Ansys Tutorial: Plate with a Hole

1. This tutorial will perform a 2D linear elastic static analysis of a plate with a hole. The geometry is shown below



- a. b. The plate is very thin (plane stress analysis applies)
- c. The Young's modulus is 200e3 MPa and Poisson's ratio is 0.3
- 2. To open ANSYS, go to the Window Menu **and select**:
 - a. ANSYS 19.0> Workbench 19.0
 - b. Note the version number might be different (Workbench is usually at the bottom of the menu)
- 3. A Workbench project will open

🔥 Unsaved Project - Workbench	
File View Tools Units Extensions J	obs Help
🎦 💕 🛃 🔣 🗍 Project	
👔 Import 🗟 Reconnect 🔯 Refresh Projec	t 🖻 Update Project 📲 ACT Start Page
Toolbox 🔽 🗸 🗙	Project Schematic
 Analysis Systems Design Assessment Eigenvalue Buckling Electric Explicit Dynamics Fluid Flow - Blow Molding (Polyflow) Fluid Flow - Extrusion (Polyflow) Fluid Flow (CFX) 	
 Fluid Flow (Fluent) Fluid Flow (Polyflow) Harmonic Acoustics 	

4. In the toolbox > Analysis systems menu drag Static Structural into the project schematic window



5. Double-click Engineering Data to open the Engineering Data Tab



a.

a.

File Edit View Tools Units Exte	nsions J	obs Help											
🗅 💕 🛃 🔍 🕞 Project 🦪 A2:	Engineeri	ng Data 🗙											
Y Filter Engineering Data 🔛 Engineering Da	ta Sources												
roolbox 🗸 🗘 🗴	Outline	of Schematic A2: Engineering Data				- û	x	Table of P	roperties Row 2:	Structu	ral Steel Field Va	iables	
Physical Properties	1	A	в	с	D	E			Α	в	с	D	
Linear Elastic	1	Contents of Engineering Data)	8	Source	Description		1	Variable Name	Unit	Default Data	Lower Limit	
🛃 Isotropic Elasticity	2	Material						2	Temperature	с 💌	22	Program Controlled	Pro
 Orthotropic Elasticity Anisotropic Elasticity 	3	📎 Structural Steel	•		•	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1	8						
Hyperelastic Experimental Data	*	Click here to add a new material											
Hyperelastic													
🗉 Chaboche Test Data	11												
Plasticity	11												
🖽 Creep	11												
⊞ Life	11												
⊞ Strength	1												
E Gasket	1												
a ounce	11												
Viscoelastic Test Data													
Viscoelastic Test Data Viscoelastic													
Viscoelastic Test Data Viscoelastic Viscoelastic Shape Memory Alloy				_					1.1				
Viscoelastic Test Data Viscoelastic Viscoelastic Shape Memory Alloy Geomechanical	Propert	es of Outline Row 3: Structural Steel				→ ‡	×	Chart: No	data				
Viscoelastic Test Data Viscoelastic Viscoelastic Shape Memory Alloy Geomechanical Damage	Propert	ies of Outline Row 3: Structural Steel				в С D	X E	Chart: No	data				
Viscoelastic Test Data Viscoelastic Shape Memory Alloy Geomechanical Damage Cohesive Zone	Propert	tes of Outline Row 3: Structural Steel A Property				B C D Value Unit 🐼	E	Chart: No	data				
Viscoelastic Test Data Viscoelastic Shape Memory Alloy Geomechanical Damage Cohesive Zone Fracture Criteria	Propert	tes of Outline Row 3: Structural Steel A Property Material Field Variables				B C D Value Unit 🐼 Table	E Ç	Chart: No	data				
Uscoelastic Test Data Uscoelastic Shape Memory Alloy Geomechanical Damage Cohesive Zone Fracture Criteria Orack Growth Laws	Propert	ke of Outine Row 3: Structural Steel			78	B C D Value Unit Table 50 kg m^-3 ♥ □	E GR	Chart: No	data				

- 6. You generally will have your own materials data or values from handbooks and will not use the pre-programmed generic materials data.
 - a. Below Structural Steel type a new material name: for this example call it "inco625". Hit Enter.

