Abagus/CAE 2016 [Viewport: 1] × Eile Model Viewport View Part Shape Feature Tools Plug-ins Help ★? 5 X 🕂 🕈 🔍 🔍 🔯 🔃 🗄 🛔 🗋 🔈 All 🗋 🗃 🖩 🖶 🇯 🖓 🖷 | 🔯 🛄 🧰 🚳 ! 🎒 🎒 👩 ! 🎲 Part defaults - 🗗 - 🖪 ` 🖥 📖 Ľ×׼↓, Ž×ĽzĽzĽz, 1 2 3 4 🎝 Module: 🗘 Part ✓ Model: ↓ Model-1 ✓ Part: ↓ Model Results 🚰 Model Database 🖌 🌲 🗈 🗞 🍟 L 📰 💠 Start Session × 🗏 🎎 Models (1) 6,2 Model-1

Model-1

Parts

Calibrations Create <u>|</u> (17) 🔹 With Standard/Explicit Model 👩 📫 With CFD Model 😃 🗄 😰 Sections 🚝 With Electromagnetic Model Profiles Abaqus/CAE 2 Assembly 🗃 Open Database 🛛 🗧 Run Script Steps (1) R -2016 Field Output Requests 👕 Start Tutorial

Indítsuk el az Abaqus CAE-t. A felugró DOS ablakkal ne törődjünk, de ne zárjuk be!



Kattintsunk a With Standard/Explicit Model-ra.

ALKATRÉSZEK RAJZOLÁSA

Készítsük el az első Part-ot. Klikk a Create Part ikonra:



A felugró ablakban nevet adhatunk neki. Állítsuk át úgy, ahogy a lenti képen van (2D Planar) majd Continue.



Előjön a szerkesztő ablak. Rajzoljuk meg a kontúrt alkotó egyeneseket. Klikk a Create Lines ikonra:



🖨 Abaqus/CAE 2016 [Viewport: 1]									-		×
Eile Model Viewport View Edit	Add Tools Plug-ins	lelp \ ?								-	5 ×
: 🗅 🗃 🖩 🖶 🛔			। 🕂 🔿 🔍 🔤	311:昌昌	All	De CR	967 👩				
			1 🗗 🗇 🗂								
					T × ×	竹美1	2 3 4	X.			
Model Results	Module: 🗘 Part	V Model: UModel-	1 🗸 Part: ‡	~		 					
Se Model Database	x:-50	v:23.75							Y		
🗆 🎎 Models (1)									•		
🖻 Model-1											
Parts	<u> </u>										
Calibrations	*1) ()								- F	×	
😰 Sections	f" #										
Profiles											
田 4 Assembly 国 4 Stens (1)	2 A										
Field Output Requests											
History Output Requests											
His Time Points											
	L 🛫 🗧										
a Interaction Properties	直 🗸										
Q Contact Controls	\square f(x)										
Contact Stabilizations											
Constraints	4) (+										
Connector Sections Fields	<u> </u>										
Amplitudes	L ₂ L ₂										
Loads	9										
BCs		Y									
Remeshing Rules											
Optimization Tasks											
Sketches											
🖻 🛊 Analysis											
Jobs 🗸	듣 🗙 Pick a starting p	oint for the lineor enter	X,Y:							25 SIN	
l nor redel database has h										,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,	
The model "Model-1" has be	en created.										
>>>											

Ezt követően be lehet írni a koordinátáit az egyes pontoknak. Adjuk meg az alábbi értékeket úgy, hogy mindegyik után **Enter**-t nyomunk.

0,0
20,0
20,30
13,30
10,20
15,17
12,7
0,7
0,0



Az utolsó után készítsük el a lekerekítéseket. Katt a Create Fillet ikonra majd adjunk meg a 2.0 értéket.



Válasszuk ki a két egyenest ami közé a lekerekítést kívánjuk szerkeszteni. Két lekerekítést készítsünk:

+ Abaqus/CAE 2016 [Viewport: 1]	- 🗆 X
Eile Model Viewport View Edit Add Tools Plug-ins Help ★?	_ 6 >
: 🗅 🚰 🖬 🖶 🛔 🛔	⊕ ♥ 및 ♥ 目 具 0
	¹ × × ¹ ↓ ¹ × ¹
Model Results Module: + Part V Model	il: 🗘 Model-1 🗸 Part: 🗘 🗸
Solution Model Database V 🗘 🗈 🗞 😵 + 💉	
Hodels (1)	R2.
- Brandar Bran	╶──┟──┟──┟──┟──┟──┟──┟──╽
The calibrations	
🔹 Sections	
+ Profiles	
Bolla Steps (1)	
Field Output Requests	
History Output Requests	B2 - R2
LE Adaptive Mesh Constraints	
Interactions	
H Contract Contracts	
$\frac{1}{2}$ Contact Initializations $\qquad \qquad \qquad$	
Contact Stabilizations	
Constraints	H
■ <i>F</i> Fields	
Amplitudes	
Loads	
Image Bos Image	
Remeshing Rules	······································
Contraction Tasks	
Appotations	
⊡‡‡ Analysis	
Jobs 🗸 🚽 🖉	end to be filleted
A new model database has been created	
The model "Model-1" has been created.	
>>>	

Ezt követően zárjuk be a szerkesztőt. Katt az X-re alul majd a Done-ra:



Ezzel kész az első 2D test:



Készítsük el a másikat is. Katt a Create Part ikonra. Ugyanolyan beállítások:

💠 Create Part	×
Name: Part-2	
Modeling Space	
⊖ 3D	○ Axisymmetric
Туре	Options
Deformable	
O Discrete rigid	Negativitable
 Analytical rigid 	None available
 Eulerian 	
Base Feature	
Shell	
() Wire	
○ Point	
Approximate size: 200	
Continue	Cancel

Vonalsereg készítése:

