis number to which range applies (defaults to 1). Valid numbers are 1 to 3 for **/GRTYP**, 2 and 1 to 6 for **/GRTYP**, 3. If ALL, apply to all Y-axes.

### Command Default

Automatically select Y-range scale to include all data being displayed.

### Notes

Specifies a linear ordinate (Y) scale range for the line graph display. Use **/YRANGE**, DEFAULT to return to automatic scaling. For multiple Y-axes graphs [**/GRTYP**], see **/GROPT**, ASCAL to automatically scale the additional Y-axes.

Automatic scaling will often yield inappropriate range values for logarithmic scales (**/GROPT**, LOGY).

This command is valid in any processor.

### Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

---



---

## Z Commands

**/ZOOM**, *WN, Lab, X1, Y1, X2, Y2*

**Zooms a region of a display window.**

GRAPHICS: Views

MP ME ST PR PRN <> <> FL EM EH DY PP <> EME MFS

**WN**

Window number to which command applies (defaults to 1).

**Lab**

Label to define the desired type of zoom:

**OFF**

Turns zoom off (refits image of entire model to the window).

**BACK**

Goes back to previous zoom setting (five successive back ups, maximum).

**SCRN**

Interprets X1,Y1 as the screen coordinates of the center of a square zoom region; X2,Y2 as the screen coordinates of a point on one side of that square.

**RECT**

Interprets X1,Y1 and X2,Y2 as the screen coordinates of two opposite corners of a rectangular zoom region.

### Notes

Zooms (centers and magnifies) the specified region of a display window. **/ZOOM** will operate on a display that has been formed by an explicit graphics action command (**APLOT**, **EPLLOT**, etc.). **/ZOOM** has no effect on an "immediate" graphics display. When **/ZOOM** is executed, the display is automatically replotted such that the specified zoom region is centered and sized to fill the window.

Auto resizing is disabled when you issue the **/ZOOM** command. To restore auto resizing, issue the **/AUTO** command, or select **FIT** from the **Pan, Zoom, Rotate** box.

This command is valid in any processor.

### Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

---





# Index

## Symbols

- \*ABBR command, 93
- \*AFUN command, 108
- \*ASK command, 166
- \*AXPY command, 180
- \*CFCLOS command, 255
- \*CFOPEN command, 255
- \*CFWRITE command, 256
- \*COMP command, 300
- \*CREATE command, 318
- \*CYCLE command, 333
- \*DEL command, 368
- \*DIM command, 382
- \*DMAT command, 400
- \*DO command, 408
- \*DOWHILE command, 411
- \*EIGEN command, 510
- \*ELSE command, 515
- \*ELSEIF command, 515
- \*END command, 525
- \*ENDDO command, 526
- \*ENDIF command, 526
- \*EXIT command, 577
- \*EXPORT command, 583
- \*FREE command, 700
- \*GET command, 717
- \*GO command, 778
- \*IF command, 835
- \*ITENGINE command, 857
- \*LIST command, 929
- \*LSBAC command, 941
- \*LSENGINE command, 949
- \*LSFACTOR command, 950
- \*MFOURI command, 998
- \*MFUN command, 1006
- \*MOPER command, 1022
- \*MSG command, 1054
- \*MULT command, 1072
- \*MWRITE command, 1072
- \*NRM command, 1110
- \*PRINT command, 1336
- \*REPEAT command, 1405
- \*RETURN command, 1414
- \*SET command, 1519
- \*SMAT command, 1575
- \*SREAD command, 1619
- \*STATUS command, 1634
- \*TAXIS command, 1659
- \*TOPER command, 1733
- \*TREAD command, 1743
- \*ULIB command, 1775
- \*USE command, 1782
- \*VABS command, 1793
- \*VCOL command, 1797
- \*VCUM command, 1799
- \*VEC command, 1803
- \*VEDIT command, 1805
- \*VFACT command, 1808
- \*VFILL command, 1809
- \*VFUN command, 1813
- \*VGET command, 1817
- \*VITRP command, 1828
- \*VLEN command, 1829
- \*VMASK command, 1831
- \*VOPER command, 1834
- \*VPLOT command, 1837
- \*VPUT command, 1839
- \*VREAD command, 1844
- \*VSCFUN command, 1850
- \*VSTAT command, 1854
- \*VWRITE command, 1884
- /AN3D command, 117
- /ANFILE command, 124
- /ANGLE command, 125
- /ANNOT command, 130
- /ANUM command, 146
- /ASSIGN command, 169
- /AUTO command, 174
- /AUX12 command, 176
- /AUX15 command, 176
- /AUX2 command, 175
- /AUX3 command, 175
- /AXLAB command, 179
- /BATCH command, 181
- /CFORMAT command, 257
- /CLABEL command, 265
- /CLEAR command, 266
- /CLOG command, 269
- /CMAP command, 272
- /COLOR command, 296
- /COM command, 299
- /CONFIG command, 302
- /CONTOUR command, 305
- /COPY command, 306
- /CPLANE command, 313
- /CTYPE command, 326
- /CVAL command, 329
- /CWD command, 330
- /CYCEXPAND command, 331
- /DELETE command, 369
- /DEVDISP command, 376

