ANALYSIS OF A PLATE WITH STRESS CONCENTRATIONS

Instructor: Professor James Sherwood

Author: Dimitri Soteropoulos

Programs Utilized: Abaqus/CAE 6.11-2

Problem Description:

This tutorial illustrates the effects of various stress concentrations on a plate. Three different stress concentrations are incorporated into the geometry of and aluminum plate: a hole, fillets, and a crack. The mesh of the plate is designed and refined to fit the conditions of the geometry.

Creating the Model Geometry

- Go to the Start Menu and open Abaqus/CAE
- You may be prompted with an **Abaqus/CAE 6.11-2 Start Session** dialog box (Figure 1). Close this box by clicking the **X** in the top right hand corner.



Figure 1. Abaqus/CAE 6.11-2 Start Session Dialog Box.

Once the Start Session box is exited, the Abaqus/CAE Viewport should look similar to Figure 2.
 (Please note the *model tree* is the series of functions listed on the left hand side of the viewport, while the *module* is the list of icons to the right of the model tree)

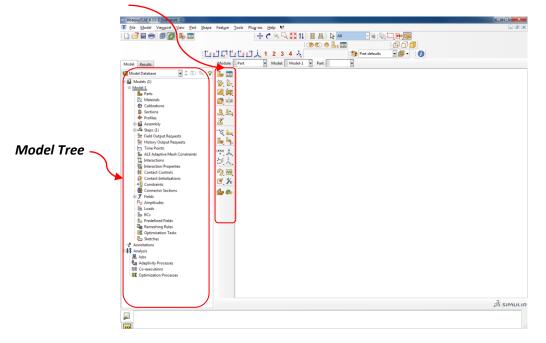


Figure 2. Abaqus/CAE Viewport

- To create the model geometry of the aluminum plate, a sketch of the face of the part must be generated.
- Using the left mouse button, double click **Parts** in the model tree and the **Create Part** (Figure 3a) dialog box appears. Enter a new name for the part (PLATE), and under the Base Feature tab choose **Shell** for shape and **Planar** for type. Change the approximate size option to **20**. The **Create Part** dialog box should look identical to Figure 3b.
- Click **Continue**... and the graphics window will change to a set of gridlines.

⇔ Create Part		×	Ŋ
Name: Part-1 Modeling Sp 3D © 2D	ace	Axisymmetric	
Type	gid	Options None available	
Base Feature Shape Solid Shell Wire Point	Type Extrusi Revolu Sweep		
Approximate si	_	Cancel	

Figure 3a. Create Part Dialog Box

Create Part		23
Name: PLATE		
- Modeling Spa	ace	
● 3D 2D	Planar	 Axisymmetric
Туре		Options
 Deformab Discrete ri Analytical Eulerian 	gid	None available
Base Feature Shape	Туре	
Solid	Plana	r
Shell	Extrus	
WirePoint	Revol Swee	
Approximate si	ze: 20	
Continue		Cancel

Figure 3b. Create Part Dialog Box (PLATE)

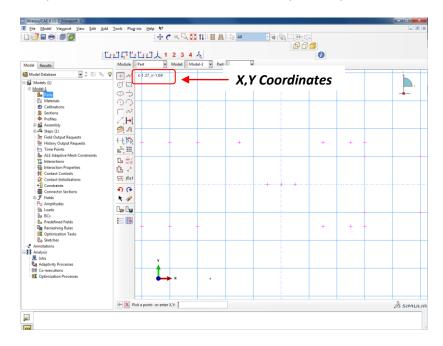
- For the first step in generating the model geometry an outline of the plate must be created.
 Click the Create Isolated Point icon + in the module. (Remember, the module is the series of icons to the right of the model tree)
- At the bottom of the viewport **the Pick a point or enter X,Y:** option will appear. Points will be entered using X,Y coordinates. Enter the X,Y coordinates of the points listed in Table 1. After each entry hit enter on the keyboard and the point will appear in the viewport. (Enter the points in the x,y format)