💠 Abaqus/CAE 2016 [Viewport: 1]										-	
🔄 Eile Model Viewport View Edit #	Add Tools Plug-ins	Help k ?									_ # ×
: 🗅 🗃 🖩 🖶 🛔 🏚			ें 🕂 🅐 🔍 🖳 💈	11日日日	All		Me CR	100			
			· # A A	0							
					i t ^y x x ^y	t 🗖 x t ^z x t ^y z z	ŸŤĚ 1	2 3 4 👗			
	Mandulas 🖉 Dast	Madah 🖹 Mada	1 Darts		· • • •	J ♥Z └→ └→ ←	_ <u>_</u> ~ x	- • • •			
Model Results		Model: 🖵 Mode	-1 V Part:								
Model Database V 🗘 🗈 🗞 😯	+ 🛷								Y		
Models (1)	0' 🗔										
B G Parts (1)	\odot										
🖉 Materials	00								2		X
Structure Calibrations	2/ 2/ (*** + †										
Sections	+ ++										
	<. I ₹.										
set (1)	<u>଼</u> ୬										
Field Output Requests	4										
History Output Requests											
Line Points											
	L 🚓										
- Haraction Properties	n 🔪										
Contact Controls											
Contact Initializations											
Constraints	A) 🚱										
Connector Sections	k 🥜										
■ <i>F</i> Fields											
Amplitudes											
	::::::::::::::::::::::::::::::::::::::										
Predefined Fields	u (1111)	Y									
Remeshing Rules											
Optimization Tasks		7 ×									
∽ ⊷ Sketches											
🖻 🔹 Analysis											
🚽 Jobs 🗸 🗸	Sketch the s	ection for the planar shell	lone							25	E IAAU U IO
< >	J Concert the s	center for the planar shell t								در	SINULIA
A new model database has be The model "Model-1" has been	een created. en created.										
<u> </u>											

Majd az értékek megadása:

Fick an end point for the lineor enter	<,V: [
	0,0	
	20,0	
	20,-7	
	13,-7	
	10,-17	
	15,-20	
	12.9,-27	
	5,-27	
	5,-7	
	0,-7	
	0,0	

💠 Abaqus/CAE 2016 [Viewport: 1]								-	ΟX
Eile Model Viewport View Edit A	Add <u>T</u> ools Plug-ins <u>H</u> elp \?								_ <i>6</i> ×
i 🗅 🗃 🗟 🚔 🛔		। 🕂 🅐 🔍 🖓	x 🔯 tl 🗄 🗄	🛔 i 📡 All			6 6		
0 100		- A A							
				: *	Y ↓ ↓ Yt ┌─\$ tZ ↓ tY		3 4 L		
				- L	→ ᆃᅴ ✦z ^ └ـ ♀ └	÷€1₂∧, ' 2	. 		
Model Results	Module: 😧 Part 🗹 Mod	el: 🗘 Model-1 🗹 Part: 🗘	~						
🧧 Model Database 🛛 🖕 🐑 🤴	+ 💉 X:-6.25, y:-12					······			• • • • • • • • • • • •
🗆 🎎 Models (1) 🔨	1 1								
Model-1									
Parts (I) Dart-1			V				V		
2 Materials	*1) 1 2								• X
S Calibrations	(~ ~ ~								
2 Sections	∠ →			1	e	<u> </u>)		
Profiles			Ľ						
Assembly	201								
Exercise Steps (1)	+-+ ` } `								
History Output Requests									
📇 Time Points									
ALE Adaptive Mesh Constraints	⊑ <u>‴</u> ⊙	+							
Laterations	🖺 🧭								
Contact Controls	$\prod_{i=1}^{n} f(x)$								
Contact Initializations				V	/				
Contact Stabilizations	4) (+			• (2				
Constraints	h 🛷								
Connector Sections									
Amplitudes						,			
Loads									
- 🛄 BCs	Y								
Predefined Fields	+								
Remeshing Rules		<u> </u>							
Optimization Tasks Sketcher		`		_ н					
Annotations									
🗄 🚼 Analysis 🗸 🗸	X Dick a starting point for the line	ineor enter X V:							2
< >	- rick a starting point for the	ine of entersyn						F	
A new model database has be The model "Model-1" has been	een created. en created.								
>>>									

Ha megvan akkor lekerekítések szintén 2-es sugárral. 2-t beírni majd Enter.



Lekerekítés helyeinek megadása:



Ha megvan akkor 🛛 aztán 🔤. Kész a második test:



ANYAGMODELL

A legördülő menüből válasszuk ki a Property-t:



Ezt követően más ikonsor lesz:



Kattintsunk a Create Material gombra: 12. Felugrik egy új ablak. Menüből az Elastic kiválasztása:

🜩 Edit Material	×
Name: Material-1	
Description:	
Material Behaviors	
General Markariant Thermal Electrical/Manustic Other	
Elasticity	7
Elestricity + Hyperelastic Damage for Tuckine Metals + Hyperform Damage for Fiber-Reinforced Composites + Hypelestic Damage for Elastomers + Porous Elastic Deformation Plasticity yscoelastic Bamping Egansion Egis Yiscosity	
ОК	Cancel

Felugró ablakban írjunk be 2800-t és 0.35-t a rugalmassági moduluszhoz és a Poisson-tényezőhöz:

🜩 Edit Material	×
Name: Material-1	
Description:	I
Material Behaviors	
Elastic	
General Mechanical Thermal Electrical/Magnetic Other	Image: A start and a start
Elastic	
Tune Instania	▼ Subortions
	- Suboptions
Number of field variables:	
Moduli time scale (for viscoelasticity): Long-term	
No compression	
No tension	
Data	
Young's Poisson's Modulus Ratio	
1 2800 0.35	
ОК	Cancel

Majd **OK**. Készítsünk egy **Section**-t. Katt az **1** ikonra. Felugró ablak:

💠 Create Section 🛛 🗙						
Name: Secti	ion-1					
Category	Туре					
Solid	Homogeneous					
◯ Shell	Generalized plane strain					
OBeam	Eulerian					
O Fluid	Composite					
O Other						
Continu	cancel					

Maradhat így, **Continue**. Új felugró ablak, fogadjuk el, **OK**.

🐥 Edit S	ection		×
Name: S	ection-1		
Type: S	olid, Homogeneous		
Material:	Material-1		Že
Plane	stress/strain thickne	ss: 1	
	ОК	Cancel	

Legördülő menüből válasszuk ki a Part 1-t:



Ezt követően az **Assign Section** ikon: **II**. Kéri, hogy jelöljük ki a tartományt. Katt egyszer a testen belülre (kijelöli pirossal):





A Part 2 esetén is végezzük el a fenti lépéseket:

🖨 Abaqus/CAE 2016 [Viewport: 1]	<u> </u>	- 🗆 X
Eile Model Viewport View Material	I Section Profile Composite Assign Special Feature Iools Plug-ins Help ▶?	_ 6 ×
: 🗅 🚅 🔲 🚔 🛔 🋔	: 中 🖉 🤤 🔂 🕇 🗄 🛔 : National 🖓 🖓 👘 🍘	🗐 : 🤲 Property defaults 🔍 📶 🕇 : 📭
		4 3
Model Results Material Library	Module: Property Model: Model-1 Part: Part-1	
🥌 Model Database 🔤 🌲 🖗	Part-1 Part-2	Y
🗆 🎎 Models (1) 🔨		•
□ Model-1 □ □ □ □ □ □ □ □ □		
Parts (2) Parts 1		
Part-2		
Ze Materials (1)		
Calibrations Br Sections (1)		_
+ Profiles		
Assembly		
ie offens Steps (1)		
History Output Requests		
Time Points		
Haterasticas		
Hinteraction Properties		
👯 Contact Controls		
Contact Initializations		
Connector Sections		
⊕ <i>F</i> Fields		
- 🕒 Loads	<u>胡夫</u> Υ	
BCs		
L Predefined Fields		
Remeshing Rules		
Annotations V		25 51001110
		<i>ps sindlin</i>
A new model database has be The model "Model-1" has bee	en created. en created.	
>>>		

Majd 💵 ikon. Belekattintani valahova és Done. Felugró ablakban OK. Ezzel kész az anyag hozzárendelése.