/DEVICE command, 378  
/DFLAB command, 381  
/DIRECTORY command, 385  
/DIST command, 386  
/DSCALE command, 413  
/DV3D command, 421  
/EDGE command, 461  
/EFACET command, 507  
/EOF command, 531  
/ERASE command, 538  
/ESHAPE command, 547  
/EXIT command, 577  
/EXPAND command, 580  
/FACET command, 592  
/FDELE command, 600  
/FILENAME command, 607  
/FOCUS command, 695  
/FORMAT command, 698  
/GCMD command, 715  
/GCOLUMN command, 715  
/GFILE command, 774  
/GFORMAT command, 774  
/GLINE command, 775  
/GMARKER command, 776  
/GO command, 779  
/GOLIST command, 779  
/GOPR command, 780  
/GRAPHICS command, 783  
/GRESUME command, 785  
/GRID command, 785  
/GROPT command, 786  
/GRTYP command, 789  
/GSAVE command, 790  
/GST command, 794  
/GTHK command, 795  
/GTYPE command, 796  
/HBC command, 799  
/HEADER command, 802  
/ICLWID command, 833  
/ICSCALE command, 833  
/IMAGE command, 839  
/INPUT command, 847  
/INQUIRE command, 848  
/LARC command, 895  
/LIGHT command, 924  
/LINE command, 926  
/LSPEC command, 952  
/LSYMBOL command, 958  
/MAIL command, 971  
/MENU command, 977  
/MKDIR command, 1011  
/MPLIB command, 1042  
/MREP command, 1050  
/MSTART command, 1068  
/NERR command, 1084  
/NOERASE command, 1101  
/NOLIST command, 1101  
/NOPR command, 1103  
/NORMAL command, 1105  
/NUMBER command, 1133  
/OPT command, 1175  
/OUTPUT command, 1181  
/PAGE command, 1189  
/PBC command, 1198  
/PBF command, 1201  
/PCIRCLE command, 1208  
/PCOPY command, 1208  
/PDS command, 1240  
/PICE command, 1261  
/PLOPTS command, 1287  
/PMACRO command, 1305  
/PMORE command, 1309  
/PNUM command, 1311  
/POLYGON command, 1314  
/POST1 command, 1315  
/POST26 command, 1315  
/PREP7 command, 1325  
/PSEARCH command, 1365  
/PSF command, 1366  
/PSPEC command, 1377  
/PSTATUS command, 1378  
/PSYMB command, 1380  
/PWEDGE command, 1384  
/QUIT command, 1388  
/RATIO command, 1396  
/RENAME command, 1403  
/REPLOTT command, 1405  
/RESET command, 1410  
/RGB command, 1418  
/RMDIR command, 1425  
/SECLIB command, 1492  
/SEG command, 1511  
/SHADE command, 1561  
/SHOW command, 1563  
/SHOWDISP command, 1565  
/SHRINK command, 1569  
/SMBC command, 1578  
/SOLU command, 1594  
/SSCALE command, 1622  
/STATUS command, 1635  
/STITLE command, 1637  
/SYP command, 1657  
/SYS command, 1657  
/TEE command, 1718

- /TITLE command, 1727
  - /TLABEL command, 1727
  - /TRIAD command, 1745
  - /TRLCY command, 1746
  - /TSPEC command, 1752
  - /TXTRE command, 1756
  - /TYPE command, 1759
  - /UCMD command, 1765
  - /UDOC command, 1766
  - /UI command, 1767
  - /UIS command, 1771
  - /UNITS command, 1778
  - /USER command, 1783
  - /VCONE command, 1798
  - /VIEW command, 1825
  - /VSCALE command, 1849
  - /VT command, 1858
  - /VUP command, 1883
  - /WAIT command, 1887
  - /WINDOW command, 1890
  - /XFRM command, 1901
  - /XRANGE command, 1902
  - /YRANGE command, 1905
  - /ZOOM command, 1907
  - ~CAT5IN command, 83
  - ~CATIAIN command, 84
  - ~PARAIN command, 85
  - ~PROEIN command, 86
  - ~SATIN command, 87
  - ~UGIN command, 88
- A**
- A command, 91
  - AADD command, 91
  - AATT command, 92
  - ABBRES command, 94
  - ABBSAV command, 95
  - ABEXTRACT command, 93
  - ABS command, 96
  - ACCAT command, 97
  - ACCOPTION command, 98
  - ACEL command, 99
  - ACLEAR command, 100
  - ADAMS command, 101
  - ADAPT command, 102
  - ADD command, 103
  - ADDAM command, 104
  - ADELE command, 104
  - ADGL command, 105
  - ADRAG command, 105
  - AESIZE command, 106
  - AFILLT command, 107
  - AFLIST command, 107
  - AFSURF command, 108
  - AGEN command, 109
  - AGLUE command, 110
  - AINA command, 111
  - AINP command, 111
  - AINV command, 112
  - AL command, 112
  - ALIST command, 113
  - ALLSEL command, 114
  - ALPHAD command, 115
  - AMAP command, 115
  - AMESH command, 116
  - ANCNTR command, 119
  - ANCUT command, 119
  - ANCYC command, 120
  - ANDATA command, 121
  - ANDSCL command, 122
  - ANDYNA command, 123
  - ANFLOW command, 124
  - ANHARM command, 126
  - ANIM command, 127
  - ANISOS command, 128
  - ANMODE command, 128
  - ANMRES command, 129
  - ANORM command, 131
  - ANSOL command, 132
  - ANSTOAQWA command, 135
  - ANSTOASAS command, 137
  - ANTIME command, 142
  - ANTYPE command, 143
  - AOFFST command, 147
  - AOVLAP command, 148
  - APLOT command, 149
  - APPEND command, 149
  - APTN command, 151
  - ARCLEN command, 151
  - ARCTRM command, 152
  - AREAS command, 153
  - AREFINE command, 154
  - AREMESH command, 155
  - AREVERSE command, 156
  - AROTAT command, 156
  - ARSCALE command, 157
  - ARSYM command, 158
  - ASBA command, 159
  - ASBL command, 161
  - ASBV command, 162
  - ASBW command, 163
  - ASEL command, 164
  - ASKIN command, 167
  - ASLL command, 168

ASLV command, 169  
ASUB command, 170  
ASUM command, 171  
ATAN command, 171  
ATRAN command, 172  
ATYPE command, 173  
AUTOTS command, 174  
AVPRIN command, 177  
AVRES command, 178

**B**

BCSOPTION command, 182  
BETAD command, 184  
BF command, 184  
BFA command, 188  
BFADELE command, 190  
BFALIST command, 191  
BFCUM command, 192  
BFDELE command, 193  
BFE command, 195  
BFECUM command, 197  
BFEDELE command, 199  
BFELIST command, 200  
BFESCAL command, 200  
BFINT command, 202  
BFK command, 203  
BFKDELE command, 205  
BFKLIST command, 206  
BFL command, 207  
BFLDELE command, 209  
BFLIST command, 210  
BFLLIST command, 210  
BFSCALE command, 211  
BFTRAN command, 212  
BFUNIF command, 212  
BFV command, 214  
BFVDELE command, 215  
BFVLIST command, 216  
BIOOPT command, 217  
BIOT command, 217  
BLC4 command, 218  
BLC5 command, 219  
BLOCK command, 220  
BOOL command, 220  
BOPTN command, 221  
BSAX command, 222  
BSM1 command, 224  
BSM2 command, 225  
BSMD command, 223  
BSPLIN command, 226  
BSS1 command, 226  
BSS2 command, 227