Outline	of Schematic A2: Engineering Data		
	А	в	С
1	Contents of Engineering Data	9	8
2	Material		
3	2 📎 inco625		
4	📎 Structural Steel	•	
*	Click here to add a new material		

b.

a. on the left double click Isotropic Elastic

🖊 t	utorial	Examp	le - Wo	orkber	nch	
File	View	Tools	Units	Exten	sion	s
1	3	I	- Proj	ect 🔇	A	2:E
T Fil	ter Engin	eering Da	ata 🏢 B	Engineer	ing	Data
Toolbox	x			•	ņ	×
⊞ Fie	eld Vari	ables				
🕀 Ph	ysical F	ropertie	es			
🗆 Lir	near Ela	stic				
'ZI	sotropio	: Elastic	ity			
20)rthotro	pic Elas	ticity			
1 🔁 A	nisotro	pic Elas	ticity			_

- c. Now define the Young's Modulus (E) as 200e3 MPa and Poisson's ratio as 0.3
- d. In the right corner see Youngs Modulus and Poisons ratio text boxes

Table of	Properties Row 3: Isotro	opic Elasticity	
	А	В	с
1	Temperature (C) 🗦	Young's Modulus (Pa) 💌	Poisson's Ratio
*			

- e.
- f. Use the Dropdown menu to change the units of Young's Modulus to 200 MPa and insert the values of Youngs Modulus and Poisson's Ratio

Table of	Properties Row 3: Isotro	opic Elasticity		д Х
	А	В	с	
1	Temperature (C) 🗦	Young's Modulus (MPa) 💌	Poisson's Ratio	Bulk Mo
2		2.0E+5	0.3	3.4417

- g. h. No temperature is needed as this analysis will be only at one temperature
- i. Close the engineering data tab



- neering Data Sources
- i. Note that we only defined materials properties that we needed for linear elastic analysis. Other properties, such as density, thermal expansion coefficient, yield strength, etc can be supplied but will not be used in a linear elastic static analysis

8. Choose 2D analysis

b.

a. In the project schematic right click on Geometry > Properties



c. In the menu to the right use the dropdown to change Analysis Type to 2D

Propertie	es of Schematic A3: Geometry			- Ļ
	А		В	
1	Property		Value	
2	General			
3	Component ID	Geometry		
4	Directory Name	SYS		
5	Notes			
6	Notes			
7	Used Licenses			
8	Last Update Used Licenses			
9	Geometry Source			
10	CAD Plug-In	DesignModeler[9940]		
11	Advanced Geometry Options			
12	Analysis Type	3D		-
13	Compare Parts On Update	3D		
		2D		
11	 Advanced Geom 	etry Options		
12	Analysis Type		2D	

- f. Close the Properties of Schematic: Geometry window
- 9. Right Click on Geometry in the project schematic window



a.
b. Click on New DesignModeler Geometry to open the DesignModeler window (it may take a second or more to open)

•		А		
1	.	Static Structural		
2	٢	Engineering Data		× .
3	Ø	Geometry	2	New SpaceClaim Geometry
4		Model	m	New DesignModeler Geometry
5	٢	Setup		Import Geometry
6	(Solution		Duckasta
7	6	Results	43	Dupiicate Transfer Data From New
		Static Structur		Transfer Data To New
			7	Update
				Update Upstream Components

10. In the DesignModeler Window

c.