<u>ÖSSZEÁLLTÁS</u>

Assembly modul kiválasztása:



Megváltozik az ikonsor:

Abaqus/CAE 2016 [Viewport: 1]		- 🗆 ×
Eile Model Viewport View Instance	e Constraint Feature Iools Plug-ins Help % ?	_ 8 ×
i 🗅 🎜 🚍 🚔 🛔	🕂 🕂 🔍 🔍 🔀 11 🗄 📥 🖒 Ali 💦 🖓 🔂 😭 🗰 🎒 🦣 Assembly defaults	V 🛱 🛛 📙 🚍
Model Results	Module Assembly Model: Model: Model-1 Step: Initial	
Model Database		
E 🎎 Models (1)		
<u>Model-1</u> Parts (2)	er 🖞 🕰 and a state of the sta	
Part-1		×
Calibrations		-
Sections (1)		
। अमे Steps (1)		
Bat History Output Requests		
Time Points		
ALE Adaptive Mesh Constraints		
Contact Controls		
to contact Initializations		
Constraints		
Connector Sections		
# <i>F</i> Fields		
Loads	Y	
BCs	•	
Predefined Fields	$z \rightarrow x$	
Optimization Tasks		
L Sketches		
Annotations		35 SIMULIA
A new model database has be	en created.	
The model "Model-1" has been	m created.	
>>>		

Illesszük be a két alkatrészt. Katt az ^{ll} ikonra. Felugró ablakban nyomjuk le a shift-et, majd egérrel válasszuk ki mindkét alkatrészt, majd **OK**.



Mozgassuk feljebb a fentre kerülő alkatrészt. Katt a **Translate Instance** ikonra: E. Kéri, hogy válasszuk ki amit mozgatni akarunk. Egyszer katt az alsó alkatrészre majd alul **Done**. Ekkor kéri az elmozdulás vektor kezdő és végpontját. Kezdő pont legyen **0,0**:



Majd Enter, aztán a végpont: 0,57. Enter. Majd OK gomb. Katt az Auto-Fit View ikonra 2, hogy lássunk mindent.



STEP

Step modul kiválasztása:



Megváltozik az ikonsor:



Katt a Create Step ikonra: . Felugró új ablak:

💠 Create Step	×
Name: Step-1	
Insert new step after	
Initial	
Procedure type: General	\sim
Dynamic, Temp-disp, Explicit	^
Geostatic	
Heat transfer	
Mass diffusion	
Soils	
Static, General	
Static, Riks	~
٢	>
	_
0.0	

Fogadjuk el. Continue gomb. Felugró új ablakban kattintsunk át On-ra az NIgeom opciót:

😓 Edit Step		
lame: Step-1		
ype: Static, General		
Basic Incrementation	Other	
Description:		
Time period: 1		
Nigeom: On of la	rge displacements and a	affects subsequent steps.)
Automatic stabilization:	None	Y
include adiabatic nea	ting effects	

Aztán katt az **Incrementation** fülre. Váltsunk át **Fixed** típusra. A maximum increment számot írjuk át 1000-re. Az **increment size** pedig legyen 0.01. Majd **OK**.

🜩 Edit Step	×
Name: Step-1	
Type: Static, General	
Basic Incrementation Other	
Type: O Automatic Fixed	
Maximum number of increments: 1000	
Increment size: 0.01	
OK	

KONTAKT

Váltsunk át az Interaction modulra:



Megváltozik az ikonsor:



Katt a Create Interaction Property-re: 5. Felugró ablak:



Continue. Felugró ablakban Mechanical / Tangential Behavior menü:

🖨 Edit Contact Property	×
Name: IntProp-1	
Contact Property Options	
Mechanical Thermal Ele	ectrical Y
<u>Tangential Behavior</u>	
Normal Behavior h	defined for this contact property. will be used.
Damping	
Eracture Criterion	
Cohesive Behavior	
Geometric Properties	
OK	Canad
UK	Cancel

Az új ablakban a legördülőből válasszuk ki a Penalty-t:

≑ Edit Contact Prope	rty	×
Name: IntProp-1		
Contact Property Opt	ions	
Tangential Behavior		
Mechanical Therm	nal <u>E</u> lectrical	1
Tangential Behavior		
Friction formulation:	Frictionless	
	Frictionless	
	Penalty	
	Static-Kinetic Exponential Decay	
	Rough	
	Lagrange Multiplier (Standard only)	
	User-defined	
OI	К	Cancel

Adjuk meg a 0.2 értéket. Majd **OK**.

🚔 Edit Contact Property	×
Name: IntProp-1	
Contact Property Options	
Tangential Behavior	
Mechanical Ihermal Electrical	1
Tangential Behavior	
Friction formulation: Penalty	
Friction Change Change Electric Clin	
Directionality Alexanic Anicatronic (Standard only)	
Use slip-rate-dependent data	
Use contact-pressure-dependent data	
Use temperature-dependent data	
Number of field variables: 0	
Friction	
Coeff	
0.2	
QK Cancel	
Cancer	

A program fenti menüsorából Tools / Surface / Create. Új ablak jelenik meg:

💠 Create Surface 🛛 🗙
Name: Surf-1
Туре
Geometry O Mesh
Warning: Native mesh surfaces will be invalidated if the mesh changes.
Continue Cancel

Continue. Válasszuk ki a felső alkatrészen lévő kontaktba kerülő éleket. Majd alul Done.



A fenti menüsorból Tools / Surface / Create. Új ablak jelenik meg:



Continue. Válasszuk ki az alsó alkatrészen lévő kontaktba kerülő éleket. Majd alul Done.