BSTE command, 228  
BSTQ command, 229  
BTOL command, 230  
BUCOPT command, 230

**C**

C\*\*\* command, 233  
CALC command, 233  
CAMPBELL command, 234  
CBDOF command, 235  
CBMD command, 237  
CBMX command, 237  
CBTE command, 238  
CBTMP command, 239  
CDOPT command, 239  
CDREAD command, 240  
CDWRITE command, 241  
CE command, 245  
CECHECK command, 246  
CECMOD command, 247  
CECYC command, 248  
CEDELE command, 249  
CEINTF command, 249  
CELIST command, 250  
CENTER command, 251  
CEQN command, 251  
CERIG command, 252  
CESGEN command, 254  
CFACT command, 254  
CGLOC command, 258  
CGOMGA command, 259  
CHECK command, 259  
CHKMSH command, 260  
CINT command, 261  
CIRCLE command, 264  
CISOL command, 264  
CLOCAL command, 267  
CLOG command, 268  
CLRMSHLN command, 269  
CM command, 270  
CMACEL command, 271  
CMATRIX command, 273  
CMDELE command, 274  
CMDOMEGA command, 275  
CMEDIT command, 276  
CMGRP command, 277  
CMLIST command, 278  
CMMOD command, 279  
CMOMEGA command, 279  
CMPLOT command, 281  
CMROTATE command, 283  
CMSEL command, 284

CMSFILE command, 285  
CMSOPT command, 287  
CMWRITE command, 288  
CNCHECK command, 289  
CNKMOD command, 293  
CNVTOL command, 294  
COMPRESS command, 300  
CON4 command, 301  
CONE command, 301  
CONJUG command, 304  
CORIOLIS command, 307  
COUPLE command, 308  
COVAL command, 309  
CP command, 309  
CPCYC command, 311  
CPDELE command, 312  
CPINTF command, 313  
CPLGEN command, 314  
CPLIST command, 314  
CPMERGE command, 315  
CPNGEN command, 316  
CPSGEN command, 317  
CQC command, 317  
CRPLIM command, 319  
CS command, 320  
CSCIR command, 321  
CSDELE command, 322  
CSKP command, 322  
CSLIST command, 323  
CSWPLA command, 324  
CSYS command, 325  
CURR2D command, 327  
CUTCONTROL command, 327  
CVAR command, 329  
CYCLIC command, 334  
CYCOPT command, 336  
CYCPHASE command, 340  
CYL4 command, 343  
CYL5 command, 344  
CYLIND command, 344  
CZDEL command, 345  
CZMESH command, 346

## D

D command, 349  
DA command, 352  
DADELE command, 355  
DALIST command, 358  
DAMORPH command, 358  
DATA command, 359  
DATADEF command, 360  
DCGOMG command, 361

DCUM command, 361  
DCVSWP command, 362  
DDELE command, 364  
DDOPTION command, 366  
DEACT command, 367  
DEFINE command, 367  
DELETE command, 369  
DELTIM command, 370  
DEMORPH command, 371  
DERIV command, 372  
DESIZE command, 373  
DESOL command, 374  
DETAB command, 376  
DIG command, 381  
DIGIT command, 382  
DISPLAY command, 386  
DJ command, 387  
DJDELE command, 389  
DJLIST command, 390  
DK command, 391  
DKDELE command, 392  
DKLIST command, 393  
DL command, 394  
DLDELE command, 397  
DLIST command, 399  
DLLIST command, 399  
DMOVE command, 402  
DMPEXT command, 403  
DMPRAT command, 404  
DNSOL command, 405  
DOF command, 409  
DOFSEL command, 409  
DOMEGA command, 411  
DSCALE command, 412  
DSET command, 414  
DSPOPTION command, 414  
DSUM command, 416  
DSURF command, 417  
DSYM command, 417  
DSYS command, 419  
DTRAN command, 420  
DUMP command, 421  
DVAL command, 424  
DVMORPH command, 425  
DYNOPT command, 426

## E

E command, 427  
EALIVE command, 427  
EDADAPT command, 428  
EDALE command, 429  
EDASMP command, 430

EDBOUND command, 431  
EDBVIS command, 434  
EDBX command, 433  
EDCADAPT command, 434  
EDCGEN command, 436  
EDCLIST command, 441  
EDCMORE command, 441  
EDCNSTR command, 442  
EDCONTACT command, 445  
EDCPU command, 447  
EDCRB command, 447  
EDCSC command, 448  
EDCTS command, 449  
EDCURVE command, 450  
EDDAMP command, 452  
EDDBL command, 454  
EDDC command, 455  
EDDRELAX command, 456  
EDDUMP command, 457  
EDELE command, 458  
EDENERGY command, 458  
EDFPLOT command, 459  
EDGCALE command, 460  
EDHGLS command, 462  
EDHIST command, 462  
EDHTIME command, 463  
EDINT command, 464  
EDIPART command, 465  
EDIS command, 467  
EDLCS command, 468  
EDLOAD command, 469  
EDMP command, 474  
EDNB command, 477  
EDNDTSD command, 478  
EDNROT command, 479  
EDOPT command, 480  
EDOUT command, 481  
EDPART command, 483  
EDPC command, 485  
EDPL command, 485  
EDPVEL command, 486  
EDRC command, 488  
EDRD command, 490  
EDREAD command, 491  
EDRI command, 493  
EDRST command, 494  
EDRUN command, 494  
EDSHELL command, 495  
EDSOLV command, 497  
EDSP command, 497  
EDSTART command, 498  
EDTERM command, 500  
EDTP command, 501  
EDVEL command, 502  
EDWELD command, 505  
EDWRITE command, 506  
EGEN command, 509  
EINTF command, 510  
EKILL command, 512  
ELBOW command, 512  
ELEM command, 514  
ELIST command, 514  
EMAGERR command, 517  
EMATWRITE command, 517  
EMF command, 518  
EMFT command, 519  
EMID command, 519  
EMIS command, 520  
EMODIF command, 521  
EMORE command, 521  
EMSYM command, 522  
EMTGEN command, 522  
EMUNIT command, 524  
EN command, 525  
ENDRELEASE command, 527  
ENERSOL command, 528  
ENGEN command, 528  
ENORM command, 529  
ENSYM command, 530  
EORIENT command, 532  
EPLOT command, 533  
EQSLV command, 534  
ERASE command, 538  
EREA command, 539  
EREFINE command, 539  
EREINF command, 541  
ERESX command, 541  
ERNORM command, 542  
ERRANG command, 543  
ESCHECK command, 543  
ESEL command, 544  
ESIZE command, 549  
ESLA command, 550  
ESLL command, 550  
ESLN command, 551  
ESLV command, 552  
ESOL command, 553  
ESORT command, 556  
ESSOLV command, 557  
ESTIF command, 560  
ESURF command, 560, 1070  
ESYM command, 562  
ESYS command, 563  
ET command, 564