a. Click XY Plane

A: Static Struct	ural - DesignModeler			
File Create Cor	ncept Tools Units V	iew Help		
🔄 📑 📑 🖒	🛯 🕽 🕄 Undo 📿 Rec	े Select: 🔭 โ	\ • • • • • • • • • • • • • • • •	R 🖪 🖉 🖉
	1- 1- K- K	• 🗶 🗹		
XYPlane 🔻	None 👫	👻 ಶ 🗍 🧚 Gene	rate 🛛 🖤 Share T	Fopology 🔀 Param
🛛 💽 Extrude 🛛 👘 F	Revolve 🐍 Sweep 🤞	Skin/Loft	Thin/Surface 🛛 🖣	🕨 Blend 💌 🦘 Cham
] 🚸 Point 📳 Co	nversion			
BladeEditor: 🔏 Ir	nport BGD 🛛 🔏 Load B	GD 😔 Load NDF	式 FlowPath	🥖 Blade 🛛 💋 Splitt
] ∦ ᡯ S ≡	(返 🖉 🛛	•		• IØI
Tree Outline			Graphics	
,	Structural lane 'lane 'lane rrts, 0 Bodies			
Sketching Model	ling			

c. Then click on the z arrow in the coordinate system to align with the XY plane



b.

g. ZXPlane 11. Click on the Sketch Tab of the Tree Outline Menu

Model Tolerance

۲

XYPlane

] ❷ 丞 S ≣ (函 ❷	🕭 🗷 S 🗉 (🖄 🖢 Sketching Toolboyes
T 0.11	Sketching roolboxes
Tree Outline	Draw
A: Static Structural	 ∖ Line ♂ Tangent Line ⊘ Line by 2 Tangents △ Polyline ○ Polyligon □ Rectangle
O Parts, 0 Bodies	Modify
· · · · · · · · · · · · · · · · · · ·	Dimensions
	Constraints
	Settings
Sketching Modeling	> Sketching Modeling

12. Draw a rectangle in the Graphics Window

a.

a. A click on rectangle in the Sketching Toolbar menu Sketching Toolboxes ņ Draw . 🔨 Line 6 Tangent Line 6 Line by 2 Tangents A Polyline Polygon Rectangle

Modify

Dimensions

Constraints

b. Draw a rectangle in the Graphics Window (don't worry about the size just now) c.

•



e. Click on dimension tab in the Sketching Toolboxes Polygon

V / J		
Rectangle	Auto-Fillet:	Γ
Modify	1	•
Dimen	sions	
Constr	aints	
6		

f. Then click general g.

	Draw	
	Modify	
	Dimensions	
General		
🛏 Horizontal		
Vertical		
ntength/Distance		
Radius		
⊖Diameter		
	Construints	

- 13. Dimension the rectangle
 - a. Now click on the upper edge of the rectangle and drag the dimension label out to your desired location