Ezt követően Create Interaction ^{III}. Felugró ablakban a Step-nél Step-1, majd Surface-to-Surface contact opció és Continue.

Create Interaction	Х
Name: Int-1	
Step: Step-1	
Procedure: Static, General	
Types for Selected Step	
Surface-to-surface contact (Standard)	
Self-contact (Standard)	
Model change	
Standard-Explicit Co-simulation	
Pressure penetration	
L	
Continue Cancel	

Kéri, hogy válasszuk kis az első **Surface**-t. Jobb alul felnyithatjuk a **Surfaces...** ablakot:

Abaqus/CAE 2016 - Model Database: c:\te	emp\vem14.cae [Viewport: 1]	Special Feature Tools Divisions Help 169			
Die Monen Arexona Kien luteya	icuon Constraint Connector		L	1 砘 😒 🗊 🗊 🗊 🗞 Ass 2 z 🖄 , 1 2 3 4 🙏	embly defaults
Vodel Results	Module: 🖕 Interaction 🗸	Model: 🗘 Model-1 🗸 Step: 🗘 Step-1 🗸		· · · ·	
Model Database					×
A Amplitudes L Loads B L BCs (2) L Predefined Fields Remeshing Rules Sketches Sketches Amotations B Jobs (1) The model database has be The surface 'Surf-2' has The surface 'Surf-2' has	V V V V V V V V V V V V V V	► X ce individually ♥ (♥ Create surface: m.Surf-3 ed. 14. cae°.	; ;) Done	(Surfaces
Amplitudes Lads Predefined Fields Remeshing Rules Amplitudes Annotations Annotations The model database recovery The andel database has be The surface 'Surf-1' has The surface 'Surf-1' has The surface 'Surf-2' has	Y Z V Z V V V V V V V V V V V V V) Done	(Surfaces
Amplitudes Lads Pedefined Fields Remeshing Rules Annotations Annotations The model database recover The model database has be The surface Surf-2' has The surface 'Surf-2' has	 Select the master sufficiency operation has completeness as weld to "c:\templeteness been created (4 edges). been created (4 edges). 	★ ce individually ♥ (Create surface: m_surface) ed, 14. cae*. Eligible Surfaces Surfaces below may contain faces. Name filter: ♥ Name Surface Surf-1 Surface Surf-2 Surface <) Done	(Surfaces

Válasszuk ki a Surf-1-t majd Continue. Kéri a második felületet. A lenti opcióból válasszuk a Surface-t:

Choose the slave type: Surface Node Region

Válasszuk ki a második felületet majd Continue. Új ablakra OK.

Edit Interaction	х
Name: Int-1	
Type: Surface-to-surface contact (Standard)	
Step: Step-1 (Static, General)	
Master surface: Surf-1 Slave surface: Surf-2	
Sliding formulation: Finite sliding Small sliding	
Discretization method: Surface to surface	
Exclude shell/membrane element thickness	
Degree of smoothing for master surface: 0.2	
Use supplementary contact points: Selectively Never Always	
Contact tracking: Two configurations (path) Single configuration (st	ate)
Slave Adjustment Surface Smoothing Clearance Bonding	
No adjustment	
O Adjust only to remove overclosure	
○ Specify tolerance for adjustment zone: 0	
○ Adjust slave nodes in set:	
Contact interaction property: IntProp-1	묩
Options: Interference Fit	
Contact controls: (Default)	
Active in this step	
OK	

LOADS

Térjünk át a Load modulra:





Katt a Create Boundary Condition ikonra: . Felugró ablak: A Step-nél az Initial legyen!

Create Boundary Condition Name: Initial Properture:		
Category Mechanical Fluid Electrical/Magnetic Other	Types for Selected Step Symmetry/Antisymmetry/Encastre Displacement/Rotation Velocity/Angular velocity Acceleration/Angular acceleration Connector velocity Connector velocity Connector acceleration	
Continue	Cancel	

Continue. (*Ha a Surface generálásból adódó ablak még aktív akkor azt a Dismiss gombbal eltűntethetjük*). Válasszuk ki a felső élt majd **Done**.



Felugró ablakban ENCASTRE, majd OK:

💠 Edit Boundary Condition	×
Name: BC-1	
Type: Symmetry/Antisymmetry/Encastre	
Step: Initial	
Region: Set-1	
CSVS: (Global) 🔉 🙏	
XSYMM (U1 = UR2 = UR3 = 0)	
YSYMM (U2 = UR1 = UR3 = 0)	
ZSYMM (U3 = UR1 = UR2 = 0)	
XASYMM (U2 = U3 = UR1 = 0; Abaqus/Standard	only)
YASYMM (U1 = U3 = UR2 = 0; Abaqus/Standard	only)
ZASYMM (U1 = U2 = UR3 = 0; Abaqus/Standard	only)
O PINNED (U1 = U2 = U3 = 0)	
ENCASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0	0)
OK	

Katt a Create Boundary Condition ikonra: . Felugró ablak: A Step-nél az Step-1 legyen!

A listából válasszuk ki a **Displacement/Rotation**-t. **Continue**.

🜩 Cri	eate Boundary Con	dition	×
Name:	BC-2		
Step:	Step-1	~	
Proced	ure: Static, Genera	l .	
Categ	gory	Types for Selected Step	
 Ma Flu Ele Oti 	echanical iid ectrical/Magnetic her	Symmetry/Antisymmetry/Encastre Displacement/Rotation Velocity/Angular velocity Connector displacement Connector velocity	
	Continue	Cancel	

Válasszuk ki az alsó élt és Done.



Felugró ablakban az U2-höz 20 máshova 0 és OK.

🖨 Edit Bou	ndary Condition	×		
Name: BC-	2			
ype: Disp	lacement/Rotation			
tep: Step	o-1 (Static, General)			
Region: Set-	2			
CSYS: (Glo	obal) 🔉 🙏			
Distribution:	Uniform 🖌	f(x)		
Z U1:	0			
✓ U2:	20			
🛛 UR3:	0	radians		
Amplitude:	(Ramp) 🗸	Ъ		
lote: The displacement value will be maintained in subsequent steps.				
ОК	Cance	I		



Mesh modul:



Változik az ikonsor:



Válasszuk ki a Part-1-t: Katt a Part gombra és legördülő menüből válasszuk ki.