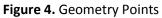
Point	X Coordinate	Y Coordinate
1	-2.5	0.75
2	-2.0	0.75
3	-1.5	0.75
4	-0.75	0.75
5	0.75	0.75
6	1.25	0.75
7	1.5	0.75
8	1.5	0.5
9	2.5	0.5
10	2.5	-0.5
11	1.5	-0.5
12	1.5	-0.75
13	1.25	-0.75
14	0.75	-0.75
15	-0.75	-0.75
16	-1.5	-0.75
17	-2.0	-0.75
18	-2.5	-0.75

Table 1. Points for Geometry

19	0	0
20	-0.25	0
21	0.25	0

• If all of the points have been entered correctly, the viewport should look similar to Figure 4. To auto scale the points to fit the screen hit **F6** on the computer keyboard.





- Click the Create Lines: Connected icon in the module, click and create lines between points
 1 & 2, 2 & 3, 3 & 4, 4 & 5, 5 & 6. After the fifth line has been created click the center scroll wheel on the mouse to exit the creation of this segment of lines.
- Create lines between points **13** & **14**, **14** & **15**, **15** & **16**, **16** & **17**, **17** & **18**. After the fifth line has been created click the center scroll wheel on the mouse to exit the creation of this segment of lines.
- Next, create lines between points **8** & **9**, **9** & **10**, **10** & **11**. After the third line has been created click the center scroll wheel on the mouse to exit the creation of this segment of lines.
- Finally, create a line between points **1** & **18**. Press **Esc** on the computer keyboard to exit the **Create Lines: Connected** tool. At this point of the sketch the geometry should look similar to that in Figure 5.

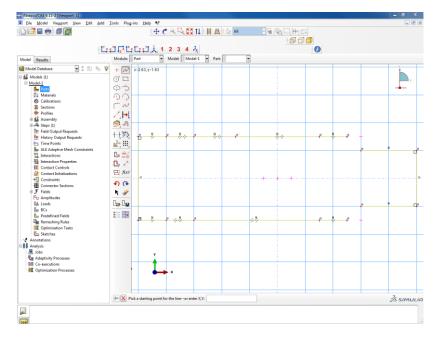


Figure 5. Geometry Lines

• The final step in drawing the plate geometry is to create a hole and two fillets. Click the **Create**

Circle: Center and Perimeter icon in the module. Using the cursor click point **19**, this will denote the center point for the circle. Next click point **20**, this will select a perimeter point for the circle. Press **Esc** on the computer keyboard to exit the **Create Circle: Center and Perimeter** tool.

- Finally, click the Create Arc: Center and Two Endpoints (i) icon in the module. For a center point the first filet click point 7. Next click point 8 to denote a start point for the arc. Finally click point 6 for an endpoint for the arc. Repeat the same steps to create the second filet for points 12 (center), 13 (start), 11 (end). Press Esc on the computer keyboard to exit the Create Arc: Center and Two Endpoints tool.
- The completed geometry should look similar to that in Figure 6.

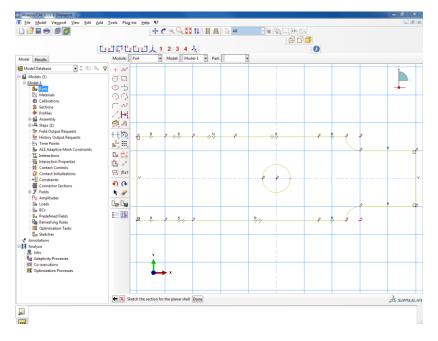


Figure 6. Geometry Complete

• The geometry of the plate is now complete. Click **Done** at the bottom of the viewport to exit the sketching viewport. Upon exiting the sketching viewport, the part should turn a solid grey color (Figure 7).