	···· d	••••				
In the details v	view chang	e the diı	mension o	of H1 to	o 10 mm	
Details View						
Details of Sketch						
Sketch Sk	ketch					
Sketch Visibility Sh	how Sketch					
Show Constraints? No	0					
Dimensions: 1	-					
H2 10) mm					
- Edges: 4	19					
LITE	112					
Line Ln	120					
Line Ln Repeat this pro	n20 Acess for th	ne left si	ide dimen	sion ch	nanging V	'2 to 10
Line Ln Repeat this pro	ocess for th	ne left si	ide dimen	sion cł	nanging V	'2 to 10
Line Line Line Repeat this pro	ocess for th	ne left si	ide dimen	sion ch	nanging V	'2 to 10
Line Line Line Repeat this pro	ocess for th	ne left si	ide dimen	sion ch	nanging V	'2 to 10
Repeat this pro	ocess for th	ne left si	ide dimen	sion ch	nanging V	'2 to 10
Repeat this pro	ocess for th	ne left si	de dimen	sion ch	nanging V	'2 to 10
Line Lr Repeat this pro	¹²⁰ ocess for th	ne left si	ide dimen	sion ch	nanging V	′2 to 10
Repeat this pro	¹²⁰ ocess for th	ne left si	ide dimen	sion ch	nanging V	′2 to 10
Line Lin Repeat this pro	¹²⁰ ocess for th	ne left si	de dimen	sion cł	nanging V	'2 to 10
Line Lr Repeat this pro	120 ocess for th	ne left si	de dimen	sion cł	nanging V	′2 to 10
Repeat this pro	120 ocess for th	ne left si	de dimen	sion ch	nanging V	′2 to 10
Line Lin Repeat this pro	120 ocess for th	ne left si	de dimen	sion cł	nanging V	′2 to 10
Line Lr Repeat this pro	120 ocess for th	ne left si	de dimen	sion cł	nanging V	'2 to 10
Line Lr Repeat this pro	120 ocess for th	ne left si	de dimen	sion cł	nanging V	′2 to 10
Line Lin Repeat this pro	Direction of the second	ne left si	de dimen	sion cł	nanging V	′2 to 10
Line Lin Repeat this pro	ocess for th	ne left si	de dimen	sion cł	nanging V	′2 to 10
Line Lin Repeat this pro	Docess for th	ne left si	de dimen	sion cł	nanging V	′2 to 10
Line Lin Repeat this pro	0.000 0.000 2.500	5.000 7.500	de dimen	sion ch	nanging V	" ² to 10
Line Lin Repeat this pro	0.000 0.000 2.000	5.000 7.500	de dimen	sion cł	nanging V	′2 to 10
Line Lin Repeat this pro	0.000 0.000 2.500 No	5.000 7.500	de dimen	sion cł	nanging V	′2 to 10
Line Lr Repeat this pro	0.000 0.000 2.000 No	5.000 7.500	de dimen	sion cł	nanging V	′2 to 10
Line Lr Repeat this pro	0.000 0.000 2.500 No 10 mm 10 mm	5.000 7:500	de dimen	sion cł	nanging V	′2 to 10
Line Lr Repeat this pro	0.000 2.500 No 10 mm	<u>5.000</u> 	de dimen	sion cł	nanging V	′2 to 10

h. Use the Zoom to Fit tool in the upper toolbar to center the image

14. Add the hole

a. Click on the circle icon in the Sketching toolbox Draw tab (you may have to scroll down using the down arrow to the right)

	Draw
	Diaw
	∧ Polyline
	- Polygon
	Rectangle
	Rectangle by 3 Points
	Oval
	🕓 Circle
	▼. Modify
ľ	Dimensions

c. Draw a circle inside of the rectangle



- 15. Dimension the hole
 - a. Click on Diameter in the Dimensions Tab

	Sketching Toolboxes	
(Draw
		Modify
		Dimensions
	General	
	Horizontal	
	I Vertical	
	Length/Distance	
	Radius	
	⊖Diameter	
b.	·	Constraints

c. Click on the circle and drag the dimension to your desired location



e. On the left change the diameter D3 to 2 mm

Show Constraints? No

=	Dimensions: 3	
	D3	2 mm
	H1	10 mm
	V2	10 mm

- f. Edges: 5
- g. Now we will position the circle in space. Click on Horizontal in the Dimensions Tab

	Draw	
	Modify	
	Dimensions	
General		
🛏 Horizontal		
Vertical		
Length/Distance		
Radius		
Diameter		
	Constraints	•
	6 m	

i. Click on the left edge of the rectangle and then click on the left of the circle and then drag the dimension label to your desired location

•	H4	D3

j.
k. Now change H4 in the menu on the left to 4 mm

Dimensions: 4	
D3	2 mm
H1	10 mm
H4	4 mm
□ V2	10 mm
F.d F	-

١.

m. Click on Vertical in the Dimensions Tab

V ·····	
🛏 Horizontal	
[Vertical	
Length/Distance	
▲ Radius	

n. A Radius
o. Click on the top edge of the rectangle and then click on the top of the circle and then drag the dimension label to your desired location



- p.
- q. Now change V6 in the menu on the left to 4 mm (It might be labeled V5, but that's ok)