Assign Mesh Control-nál adjuk meg, hogy csak Quad elemeket használjon. Felugró ablakban Quad és alábbi opciók (szedjük most ki az *Use mapped meshing…* opciót):

⇔ Mesh Controls		×
Element Shape		
Quad () Quad	-dominated () Tri	
Technique	Algorithm	
🔿 As is	O Medial axis	
Free	Minimize the mesh transition	
O Structured	Advancing front	
O Sweep	Use mapped meshing where appropriate	
 Multiple 		
ОК	Defaults Cancel	

Ezt követően Seed Part 🖳 Felugróban legyen 1 az elemméret:

💠 Global Seeds 🛛 🗙
Sizing Controls
Approximate global size: 1
Curvature control
Maximum deviation factor (0.0 < h/L < 1.0): 0.1
(Approximate number of elements per circle: 8)
Minimum size control
By fraction of global size (0.0 < min < 1.0) 0.1
O By absolute value (0.0 < min < global size) 0.1
OK Apply Defaults Cancel

Majd OK. Aztán Done alul. Ezt követően Mesh Part 🖳 és alul Yes:

- X OK to mesh the part? Yes No

Eredmény:



Másik Part-nál is végezzük el a fenti műveleteket.

Ha átkattintunk az Assembly gombra akkor együtt látjuk:



<u>JOB</u>

Job modul kiválasztása:



Create Job ikon: 4. Felugró ablakban adhatunk nevet neki: vem14. Continue.



Felugró ablak:

🖨 Edit Job					×
Name: vem1-	4				
Model: Mode	I-1				
Analysis produ	ict: Abaqu	s/Standard			
Description:					
Submission	General	Memory	Parallelization	Precision	
Job Type					
Full analy	/sis				
O Recover	(Explicit)				
○ Restart					
Run Mode					
Backgrou	nd 🔿 Que	ue:	Hos	t name: s	
Submit Tim	ie				
Immedia	tely				
🔿 Wait:	hrs. m	in.			
O At:		- Q -			
	ОК]		Cance	I

OK. Mentsük el a modellt. File / Save as.

Job manager ikon: . Látjuk a kész Job-okat:

Name	Model	Туре	Status	Write Input
vem14	Model-1	Full Analysis	None	Data Check
				Submit
				Continue
				Monitor
				Results

Katt a Submit ikonra jobb oldalon. Ha végzett akkor átvált Completed-re.

Name	Model	Туре	Status	Write Inp
vem14	Model-1	Full Analysis	Completed	Data Che
				Submit
				Continu
				Monitor
				Results

Katt a **Results** gombra. Alapból a kezdeti alakot mutatja:



Katt a **Plot Contours on Deformed Shape** ikonra: **S**. Standard beállítás, hogy ekkor a Mises-féle egyenértékű feszültséget mutatja a Step végén:



Ha a megoldás során lévő értékek is érdekelnek, akkor használhatjuk a **Frame Selector** csúszkát jobb oldalon felül: **S**. Felugró ablak:





Animáció: 🛎 Ki/Be kapcsolható.

Kérdezzük le a reakcióerőket. Reakció erők ébrednek a node-okban. Deformálatlan alak: [■]. Auto Fit [⊠]. **Create XY Data** ^{III}. Felugróban ablakban **ODB field output**. Majd **Continue**.



Felugró ablakban a Position-nál Unique Nodal, majd RF2 a lenti választható értékekből

- Al bac	a from ODB Field Output	
Steps/Fra	mes	
Note: XY	Data will be extracted from the active steps/frames	Active Steps/Frames
Variables	Elements/Nodes	
Output	Variables	
Position:	Unique Nodal 🗸	
Click che	eckboxes or edit the identifiers shown next to Edit belo	w.
	PEEQ: Equivalent plastic strain	^
	PEMAG: Magnitude of plastic strain	
▼ ■	RF: Reaction force	
	Magnitude	
	L RF1	
	✓ KF2 C. Channel and an annual state	
	5: Stress components	
	o: spatial displacement	~
Edit: RF.	RF2	
	oint All Select Settinger	

Váltsunk át az **Element/Nodes** fülre. Ott válasszuk ki a **Node sets**-t majd a **Highligth items in viewport** opciót kapcsoljuk be. Válasszuk ki az a **Set**-t ami az alsó élhez tartozik (jelen segédletben SET-2). Mutatja az ábrán pirossal.



Aztán **Plot**. **Dissmiss**-sel eltűntethetjük az ablakot. Ez most a reakcióerőket (N/mm dimenzióban tekintettel a síkfeladatra) mutatja minden **node**-hoz.



Ha minket az összegük érdekel (vagyis az eredő erő a felső élen) akkor össze lehet adni. Create XY Data ⊞, majd Operate on XY data. Continue.

🖨 Create XY Data 🛛 🗙
Source
ODB history output
ODB field output
O Thickness
○ Free body
Operate on XY data
O ASCII file
○ Keyboard
○ Path
Continue Cancel

Jobb oldali függvénylistából válasszuk ki a sum((A, A, ...))-t, ezzel bekerül az egyenlet mezőbe.



Válasszuk ki az első elemet majd shift lenyomása után az utolsó elemet. Ezzel az összes ki van jelölve. Add to Expression.

er an expression by typing and selecting XY Data and Operators below. mple: maxEnvelope("XYData-2", "XYData-4") * 2.5 + "XYData-5" RT-1-1 N: 29", "RF:RF2 PL PART-1-1 N: 30", "RF:RF2 PL PART-1-1 N: 31", "RF:RF2 PL PART-1 N: 33", "RF:RF2 PL PART-1-1 N: 34", "RF:RF2 PL PART-		
mple: maxEnvelope("XYData-2", "XYData-4") * 2.5 + "XYData-5" RT-1-1 N: 29", "_RF:RF2 PI: PART-1-1 N: 30", "_RF:RF2 PI: PART-1-1 N: 31 IF:RF2 PI: PART-1-1 N: 33", "_RF:RF2 PI: PART-1-1 N: 34", "_RF:RF2 PI: PAF		
RT-1-1 N: 29", "_RF:RF2 PI: PART-1-1 N: 30", "_RF:RF2 PI: PART-1-1 N: 31" IF:RF2 PI: PART-1-1 N: 33", "_RF:RF2 PI: PART-1-1 N: 34", "_RF:RF2 PI: PAF		
RT-1-1 N: 36", "_RF:RF2 PI: PART-1-1 N: 37", "_RF:RF2 PI: PART-1-1 N: 38	, _RF: RT-1-1 , _RF:	RF2 PI: PART-1-1 N: 32", N: 35", "_RF:RF2 PI: RF2 PI: PART-1-1 N: 39"))
Y Data		Operators
ame filter:	· `@ `	A - XYData, float, or integer
ame Description	^	X - XYData
RF:RF2 PI: PART-1 From Field Data: RF:RF2 at part instance PART-1-1 nod	le	l - integer
RF:RF2 PI: PART-1 From Field Data: RF:RF2 at part instance PART-1-1 nod	le	F - float
RF:RF2 PI: PART-1 From Field Data: RF:RF2 at part instance PART-1-1 nod	le	sineButterworthFilter(X,F)
RF:RF2 PI: PART-' From Field Data: RF:RF2 at part instance PART-1-1 nod	le	sin(A)
RF:RF2 PI: PART- [*] From Field Data: RF:RF2 at part instance PART-1-1 nod	e	sinh(A)
RF:RF2 PI: PART-1 From Field Data: RF:RF2 at part instance PART-1-1 nod	e	smooth(X,I)
REREZ PILPART - From Field Data: REREZ at part instance PART-T-T nod	e	sart(A)
RF:RF2 PI: PARTE FIOM FIEld Data: RF:RF2 at part instance PARTETED on PE/PE2 DI: DAPT 1 From Field Data: PE/PE2 at part instance DAPT 1.1 pod	e	srss((X,X,))
RF:RE2 PI: PART- From Field Data: RE:RE2 at part instance PART-1-1 nod	-	sum((A,A,))
RF:RF2 PI: PART-' From Field Data: RF:RF2 at part instance PART-1-1 nod	e	swap(X)
RF:RF2 PI: PART-1 From Field Data: RF:RF2 at part instance PART-1-1 nod	le	tan(A)
SERF2 PT PART- From Field Data: RF:RF2 at part instance PART-1-1 nod	le 🗸	tanh(A)
dd to Expression 🗌 Skip checks		vectorMagnitude(X,X,X)