ETABLE command, 565  
ETCHG command, 572  
ETCONTROL command, 574  
ETDELE command, 575  
ETLIST command, 575  
ETYPE command, 575  
EUSORT command, 576  
EWRITE command, 576  
EXP command, 578  
EXPAND command, 579  
EXPASS command, 582  
EXPROFILE command, 584  
EXPSOL command, 586  
EXTOPT command, 586  
EXTREM command, 588  
EXUNIT command, 589

## F

F command, 591  
FATIGUE command, 593  
FC command, 593  
FCCHECK command, 595  
FCDELE command, 595  
FCLIST command, 596  
FCTYP command, 597  
FCUM command, 596  
FDELE command, 599  
FE command, 601  
FEBODY command, 601  
FECONS command, 602  
FEFOR command, 602  
FELIST command, 603  
FESURF command, 603  
FILE command, 603  
FILEAUX2 command, 604  
FILEAUX3 command, 605  
FILEDISP command, 605  
FILL command, 606  
FILLDATA command, 607  
FINISH command, 608  
FILOT command, 609  
FITEM command, 610  
FJ command, 611  
FJDELE command, 611  
FJLIST command, 612  
FK command, 613  
FKDELE command, 614  
FKLIST command, 615  
FL command, 615  
FLDATA command, 616  
FLDATA1 command, 619  
FLDATA10 command, 638

FLDATA11 command, 640  
FLDATA12 command, 641  
FLDATA13 command, 642  
FLDATA14 command, 643  
FLDATA15 command, 644  
FLDATA16 command, 644  
FLDATA17 command, 645  
FLDATA18 command, 646  
FLDATA19 command, 648  
FLDATA2 command, 620  
FLDATA20 command, 649  
FLDATA20A command, 651  
FLDATA20B command, 652  
FLDATA21 command, 653  
FLDATA22 command, 654  
FLDATA23 command, 656  
FLDATA24 command, 657  
FLDATA24A command, 660  
FLDATA24B command, 662  
FLDATA24C command, 663  
FLDATA24D command, 664  
FLDATA24E command, 665  
FLDATA24F command, 666  
FLDATA24G command, 667  
FLDATA24H command, 668  
FLDATA25 command, 669  
FLDATA26 command, 670  
FLDATA27 command, 671  
FLDATA28 command, 673  
FLDATA29 command, 674  
FLDATA3 command, 622  
FLDATA30 command, 676  
FLDATA31 command, 677  
FLDATA32 command, 678  
FLDATA33 command, 681  
FLDATA34 command, 682  
FLDATA35 command, 682  
FLDATA36 command, 683  
FLDATA37 command, 684  
FLDATA38 command, 685  
FLDATA39 command, 686  
FLDATA4 command, 623  
FLDATA40 command, 688  
FLDATA4A command, 626  
FLDATA5 command, 627  
FLDATA6 command, 631  
FLDATA7 command, 632  
FLDATA8 command, 636  
FLDATA9 command, 637  
FLIST command, 689  
FLLIST command, 689  
FLOCHECK command, 690

FLOTRAN command, 691  
FLREAD command, 691  
FLST command, 692  
FLUXV command, 693  
FMAGBC command, 693  
FMAGSUM command, 694  
FOR2D command, 696  
FORCE command, 696  
FORM command, 697  
FP command, 699  
FPLIST command, 700  
FREQ command, 700  
FRQSCL command, 701  
FS command, 702  
FSCALE command, 703  
FSDELE command, 703  
FSLIST command, 704  
FSNODE command, 704  
FSPLOT command, 705  
FSSECT command, 706  
FSSPARM command, 707  
FSUM command, 707  
FTCALC command, 709  
FTRAN command, 709  
FTSIZE command, 709  
FTWRITE command, 710  
FVMESH command, 711

## G

GAP command, 713  
GAPF command, 713  
GAUGE command, 714  
GENOPT command, 716  
GEOM command, 716  
GEOMETRY command, 717  
GMATRIX command, 777  
GMFACE command, 778  
GP command, 780  
GPDELE command, 782  
GPLIST command, 782  
GPLOT command, 783  
GRP command, 788  
GSBDATA command, 791  
GSGDATA command, 792  
GSLIST command, 792  
GSSOL command, 793  
GSUM command, 795

## H

HARFRQ command, 799  
HBMAT command, 800  
HELP command, 803

HELPDISP command, 804  
HEMIOPT command, 804  
HFADP command, 804  
HFANG command, 805  
HFARRAY command, 805  
HFDEEM command, 806  
HFEIGOPT command, 807  
HFREFINE command, 808  
HFMODPRT command, 808  
HFPA command, 809  
HFPCSWP command, 809  
HFPORT command, 811  
HFPOWER command, 810  
HFSCAT command, 817  
HFSYM command, 818  
HMAGSOLV command, 819  
HPGL command, 820  
HPTCREATE command, 821  
HPTDELETE command, 822  
HRCPLX command, 823  
HREXP command, 824  
HROPT command, 825  
HROUT command, 827