=	Dimensions: 5	
	D3	2 mm
	H1	10 mm
	H4	4 mm
	V2	10 mm
	🗌 V6	4 mm

r. s. Now the whole is centered



16. Generate the sketch

a. On the upper toolbar click on Concept > Surface from Sketches



- b. ± yvplane
- c. In the details view click on Base Object > Apply

	Details of SurfaceSk		
	Surface From Sketches	SurfaceSk	
	Base Objects	Apply	Cancel
	Operation	Add Material	
	Orient With Plane Normal?	Yes	
	Thickness (>=0)	0 mm	

- e. Now we want to select the entire sketch
- f. Select the Box Select from the Select menu



h. You can now draw a box around the sketch



j. Base Object should say "1 Sketch", if it does not try Apply and select again

Details View				
Ξ	Details of SurfaceSk			
	Surface From Sketches	SurfaceSk		
	Base Objects	1 Sketch		
	Operation	Add Material		
	Orient With Plane Normal?	Yes		
	Thickness (>=0)	0 mm		

k.

d.

I. Now click generate (in the upper 3rd of the screen)



- m.
- n. The sketch should change color



- o. 2.500 7.500 p. Save our project and close Design Modeler
- 17. Open the Modeler
 - a. Double click on Model in the Project Schematic Window

▼	А	
1	😇 Static Structural	
2	🥏 Engineering Data	× .
3	🕅 Geometry	× .
4	🎯 Model	2
5	🍓 Setup	? 🖌
6	Solution	? 🖌
7	🥪 Results	? 🖌
	Static Structural	

- b.
- c. It may take a couple seconds to open, but should load your sketch geometry

😭 A : Stat	c Structural - Mechanical (ANSYS Academic Research Mechanical an	d CFD]							- 0	
File Edit	View Units Tools Help	ew Analysis 👻 ?/S	how Errors 1 🖬 🖬	i 🕅 🕅 👰 •	Worksheet	N				
₽ /~	9 5 - 10 10 10 10 10 10 10 4 - 12 4 (① ① ① ① ① ② ② 	0 0 10 10			×				
P Show	/ertices A Close Vertices 1.4e-005 (Auto Scale) - WWirefra	me Di Show Me	h 🍌 🕌 Randon	n 🐼 Preference	s L L L L	I.,				
↔ Size ▼	. Location • Convert • Miscellaneous • O Tolerance	Clipboard	[Empty]							
≩t Ω⊷ R	eset Explode Factor: Assembly Center	- Ed	e Coloring 👻 🔏 🗸	1. 1. 1	(- <i>/.</i> .	Thicken				
Model 6	Construction Geometry	Remote Point	Connections	@ Fracture	Condensed Geometr	Mesh Edit	Mesh Numbering	Solution Combination	Fatique Combin	natio
Dutline										
Filter: Na	ne 💌	-							ANS	YS
S 21 -	> H 🗟 👭									.9.0
Projec	t	1							Acader	
🖻 – 🎯 M	odel (A4)									
÷ 7	Geometry									
	Mesh									
6-1	Static Structural (A5)									
	Analysis Settings									
1	- 200 Solution (A6)									
	-									
Details of "N	fodel (64)"									
- Filter Opt	ions	1								
Control	Enabled									
Elighting										
Ambient	0.1									
Diffuse	0.6								Y	
Specular	1	-1							+	
Color		-								•
										-
					0	0.003	0.00)6 (m)		
					0.0015		0.0045	and the second second		

- d. 18. Model Geometry
 - a. Expand the Geometry option in the project outline



i. Below Assignment change Nonlinear Effects and Thermal Strain Effects both to NO

	Demanor	
-	Material	
	Assignment	inco625
	Nonlinear Effects	No
	Thermal Strain Effects	No
+	Bounding Box	

- + Properties
- k. Under Details of Geometry>definition keep the 2D behavior as plane stress (as the plate is very thin)
- 19. Finite Element Mesh

b.

a. Click on Mesh in the project outline



c. In the Details of Mesh menu you could change the Physics Preference to other types of systems but DO NOT, leave it as Mechanical