Plot Expression. A Time változó (ami jelen esetben 0...1 volt) mentén a reakcióerő (Y irányú) változása:



A megoldás elég "darabos" a durva háló miatt. Az eredmények sűrűbb hálóval:



A feszültség eloszláson a terhelés során most szebben látszik a pontszerű érintkezés az elemek között:



Az eredő Y reakcióerő változása a terhelés során:



SZKRIPT

A modell felépítése Python szkript segítéségével.

Indítsuk el a szoftvert.



Zárjuk be a kezdő kisablakot. Majd alul kattintsunk a 🔛 ikonra. Ezt követően lehetőségünk van a prompt helyére Python parancsokat beírni.



Jelöljük ki a lenti kódrészt és illesszük be CTRL+C és CTRL+V kombinációval. Láthatjuk, ahogy végigfut és elkészít mindent. Most már futtathatjuk is. Váltsunk át a **Job** modulra és futtassuk le az elkészített **Job**-ot.

```
# -*- coding: mbcs -*-
# Abaqus/CAE Release 2016 replay file
# Internal Version: 2015 09 24-22.31.09 126547
# Run by Attila Kossa on Fri Dec 01 14:30:53 2017
#
# from driverUtils import executeOnCaeGraphicsStartup
# executeOnCaeGraphicsStartup()
#: Executing "onCaeGraphicsStartup()" in the site directory ...
from abaqus import *
from abaqusConstants import *
session.Viewport(name='Viewport: 1', origin=(0.0, 0.0), width=184.004180908203,
    height=117.765747070313)
session.viewports['Viewport: 1'].makeCurrent()
session.viewports['Viewport: 1'].maximize()
from caeModules import
from driverUtils import executeOnCaeStartup
executeOnCaeStartup()
session.viewports['Viewport: 1'].partDisplay.geometryOptions.setValues(
    referenceRepresentation=ON)
Mdb()
#: A new model database has been created.
#: The model "Model-1" has been created.
session.viewports['Viewport: 1'].setValues(displayedObject=None)
cliCommand("""session.journalOptions.setValues(replayGeometry=COORDINATE, recoverGeometry=COORDINATE)""")
s = mdb.models['Model-1'].ConstrainedSketch(name='__profile__',
    sheetSize=200.0)
g, v, d, c = s.geometry, s.vertices, s.dimensions, s.constraints
s.setPrimaryObject(option=STANDALONE)
s.Line(point1=(0.0, 0.0), point2=(20.0, 0.0))
s.HorizontalConstraint(entity=g.findAt((10.0, 0.0)), addUndoState=False)
s.Line(point1=(20.0, 0.0), point2=(20.0, 30.0))
s.VerticalConstraint(entity=g.findAt((20.0, 15.0)), addUndoState=False)
s.PerpendicularConstraint(entity1=g.findAt((10.0, 0.0)), entity2=g.findAt((
    20.0, 15.0)), addUndoState=False)
s.Line(point1=(20.0, 30.0), point2=(13.0, 30.0))
s.HorizontalConstraint(entity=g.findAt((16.5, 30.0)), addUndoState=False)
s.PerpendicularConstraint(entity1=g.findAt((20.0, 15.0)), entity2=g.findAt((
    16.5, 30.0)), addUndoState=False)
s.Line(point1=(13.0, 30.0), point2=(10.0, 20.0))
s.Line(point1=(10.0, 20.0), point2=(15.0, 17.0))
s.Line(point1=(15.0, 17.0), point2=(12.0, 7.0))
s.Line(point1=(12.0, 7.0), point2=(0.0, 7.0))
s.HorizontalConstraint(entity=g.findAt((6.0, 7.0)), addUndoState=False)
s.Line(point1=(0.0, 7.0), point2=(0.0, 0.0))
s.VerticalConstraint(entity=g.findAt((0.0, 3.5)), addUndoState=False)
s.PerpendicularConstraint(entity1=g.findAt((6.0, 7.0)), entity2=g.findAt((0.0,
    3.5)), addUndoState=False)
s.FilletByRadius(radius=2.0, curve1=g.findAt((16.5, 30.0)), nearPoint1=(
    15.6112060546875, 29.3963203430176), curve2=g.findAt((11.5, 25.0)),
    nearPoint2=(12.6639556884766, 26.9160079956055))
s.FilletByRadius(radius=2.0, curve1=g.findAt((11.286204, 24.287348)),
    nearPoint1=(11.4666366577148, 23.4251937866211), curve2=g.findAt((12.5,
    18.5)), nearPoint2=(12.1113433837891, 19.107608795166))
p = mdb.models['Model-1'].Part(name='Part-1', dimensionality=TWO D PLANAR,
    type=DEFORMABLE BODY)
p = mdb.models['Model-1'].parts['Part-1']
p.BaseShell(sketch=s)
s.unsetPrimaryObject()
p = mdb.models['Model-1'].parts['Part-1']
session.viewports['Viewport: 1'].setValues(displayedObject=p)
del mdb.models['Model-1'].sketches[' profile ']
s1 = mdb.models['Model-1'].ConstrainedSketch(name=' profile ',
   sheetSize=200.0)
g, v, d, c = sl.geometry, sl.vertices, sl.dimensions, sl.constraints
s1.setPrimaryObject(option=STANDALONE)
s1.Line(point1=(0.0, 0.0), point2=(20.0, 0.0))
s1.HorizontalConstraint(entity=g.findAt((10.0, 0.0)), addUndoState=False)
s1.Line(point1=(20.0, 0.0), point2=(20.0, -7.0))
s1.VerticalConstraint(entity=g.findAt((20.0, -3.5)), addUndoState=False)
s1.PerpendicularConstraint(entity1=g.findAt((10.0, 0.0)), entity2=g.findAt((
    20.0, -3.5)), addUndoState=False)
```

```
s1.Line(point1=(20.0, -7.0), point2=(13.0, -7.0))
sl.HorizontalConstraint(entity=g.findAt((16.5, -7.0)), addUndoState=False)
s1.PerpendicularConstraint(entity1=g.findAt((20.0, -3.5)), entity2=g.findAt((