## I

IC command, 829  
ICDELE command, 830  
ICE command, 831  
ICEDELE command, 831  
ICELIST command, 832  
ICLIST command, 832  
ICVFRC command, 834  
IGESIN command, 837  
IGESOUT command, 838  
IMAGIN command, 840  
IMESH command, 841  
IMMED command, 842  
IMPD command, 843  
INISTATE command, 844  
INRES command, 851  
INRTIA command, 852  
INT1 command, 852  
INTSRF command, 853  
IOPTN command, 854  
IRLF command, 856  
IRLIST command, 857

## J

JPEG command, 859  
JSOL command, 860



**K**

K command, 861  
KATT command, 861  
KBC command, 862  
KBETW command, 863  
KCALC command, 864  
KCENTER command, 865  
KCLEAR command, 867  
KDELE command, 868  
KDIST command, 868  
KEEP command, 869  
KESIZE command, 870  
KEYOPT command, 870  
KEYPTS command, 871  
KEYW command, 871  
KFILL command, 872  
KGEN command, 873  
KL command, 874  
KLIST command, 874  
KMESH command, 875  
KMODIF command, 875  
KMOVE command, 876  
KNODE command, 877  
KPLOT command, 877  
KPSCALE command, 878  
KREFINE command, 879  
KSCALE command, 880  
KSCON command, 881  
KSEL command, 882  
KSLL command, 884  
KSLN command, 884  
KSUM command, 885  
KSYMM command, 885  
KTRAN command, 886  
KUSE command, 887  
KWPAVE command, 888  
KWPLAN command, 888

**L**

L command, 891  
L2ANG command, 892  
L2TAN command, 892  
LANG command, 893  
LARC command, 894  
LAREA command, 895  
LARGE command, 896  
LATT command, 897  
LAYER command, 898  
LAYERP26 command, 899  
LAYLIST command, 900  
LAYPLOT command, 900  
LCABS command, 901

LCASE command, 902  
LCCALC command, 902  
LCCAT command, 903  
LCDEF command, 904  
LCFACT command, 904  
LCFILE command, 905  
LCLEAR command, 906  
LCOMB command, 906  
LCOPER command, 907  
LCSEL command, 909  
LCSL command, 910  
LCSUM command, 910  
LCWRITE command, 911  
LCZERO command, 911  
LDELE command, 912  
LDIV command, 912  
LDRAG command, 913  
LDREAD command, 914  
LESIZE command, 918  
LEXTND command, 920  
LFILLT command, 920  
LFSURF command, 921  
LGEN command, 922  
LGLUE command, 923  
LGWRITE command, 923  
LINA command, 925  
LINE command, 926  
LINES command, 927  
LINL command, 927  
LINP command, 928  
LINV command, 928  
LIST command, 929  
LLIST command, 930  
LMATRIX command, 931  
LMESH command, 932  
LNSRCH command, 933  
LOCAL command, 934  
LOVLAP command, 935  
LPLOT command, 935  
LPRT command, 936  
LPTN command, 936  
LREFINE command, 937  
LREVERSE command, 938  
LROTAT command, 939  
LSBA command, 940  
LSBL command, 941  
LSBV command, 943  
LSBW command, 944  
LSCLEAR command, 945  
LSDELE command, 946  
LSEL command, 946  
LSLA command, 950

LSLK command, 951  
LSOPER command, 951  
LSREAD command, 953  
LSSCALE command, 954  
LSSOLVE command, 955  
LSTR command, 956  
LSUM command, 956  
LSWRITE command, 957  
LSYMM command, 959  
LTAN command, 960  
LTRAN command, 961  
LUMPM command, 962  
LVSCALE command, 962  
LWPLAN command, 963

## M

M command, 965  
MADAPT command, 966  
MAGOPT command, 968  
MAGSOLV command, 969  
MAPSOLVE command, 972  
MASTER command, 972  
MAT command, 973  
MATER command, 973  
MCHECK command, 973  
MDAMP command, 974  
MDELE command, 975  
MDPLOT command, 976  
MEMM command, 976  
MESHING command, 978  
MFANALYSIS command, 978  
MFBUCKET command, 979  
MFCALC command, 980  
MFCI command, 981  
MFCLEAR command, 981  
MFCMMAND command, 982  
MFCONV command, 983  
MFDTIME command, 984  
MFELEM command, 985  
MFEM command, 986  
MFEXTER command, 986  
MFFNAME command, 987  
MFFR command, 988  
MFIMPORT command, 989  
MFINTER command, 990  
MFITER command, 991  
MFLC command, 991  
MFLIST command, 994  
MFMAP command, 995  
MFORDER command, 996  
MFOUTPUT command, 996  
MFPS command, 999

MFRC command, 1000  
MFRELAX command, 1001  
MFRSTART command, 1002  
MFSO command, 1003  
MFSURFACE command, 1004  
MFTIME command, 1005  
MFTOL command, 1005  
MFVOLUME command, 1007  
MFWR command, 1008  
MGEN command, 1009  
MIDTOL command, 1010  
MLIST command, 1011  
MMASS command, 1012  
MMF command, 1013  
MODCONT command, 1013  
MODE command, 1014  
MODIFY command, 1015  
MODMSH command, 1016  
MODOPT command, 1017  
MONITOR command, 1020  
MOPT command, 1025  
MORPH command, 1029  
MOVE command, 1031  
MP command, 1032  
MPAMOD command, 1036  
MPCHG command, 1037  
MPCOPY command, 1038  
MPDATA command, 1038  
MPDELE command, 1041  
MPDRES command, 1042  
MPLIST command, 1043  
MPPLOT command, 1044  
MPREAD command, 1044  
MPRINT command, 1046  
MPTEMP command, 1046  
MPTGEN command, 1047  
MPTRES command, 1048  
MPWRITE command, 1048  
MSADV command, 1050  
MSAVE command, 1051  
MSCAP command, 1052  
MSDATA command, 1053  
MSHAPE command, 1056  
MSHCOPY command, 1057  
MSHKEY command, 1058  
MSHMID command, 1058  
MSHPATTERN command, 1059  
MSMASS command, 1060  
MSMETH command, 1061  
MSMIR command, 1062  
MSNOMF command, 1063  
MSPROP command, 1063