	Display				
	Display Style	Body Color			
-	Defaults				
	Physics Preference	Mechanical 🗸			
	Element Order	Mechanical			
÷	Sizing	Nonlinear Mechanical			
t	Quality	CFD			
+	Inflation	Explicit			
+	Advanced	Hydrodynamics			

d. + Statistics

f. g.

h.

e. In the Details of Mesh menu change the element order (I.e., the order of the element shape function) to linear

-	Physics Preference	Mechanical	
ľ	Element Order	Program Controlled	
● Sizing Program Controlled ● Quality Quadratic		Program Controlled	
		Linear	
		Quadratic	
l	Inflation		

Coordinate Systems	
⊡	Method
🖃 🦩 Update	🔍 Sizing
📁 Generate Mesh	A Refinement

i. In the Details of Automatic Method Menu click geometry, select the plate to the right, then click apply

E Scope		
Scoping N	/lethod	Geometry Selection
Geometry		1 Body
Geometry		TBODY

- Definition
- k. The plate should change color and say Automatic Method in the upper left corner

	A u 11	itomatic Method /27/2018 4:11 PM	
	Automatic Method		
I.			
m. For method select Qu			Quadrilateral dominant
		Geometry	1 Body
	E	 Definition 	
		Suppressed	No
		Method	Quadrilateral Dominant

- n. Element Order Use Global Setting
- o. For Free Face Mesh Type select All Quad

Method	Quadrilateral Dominant	
Element Order	Use Global Setting	
Free Face Mesh Type	All Quad	

- p.
- q. We will now set an approximate element size
- r. Right click mesh and select sizing

Surface Body	
Insert	Method
□ ? = 51 🔰 Update	🔍 Sizing
✓4	A Refinement

•

s.

u.

t. Under body sizing select the plate to the right then click Geometry Apply

_		<u> </u>
-	Scope	
	Scoping Method	Geometry Selection
	Geometry	1 Body

v. Change element size to 0.0001

		Geometry	1 Body		
	Ξ	Definition			
		Suppressed	No		
		Туре	Element Size		
		Element Size	0.00001		

- w. Advanced
- x. Right click on mesh and select Generate Mesh (this may take a second or two)



- у. 7
- z. It is ok if a few triangles are used with the quad elements, you should get a mesh that looks something like this (In general 2D meshes are cheap computationally and should be very fine, 3D meshes are much more expensive computationally and might need to be coarser)



aa.

bb. Details of Mesh > Statistics will show how many elements and nodes20. Apply Boundary Conditions (I.e. loads and constraints)

a. Apply the load. Right click on Static Structural an select Insert > Pressure

Name 🔻	🕸 Standard Earth Gravity
🖉 🗠 🕀 🗟 😫	🔍 Rotational Velocity
roject	🔍, Rotational Acceleration
Geometry	🔍 Pressure
🔍 🗤 Surface Body	🥰 Hydrostatic Pressure
E Coordinate Systems	🔍 Force
∃	🚱 Remote Force
Automatic Method	🔩 Bearing Load
Static Struct	Bolt Pretension
Analysis S Insert	🖳 🧐 Moment
Solution Solution Solution	📲 Thermal Condition

b.

c. Select the upper edge of the plate



Static St	AC)	👬 Bolt Pretension
Ana Insert		🕐 Moment
Solve ≯ Solve	(F5)	🔊 Thermal Condition
🖉 📴 🖗 Dupli	cate	G South Load
Clear	Generated Data	Image: Construction of the second support Image: Consecond support Image: Consecond
alb Renar	ne (F2)	🔍, Remote Displacement

- j. k. Select the lower edge of the plate
- Click Geometry > Apply (if it does not work, try it again) Ι.

o	0.004	0.008 (m)
7 – X	0.002	0.006

m.

n. Change the y-component displacement to 0 (leave the x-component free)

	Туре	Displacement
	Define By	Components
	Coordinate System	Global Coordinate System
	X Component	Free
	🔲 Y Component	0. m (ramped)
	Suppressed	No

- о. Suppressed
- p. Apply x-constraint on the right edge. Right click on Static Structural an select Insert > Displacement
- q. Select the right edge of the plate
- r. Click Geometry > Apply (if it does not work, try it again)



- s.
- Change the x-component displacement to 0 (leave the x-component free) t.