    16.5, -7.0)), addUndoState=False)
s1.Line(point1=(13.0, -7.0), point2=(10.0, -17.0))
s1.Line(point1=(10.0, -17.0), point2=(15.0, -20.0))
sl.Line(point1=(15.0, -20.0), point2=(12.9, -27.0))
sl.Line(point1=(12.9, -27.0), point2=(5.0, -27.0))
sl.HorizontalConstraint(entity=g.findAt((8.95, -27.0)), addUndoState=False)
s1.Line(point1=(5.0, -27.0), point2=(5.0, -7.0))
s1.VerticalConstraint(entity=g.findAt((5.0, -17.0)), addUndoState=False)
s1.PerpendicularConstraint(entity1=g.findAt((8.95, -27.0)), entity2=g.findAt((
    5.0, -17.0)), addUndoState=False)
s1.Line(point1=(5.0, -7.0), point2=(0.0, -7.0))
s1.HorizontalConstraint(entity=g.findAt((2.5, -7.0)), addUndoState=False)
s1.PerpendicularConstraint(entity1=g.findAt((5.0, -17.0)), entity2=g.findAt((
    2.5, -7.0)), addUndoState=False)
s1.Line(point1=(0.0, -7.0), point2=(0.0, 0.0))
s1.VerticalConstraint(entity=g.findAt((0.0, -3.5)), addUndoState=False)
s1.PerpendicularConstraint(entity1=g.findAt((2.5, -7.0)), entity2=g.findAt((
    0.0, -3.5)), addUndoState=False)
s1.FilletByRadius(radius=2.0, curve1=g.findAt((12.5, -18.5)), nearPoint1=(
    14.3217849731445, -19.1994743347168), curve2=g.findAt((13.95, -23.5)),
    nearPoint2=(14.690185546875, -22.2309684753418))
s1.FilletByRadius(radius=2.0, curve1=g.findAt((8.95, -27.0)), nearPoint1=(
    10.7298202514648, -27.0997352600098), curve2=g.findAt((13.726565,
    -24.244782)), nearPoint2=(13.4928665161133, -25.1706008911133))
p = mdb.models['Model-1'].Part(name='Part-2', dimensionality=TWO_D_PLANAR,
    type=DEFORMABLE BODY)
p = mdb.models['Model-1'].parts['Part-2']
p.BaseShell(sketch=s1)
s1.unsetPrimaryObject()
p = mdb.models['Model-1'].parts['Part-2']
session.viewports['Viewport: 1'].setValues(displayedObject=p)
del mdb.models['Model-1'].sketches[' profile
session.viewports['Viewport: 1'].partDisplay.setValues(sectionAssignments=ON,
    engineeringFeatures=ON)
session.viewports['Viewport: 1'].partDisplay.geometryOptions.setValues(
    referenceRepresentation=OFF)
mdb.models['Model-1'].Material(name='Material-1')
mdb.models['Model-1'].materials['Material-1'].Elastic(table=((2800.0, 0.35), ))
mdb.models['Model-1'].HomogeneousSolidSection(name='Section-1',
    material='Material-1', thickness=None)
p = mdb.models['Model-1'].parts['Part-1']
session.viewports['Viewport: 1'].setValues(displayedObject=p)
p = mdb.models['Model-1'].parts['Part-1']
f = p.faces
faces = f.findAt(((4.0, 4.666667, 0.0), ))
region = p.Set(faces=faces, name='Set-1')
p = mdb.models['Model-1'].parts['Part-1']
p.SectionAssignment(region=region, sectionName='Section-1', offset=0.0,
    offsetType=MIDDLE SURFACE, offsetField='',
    thicknessAssignment=FROM SECTION)
p = mdb.models['Model-1'].parts['Part-2']
session.viewports['Viewport: 1'].setValues(displayedObject=p)
p = mdb.models['Model-1'].parts['Part-2']
f = p.faces
faces = f.findAt(((1.666667, -4.6666667, 0.0), ))
region = p.Set(faces=faces, name='Set-1')
p = mdb.models['Model-1'].parts['Part-2']
p.SectionAssignment(region=region, sectionName='Section-1', offset=0.0,
    offsetType=MIDDLE SURFACE, offsetField='',
    thicknessAssignment=FROM SECTION)
a = mdb.models['Model-1'].rootAssembly
session.viewports['Viewport: 1'].setValues(displayedObject=a)
session.viewports['Viewport: 1'].assemblyDisplay.setValues(
    optimizationTasks=OFF, geometricRestrictions=OFF, stopConditions=OFF)
a = mdb.models['Model-1'].rootAssembly
a.DatumCsysByDefault(CARTESIAN)
p = mdb.models['Model-1'].parts['Part-1']
a.Instance(name='Part-1-1', part=p, dependent=ON)
p = mdb.models['Model-1'].parts['Part-2']
a.Instance(name='Part-2-1', part=p, dependent=ON)
a = mdb.models['Model-1'].rootAssembly
a.translate(instanceList=('Part-2-1', ), vector=(0.0, 57.0, 0.0))
#: The instance Part-2-1 was translated by 0., 57., 0. with respect to the assembly coordinate system
session.viewports['Viewport: 1'].view.fitView()
session.viewports['Viewport: 1'].assemblyDisplay.setValues(
    adaptiveMeshConstraints=ON)
```

```
mdb.models['Model-1'].StaticStep(name='Step-1', previous='Initial',
    maxNumInc=1000, timeIncrementationMethod=FIXED, initialInc=0.01,
    noStop=OFF, nlgeom=ON)
session.viewports['Viewport: 1'].assemblyDisplay.setValues(step='Step-1')
session.viewports['Viewport: 1'].assemblyDisplay.setValues(interactions=ON,
    constraints=ON, connectors=ON, engineeringFeatures=ON,
    adaptiveMeshConstraints=OFF)
mdb.models['Model-1'].ContactProperty('IntProp-1')
mdb.models['Model-1'].interactionProperties['IntProp-1'].TangentialBehavior(