MSQUAD command, 1065  
MSRELAX command, 1066  
MSSOLU command, 1066  
MSSPEC command, 1067  
MSTERM command, 1069  
MSVARY command, 1071  
MXPAND command, 1074

## N

N command, 1077  
NANG command, 1077  
NAXIS command, 1078  
NCNV command, 1079  
NDELE command, 1080  
NDIST command, 1081  
NDSURF command, 1082  
NEQIT command, 1083  
NFORCE command, 1085  
NGEN command, 1086  
NKPT command, 1087  
NLDIAG command, 1087  
NLDPOST command, 1090  
NLGEOM command, 1092  
NLHIST command, 1093  
NLIST command, 1097  
NLOG command, 1098  
NLOPT command, 1098  
NMODIF command, 1099  
NOCOLOR command, 1100  
NODES command, 1100  
NOOFFSET command, 1102  
NOORDER command, 1103  
NORA command, 1104  
NORL command, 1105  
NPLOT command, 1106  
NPRINT command, 1107  
NREAD command, 1107  
NREFINE command, 1108  
NRLSUM command, 1109  
NROPT command, 1111  
NROTAT command, 1112  
NRRANG command, 1113  
NSCALE command, 1113  
NSEL command, 1114  
NSLA command, 1120  
NSLE command, 1120  
NSLK command, 1122  
NSLL command, 1122  
NSLV command, 1123  
NSMOOTH command, 1124  
NSOL command, 1124  
NSORT command, 1126

NSTORE command, 1130  
NSUBST command, 1131  
NSVR command, 1132  
NSYM command, 1132  
NUMCMP command, 1134  
NUMEXP command, 1135  
NUMMRG command, 1137  
NUMOFF command, 1141  
NUMSTR command, 1142  
NUMVAR command, 1143  
NUSORT command, 1144  
NWPAVE command, 1144  
NWPLAN command, 1144  
NWRITE command, 1145

## O

OCDATA command, 1147  
OCDELETE command, 1151  
OCLIST command, 1152  
OCTABLE command, 1153  
OCTYPE command, 1155  
OMEGA command, 1157  
OPADD command, 1158  
OPANL command, 1158  
OPCLR command, 1159  
OPDATA command, 1159  
OPDEL command, 1160  
OPEQN command, 1161  
OPERATE command, 1162  
OPEXE command, 1163  
OPFACT command, 1163  
OPFRST command, 1164  
OPGRAD command, 1165  
OPKEEP command, 1165  
OPLFA command, 1166  
OPLGR command, 1167  
OPLIST command, 1167  
OPLOOP command, 1168  
OPLSW command, 1169  
OPMAKE command, 1169  
OPNCONTROL command, 1170  
OPPRNT command, 1170  
OPRAND command, 1171  
OPRESU command, 1171  
OPRFA command, 1172  
OPRGR command, 1172  
OPRSW command, 1173  
OPSAVE command, 1173  
OPSEL command, 1174  
OPSUBP command, 1174  
OPSWEEP command, 1175  
OPTYPE command, 1176

OPUSER command, 1177  
OPVAR command, 1178  
OUTOPT command, 1179  
OUTPR command, 1179  
OUTRES command, 1182

## P

PADELE command, 1189  
PAGET command, 1190  
PAPUT command, 1191  
PARESU command, 1191  
PARRES command, 1194  
PARSAV command, 1194  
PARTSEL command, 1192  
PASAVE command, 1195  
PATH command, 1196  
PAUSE command, 1197  
PCALC command, 1202  
PCGOPT command, 1205  
PCIRC command, 1207  
PCROSS command, 1209  
PDANL command, 1210  
PDCDF command, 1210  
PDCFLD command, 1212  
PDCLR command, 1214  
PDCMAT command, 1214  
PDCORR command, 1216  
PDDMCS command, 1217  
PDDOEL command, 1219  
PDEF command, 1221  
PDEXE command, 1224  
PDHIST command, 1226  
PDINQR command, 1227  
PDLHS command, 1228  
PDMETH command, 1230  
PDOT command, 1232  
PDPINV command, 1233  
PDPLOT command, 1234  
PDPROB command, 1235  
PDRESU command, 1236  
PDROPT command, 1237  
PDSAVE command, 1240  
PDSCAT command, 1241  
PDSSENS command, 1242  
PDSHIS command, 1243  
PDUSER command, 1244  
PDVAR command, 1245  
PDWRITE command, 1248  
PERBC2D command, 1248  
PERI command, 1250  
PERTURB command, 1250  
PFACT command, 1253

PGRAPH command, 1254  
PGRSET command, 1254  
PGSAVE command, 1256  
PGSELE command, 1257  
PGWRITE command, 1258  
PHYSICS command, 1260  
PIVCHECK command, 1262  
PLCAMP command, 1263  
PLCINT command, 1265  
PLCPLX command, 1266  
PLCRACK command, 1267  
PLDISP command, 1268  
PLESOL command, 1268  
PLETAB command, 1273  
PLF2D command, 1278  
PLFAR command, 1274  
PLFSS command, 1277  
PLLS command, 1279  
PLNEAR command, 1280  
PLNSOL command, 1281  
PLORB command, 1289  
PLOT command, 1289  
PLOTING command, 1290  
PLPAGM command, 1290  
PLPATH command, 1291  
PLSCH command, 1291  
PLSECT command, 1292  
PLSYZ command, 1293  
PLTD command, 1295  
PLTIME command, 1296  
PLTLINE command, 1297  
PLTRAC command, 1297  
PLVAR command, 1300  
PLVAROPT command, 1301  
PLVECT command, 1301  
PLVFRC command, 1304  
PLWAVE command, 1305  
PMAP command, 1306  
PMGTRAN command, 1306  
PMLOPT command, 1307  
PMLSIZE command, 1308  
PNGR command, 1310  
POINT command, 1313  
POLY command, 1314  
POUTRES command, 1316  
POWERH command, 1317  
PPATH command, 1318  
PRANGE command, 1319  
PRCAMP command, 1319  
PRCINT command, 1321  
PRCPLX command, 1322  
PRED command, 1323