	Scoping Method	Geometry Selection	
	Geometry	1 Edge	
Ξ	Definition		
	Туре	Displacement	
	Define By	Components	
	Coordinate System	Global Coordinate System	
	🗌 X Component	0. m (ramped)	
	Y Component	Free	
	Suppressed	No	

- u.
- v. Intersection of bottom and left edge. The intersection of the bottom and left edge must have both an x-constraint and y-constraint
- w. Right click on Static Structural an select Insert > Fixed Support (I.e. no displacement in any direction)

	Body Sizing	Searing Load	
		Bolt Pretension	
	Analysis Insert	• 🔍 Moment	
	R Pressure	🔍 Thermal Condition	
	Displacer	🔷 Joint Load	
	Duplicate	R Find Connect	
	Sol Z Clear Generated Data	Pixed Support	
х.	alle Pename (E2)	Displacement	
у.	Change the selection method to Vertex		
	Units Tools Help		
	s 🔁 💽 💽 🐚		
	后 Close Vertices 1		
	enting an DC convert of		
	ation V 💽 Convert		
Z.	Coloct the lower right corner		
aa.	Select the lower right corner		
	0.004 0.008 (m)		
bb.	0.002 0.006		

- cc. Click Geometry > Apply (if it does not work, try it again
- 21. Solve
 - a. Now tell ANSYS what results you want. Lets select y-displacement, y-normal stress, and Equivalent (I.e. von Mises) stress
 - b. Right click on Solution and select Directional Displacement

C. of "Solution (A6)" Clear Generated Data C. of "Solution (A6)" Clear Generated Data C. of "Solution (A6)" Clear Generated Data C. of the solution (Clear Generated Data Clear Generated Data Clear Generated Data C. of the solution (Clear Generated Data Clear Generated Data Clear Generated Data C. of the solution (Clear Generated Data Clear					
d. Change Orientation to Y-Axis Definition Type Directional Deformation Orientation Y Axis By Time					
Definition Type Directional Deformation Orientation Y Axis By Time					
Definition Type Directional Deformation Orientation Y Axis By Time					
Type Directional Deformation Orientation Y Axis By Time					
Orientation Y Axis By Time					
By Time					
Display Time Last					
f. Right click on Solution and select Stress > Normal					
Solution (AE)					
Insert Deformation					
"Solution (A6					
ve Mesh Refir 🖉 Clear Generated Data Energy 🔸 🍕 Maximum Principal					
finement Loo alb Rename (F2) Linearized Stress 🕨 💁 Middle Principal					
ment Depth Group All Similar Children Stress Tool Stress Tool					
Compensation Solver Files Directory Fatigue					
Elapsed Time 🗊 Worksheet: Result Summary					
g. Memory Used 276. MB Charren to V-Avis					
i Right click on Solution and select Stress > Equivalent (yon Mises)					
i. Right Click on Solution and select Selve					
	Fight Click on Solution and select Solve				
Displacement 2					
Fixed Support					
Solution (AC)					
So Insert					
of "Solution (A6)" 🔰 Solve (F5)					
tive Mesh Refineme					
Clear Generated Data					
k. (ED) Denome (ED)					
22. Results					
a. Click on each result to displace the contour					
, Fixed Support					
Solution (A6)					
Solution Information					
b.					



 c.
 d. For a cleaner plot without elements click Edges and changes to No Wireframe Assembly Center



f. Use the Pan button to move the plate near the legend



g.
h. The analytical solution is a stress concentration factor of 3.0, so the peak stress should be 300 MPa, the predicted stress is 311 MPa.