    formulation=PENALTY, directionality=ISOTROPIC, slipRateDependency=OFF,
    pressureDependency=OFF, temperatureDependency=OFF, dependencies=0, table=((
    0.2, ), ), shearStressLimit=None, maximumElasticSlip=FRACTION,
    fraction=0.005, elasticSlipStiffness=None)
#: The interaction property "IntProp-1" has been created.
a = mdb.models['Model-1'].rootAssembly
s1 = a.instances['Part-2-1'].edges
side1Edges1 = s1.findAt(((12.040758, 30.101425, 0.0), ), ((14.246746,
    34.489153, 0.0), ), ((14.167268, 37.373443, 0.0), ), ((10.916617, 39.45003,
    0.0), ))
a.Surface(side1Edges=side1Edges1, name='Surf-1')
#: The surface 'Surf-1' has been created (4 edges).
a = mdb.models['Model-1'].rootAssembly
s1 = a.instances['Part-1-1'].edges
side1Edges1 = s1.findAt(((13.859242, 29.898575, 0.0), ), ((12.041024,
    26.803413, 0.0), ), ((10.364043, 20.836884, 0.0), ), ((12.250148,
    18.649911, 0.0), ))
a.Surface(side1Edges=side1Edges1, name='Surf-2')
#: The surface 'Surf-2' has been created (4 edges).
a = mdb.models['Model-1'].rootAssembly
region1=a.surfaces['Surf-1']
a = mdb.models['Model-1'].rootAssembly
region2=a.surfaces['Surf-2']
mdb.models['Model-1'].SurfaceToSurfaceContactStd(name='Int-1',
    createStepName='Step-1', master=region1, slave=region2, sliding=FINITE,
    thickness=ON, interactionProperty='IntProp-1', adjustMethod=NONE,
initialClearance=OMIT, datumAxis=None, clearanceRegion=None)
#: The interaction "Int-1" has been created.
session.viewports['Viewport: 1'].assemblyDisplay.setValues(loads=ON, bcs=ON,
    predefinedFields=ON, interactions=OFF, constraints=OFF,
    engineeringFeatures=OFF)
session.viewports['Viewport: 1'].assemblyDisplay.setValues(step='Initial')
a = mdb.models['Model-1'].rootAssembly
e1 = a.instances['Part-2-1'].edges
edges1 = e1.findAt(((5.0, 57.0, 0.0), ))
region = a.Set(edges=edges1, name='Set-1')
mdb.models['Model-1'].EncastreBC(name='BC-1', createStepName='Initial',
region=region, localCsys=None)
session.viewports['Viewport: 1'].assemblyDisplay.setValues(step='Step-1')
a = mdb.models['Model-1'].rootAssembly
e1 = a.instances['Part-1-1'].edges
edges1 = e1.findAt(((5.0, 0.0, 0.0), ))
region = a.Set(edges=edges1, name='Set-2')
mdb.models['Model-1'].DisplacementBC(name='BC-2', createStepName='Step-1',
    region=region, u1=0.0, u2=20.0, ur3=0.0, amplitude=UNSET, fixed=OFF,
    distributionType=UNIFORM, fieldName='', localCsys=None)
session.viewports['Viewport: 1'].assemblyDisplay.setValues(mesh=ON, loads=OFF,
    bcs=OFF, predefinedFields=OFF, connectors=OFF)
session.viewports['Viewport: 1'].assemblyDisplay.meshOptions.setValues(
    meshTechnique=ON)
p = mdb.models['Model-1'].parts['Part-2']
session.viewports['Viewport: 1'].setValues(displayedObject=p)
session.viewports['Viewport: 1'].partDisplay.setValues(sectionAssignments=OFF,
    engineeringFeatures=OFF, mesh=ON)
session.viewports['Viewport: 1'].partDisplay.meshOptions.setValues(
   meshTechnique=ON)
p = mdb.models['Model-1'].parts['Part-1']
session.viewports['Viewport: 1'].setValues(displayedObject=p)
p = mdb.models['Model-1'].parts['Part-1']
f = p.faces
pickedRegions = f.findAt(((4.0, 4.666667, 0.0), ))
p.setMeshControls(regions=pickedRegions, elemShape=QUAD, allowMapped=False)
p = mdb.models['Model-1'].parts['Part-1']
p.seedPart(size=1.0, deviationFactor=0.1, minSizeFactor=0.1)
p = mdb.models['Model-1'].parts['Part-1']
p.generateMesh()
p = mdb.models['Model-1'].parts['Part-2']
session.viewports['Viewport: 1'].setValues(displayedObject=p)
p = mdb.models['Model-1'].parts['Part-2']
f = p.faces
```

```
pickedRegions = f.findAt(((1.666667, -4.666667, 0.0), ))
p.setMeshControls(regions=pickedRegions, elemShape=QUAD, allowMapped=False)
p = mdb.models['Model-1'].parts['Part-2']
p.seedPart(size=1.0, deviationFactor=0.1, minSizeFactor=0.1)
p = mdb.models['Model-1'].parts['Part-2']
p.generateMesh()
a = mdb.models['Model-1'].rootAssembly
session.viewports['Viewport: 1'].setValues(displayedObject=a)
a1 = mdb.models['Model-1'].rootAssembly
al.regenerate()
session.viewports['Viewport: 1'].assemblyDisplay.setValues(mesh=OFF)
session.viewports['Viewport: 1'].assemblyDisplay.meshOptions.setValues(
   meshTechnique=OFF)
mdb.Job(name='vem14', model='Model-1', description='', type=ANALYSIS,
   atTime=None, waitMinutes=0, waitHours=0, queue=None, memory=90,
    memoryUnits=PERCENTAGE, getMemoryFromAnalysis=True,
    explicitPrecision=SINGLE, nodalOutputPrecision=SINGLE, echoPrint=OFF,
```

```
modelPrint=OFF, contactPrint=OFF, historyPrint=OFF, userSubroutine='',
scratch='', resultsFormat=ODB, multiprocessingMode=DEFAULT, numCpus=1,
numGPUs=0)
```