PREENERGY command, 1324  
 PRERR command, 1325  
 PRESOL command, 1326  
 PRETAB command, 1331  
 PRFAR command, 1331  
 PRI2 command, 1334  
 PRIM command, 1335  
 PRINT command, 1335  
 PRISM command, 1336  
 PRITER command, 1337  
 PRJSOL command, 1337  
 PRNEAR command, 1338  
 PRNLD command, 1340  
 PRNSOL command, 1341  
 PROD command, 1346  
 PRORB command, 1347  
 PRPATH command, 1347  
 PRRFOR command, 1348  
 PRRSOL command, 1349  
 PRSECT command, 1349  
 PRSYZ command, 1351  
 PRTIME command, 1352  
 PRVAR command, 1352  
 PRVAROPT command, 1353  
 PRVECT command, 1353  
 PSCONTROL command, 1355  
 PSCR command, 1356  
 PSDCOM command, 1359  
 PSDFRQ command, 1359  
 PSDGRAPH command, 1360  
 PSDRES command, 1361  
 PSDSPL command, 1362  
 PSDUNIT command, 1362  
 PSDVAL command, 1363  
 PSDWAV command, 1364  
 PSEL command, 1365  
 PSMAT command, 1368  
 PMESH command, 1371  
 PSOLVE command, 1375  
 PSTRES command, 1379  
 PTR command, 1382  
 PTXY command, 1382  
 PVECT command, 1383

## Q

QDVAL command, 1385  
 QFACT command, 1386  
 QRDOPT command, 1386  
 QSOPT command, 1387  
 QUAD command, 1387  
 QUOT command, 1389

## R

R command, 1391  
 RACE command, 1392  
 RADOPT command, 1393  
 RAPPND command, 1394  
 RATE command, 1395  
 RBE3 command, 1396  
 RCON command, 1397  
 RDEC command, 1398  
 RDELE command, 1399  
 REAL command, 1400  
 REALVAR command, 1400  
 RECTNG command, 1401  
 reinforcing, 541  
 REMESH command, 1401  
 REORDER command, 1404  
 RESCONTROL command, 1407  
 RESET command, 1409  
 RESP command, 1410  
 RESUME command, 1412  
 RESVEC command, 1413  
 RESWRITE command, 1414  
 REXPORT command, 1415  
 REZONE command, 1416  
 RFORCE command, 1417  
 RIGID command, 1419  
 RIGRESP command, 1419  
 RIMPORT command, 1420  
 RLIST command, 1422  
 RMALIST command, 1422  
 RMANL command, 1423  
 RMASTER command, 1424  
 RMCAP command, 1424  
 RMCLIST command, 1425  
 RMFLVEC command, 1426  
 RMLVSCALE command, 1426  
 RMMLIST command, 1427  
 RMMRANGE command, 1427  
 RMMSELECT command, 1428  
 RMNDISP command, 1429  
 RMNEVEC command, 1430  
 RMODIF command, 1430  
 RMORE command, 1431  
 RMPORDER command, 1432  
 RMRESUME command, 1432  
 RMRGENERATE command, 1433  
 RMROPTIONS command, 1433  
 RMRPLOT command, 1434  
 RMRSTATUS command, 1435  
 RMSAVE command, 1435  
 RMSMPLE command, 1436  
 RMUSE command, 1437

RMXPORT command, 1438  
ROCK command, 1438  
ROSE command, 1439  
RPOLY command, 1439  
RPR4 command, 1440  
RPRISM command, 1441  
RPSD command, 1442  
RSFIT command, 1443  
RSOPT command, 1445  
RSPLIT command, 1446  
RSPLIT command, 1447  
RSPRINT command, 1448  
RSSIMS command, 1449  
RSTMAC command, 1450  
RSTOFF command, 1452  
RSURF command, 1453  
RSYMM command, 1454  
RSYS command, 1456  
RTHICK command, 1458

## S

SABS command, 1461  
SADD command, 1461  
SALLOW command, 1462  
SAVE command, 1463  
SBCLIST command, 1464  
SBCTRAN command, 1464  
SDELETE command, 1464  
SE command, 1465  
SECCONTROLS command, 1466  
SECDATA command, 1468  
SECFUNCTION command, 1490  
SECJOINT command, 1491  
SECLOCK command, 1493  
SECMODIF command, 1493  
SECNUM command, 1494  
SECOFFSET command, 1494  
SECPLOT command, 1497  
SECREAD command, 1500  
SECSTOP command, 1503  
SECTYPE command, 1504  
SECWRITE command, 1508  
SED command, 1509  
SEDLIST command, 1509  
SEEXP command, 1510  
SEGEN command, 1512  
SELIST command, 1514  
SELM command, 1515  
SELTOL command, 1515  
SENERGY command, 1516  
SEOPT command, 1517  
SESYMM command, 1518

SET command, 1523  
SETFGAP command, 1526  
SETRAN command, 1527  
SEXP command, 1529  
SF command, 1529  
SFA command, 1533  
SFACT command, 1536  
SFADELE command, 1537  
SFALIST command, 1538  
SFBEAM command, 1539  
SFCALC command, 1540  
SFCUM command, 1541  
SFDELE command, 1542  
SFE command, 1544  
SFEDELE command, 1548  
SFELIST command, 1549  
SFFUN command, 1551  
SFGRAD command, 1552  
SFL command, 1553  
SFLDELE command, 1556  
SFLEX command, 1557  
SFLIST command, 1558  
SFLLIST command, 1559  
SFSCALE command, 1559  
SFTRAN command, 1560  
SHELL command, 1562  
SHPP command, 1566  
SLIST command, 1570  
SLOAD command, 1572  
SMALL command, 1575  
SMAX command, 1577  
SMBODY command, 1578  
SMCONS command, 1579  
SMFOR command, 1579  
SMIN command, 1579  
SMOOTH command, 1580  
SMRTSIZE command, 1582  
SMSURF command, 1584  
SMULT command, 1584  
SNOPTION command, 1585  
SOCEAN command, 1587  
SOLCONTROL command, 1588  
SOLU command, 1593  
SOLUOPT command, 1595  
SOLVE command, 1595  
SORT command, 1596  
SOURCE command, 1596  
SPACE command, 1597  
SPADP command, 1597  
SPARM command, 1598  
SPCNOD command, 1599  
SPCTEMP command, 1599

SPDAMP command, 1600  
SPEC command, 1601  
SPFREQ command, 1601  
SPFSS command, 1602  
SPGRAPH command, 1604  
SPH4 command, 1604  
SPH5 command, 1605  
SPHERE command, 1605  
SPICE command, 1606  
SPLINE command, 1607  
SPLOT command, 1607  
SPMWRITE command, 1608  
SPOINT command, 1610  
SPOPT command, 1610  
SPREAD command, 1612  
SPSCAN command, 1612  
SPSWP command, 1614  
SPTOPT command, 1616  
SPUNIT command, 1617  
SPVAL command, 1618  
SQRT command, 1619  
SRSS command, 1620  
SSBT command, 1621  
SSLN command, 1623  
SSMT command, 1623  
SSPA command, 1624  
SSPB command, 1625  
SSPD command, 1626  
SSPE command, 1627  
SSPM command, 1627  
SSTIF command, 1628  
SSUM command, 1629  
STABILIZE command, 1629  
STAOPT command, 1631  
STAT command, 1632  
STEF command, 1636  
STORE command, 1638  
SUBSET command, 1639  
SUCALC command, 1641  
SUCR command, 1642  
SUDEL command, 1644  
SUEVAL command, 1644  
SUGET command, 1645  
SUMAP command, 1645  
SUMTYPE command, 1646  
SUPL command, 1647  
SUPR command, 1647  
SURESU command, 1648  
SUSAVE command, 1648  
SUSEL command, 1649  
SUVECT command, 1650  
SV command, 1651

SVPLOT command, 1651  
SVTYP command, 1652  
SWADD command, 1653  
SWDEL command, 1654  
SWGGEN command, 1654  
SWLIST command, 1655  
SYNCHRO command, 1656

## T

TALLOW command, 1659  
TB command, 1660  
TBCOPY command, 1699  
TBDATA command, 1699  
TBDELE command, 1700  
TBEO command, 1701  
TBFIELD command, 1701  
TBFT command, 1703  
TBLE command, 1712  
TBLIST command, 1712  
TBMODIF command, 1713  
TBPLOT command, 1713  
TBPT command, 1715  
TBTEMP command, 1716  
TCHG command, 1716  
TERM command, 1719  
THOPT command, 1720  
TIFF command, 1721  
TIME command, 1722  
TIMERANGE command, 1722  
TIMINT command, 1723  
TIMP command, 1724  
TINTP command, 1725  
TOCOMP command, 1728  
TODEF command, 1729  
TOEXE command, 1729  
TOFFST command, 1729  
TOFREQ command, 1730  
TOGRAPH command, 1731  
TOLIST command, 1732  
TOLOOP command, 1732  
TOPLOT command, 1734  
TOPRINT command, 1734  
TORQ2D command, 1735  
TORQC2D command, 1735  
TORQSUM command, 1736  
TORUS command, 1737  
TOSTAT command, 1738  
TOTAL command, 1738  
TOTYPE command, 1740  
TOVAR command, 1740  
TRANS command, 1742  
TRANSFER command, 1742

TREF command, 1744  
TRNOPT command, 1747  
TRPDEL command, 1749  
TRPLIS command, 1749  
TRPOIN command, 1750  
TRTIME command, 1750  
TSHAP command, 1751  
TSRES command, 1754  
TUNIF command, 1755  
TVAR command, 1756  
TYPE command, 1761  
TZAMESH command, 1762  
TZDELE command, 1762  
TZEGEN command, 1763

## U

UIMP command, 1771  
UNDELETE command, 1776  
UNDO command, 1777  
UNPAUSE command, 1779  
UPCOORD command, 1780  
UPGEOM command, 1781  
USRCAL command, 1784  
USRDOF command, 1785  
USRELEM command, 1786

## V

V command, 1791  
V2DOPT command, 1792  
VA command, 1793  
VADD command, 1794  
VARDEL command, 1795  
VARNAM command, 1795  
VATT command, 1796  
VCLEAR command, 1796  
VCROSS command, 1799  
VDDAM command, 1800  
VDELE command, 1801  
VDGL command, 1801  
VDOT command, 1802  
VDRAG command, 1802  
VEORIENT command, 1806  
VEXT command, 1807  
VFOPT command, 1810  
VFQUERY command, 1811  
VFSM command, 1812  
VGEN command, 1816  
VGET command, 1823  
VGLUE command, 1824  
VIMP command, 1825  
VINP command, 1827  
VINV command, 1827

VLIST command, 1830  
VLSCALE command, 1830  
VMESH command, 1832  
VOFFST command, 1833  
VOLUMES command, 1833  
VOVLAP command, 1837  
VPLOT command, 1838  
VPTN command, 1839  
VPUT command, 1843  
VROTAT command, 1845  
VSBA command, 1846  
VSBV command, 1847  
VSBW command, 1848  
VSEL command, 1852  
VSLA command, 1854  
VSUM command, 1855  
VSWEEP command, 1855  
VSYMM command, 1857  
VTCLR command, 1858  
VTDISC command, 1859  
VTEVAL, 1860  
VTFREQ command, 1860  
VTGEOM, 1861  
VTIN command, 1862  
VTMETH command, 1864  
VTMP command, 1865  
VTOP command, 1868  
VTPOST command, 1869  
VTRAN command, 1869  
VTREAL command, 1870  
VTRFIL command, 1872  
VTRSLT command, 1873  
VTSEC, 1875  
VTSFE command, 1877  
VTSL command, 1879  
VTSTAT command, 1880  
VTTEMP, 1880  
VTVMOD command, 1881  
VTYPE command, 1882

## W

WAVES command, 1887  
WERASE command, 1889  
WFRONT command, 1889  
WMID command, 1891  
WMORE command, 1891  
WPAVE command, 1892  
WPCSYS command, 1892  
WPLANE command, 1893  
WPOFFS command, 1894  
WPROTA command, 1894  
WPSTYL command, 1895



WRFULL command, 1896  
WRITE command, 1897  
WSORT command, 1898  
WSPRINGS command, 1899  
WSTART command, 1899

## **X**

XVAR command, 1903  
XVAROPT command, 1904