Abaqus/CAE 6.11-2 [Viewport: 1] File Model Viewport View Part Sha	Fortune Taulo Nucleo Hide M	
_ (m 🛋 : :::: (iii) : :iii iiii		
	000001,0000000000000000000000000000000	
i L		• 10
Addel Results	Module: Part Model: Model-1 Part PLATE	
Model Database	1. 🖬	Y
Models (1)		
Model-1	1 () () () () () () () () () (×
🖲 🦾 Parts (1)	🔟 💷	
 Materials Calibrations 	🗿 🚧	
- Sections		
+ Profiles	3	
🗄 🎎 Assembly		
General Steps (1) Field Output Requests	-ta 📙	
History Output Requests		
Time Points		
LE Adaptive Mesh Constraints	ovin, 木,	
Interactions	超見	
Contact Controls		
Contact Initializations		
Constraints		
 		
Amplitudes	\sim	
Loads		
BCs		
Predefined Fields Remeshing Rules		
Remesning Rules Optimization Tasks		
L Sketches		
Annotations		
🛊 Analysis 🔜 Jobs		
Adaptivity Processes	Y	
Co-executions		
Optimization Processes	k→ ×	
		2
		25 simul

Figure 7. Plate with Hole (Part Module)

Defining Material Properties

- To define material properties for this model, double click on Materials in the model tree and the Edit Material dialog box will appear (Figure 8a). Enter a Name for the material (ALUMINUM), and click the Mechanical tab, highlight Elasticity and click Elastic. Enter values of Young's Modulus = 10E06 Psi, and Poisson's Ratio = 0.3. After the material properties have been entered, the Edit Material dialog box should look identical to Figure 8b.
- Click OK.

Edit Mat				X
Description				4
Material B	ehaviors			
wateriaru	enaviors			
General	Mechanical	Thermal	Other	y
general	Meenanica	Tuesting	<u>S</u> arci	

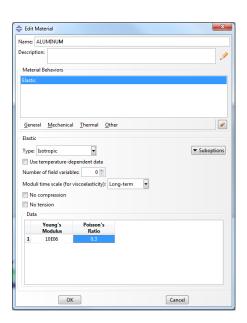


Figure 8a. Edit Material Dialog Box

Figure 8b. Edit Material Dialog Box (ALUMINUM)

- Please note there is no dropdown menu or feature in Abaqus that sets specific **units**. All of the dimensions have been input in inches; therefore the respective Young's Modulus units should be entered in Psi (Pounds per square inch). The units chosen for the definition of the material properties should be consistent and dictate what units should be used for the dimensions of the structure.
- At this point in preprocessing, the model should be saved. Click **File** then click **Save**. Name the file **Plate With Hole Tutorial**. The file will save as a Model Database (*.cae*) file. It may be of interest to save the file after each section of this tutorial.

Creating Sections

- To create a shell section in Abaqus, double click Sections in the model tree and the Create Section dialog box will appear (Figure 9a). Enter a Name for the section (SHELL), and choose Shell under the Category Tab, and Homogeneous under the Type tab. Your Create Section dialog box should look identical to that in Figure 9b.
- Click Continue...

+ Create Section			
Name: Secti	on-1		
Category	Туре		
Solid	Homogeneous		
Shell	Generalized plane strain		
🔘 Beam	Eulerian		
Fluid	Composite		
Other			
Continue Cancel			

Create Section				
Name: SHEL	L			
Category	Туре			
Solid	Homogeneous			
Shell	Composite			
🔘 Beam	Membrane			
Fluid	Surface			
Other	General Shell Stiffness			
Continue Cancel				

Figure 9a. Create Section Dialog Box

Figure 9b. Create Section Dialog Box (SHELL)

- The Edit Section dialog box will then appear where a value for the respective Shell thickness can be prescribed for this section. Because only one material has been created, the Material is defaulted to ALUMINUM. If multiple materials had been created, the dropdown menu could be used to prescribe a different material to this section.
- Under the basic tab enter 0.25 for the Shell thickness. Change the Thickness integration rule: to Gauss. When this is done the number of Thickness integration points will default to 3. The Edit Section dialog box should look identical to that in Figure 10.
- Click OK.

🜩 Edit Section
Name: SHELL
Type: Shell / Continuum Shell, Homogeneous
Section integration: During analysis Before analysis
Basic Advanced
- Thickness
Shell thickness:
Element distribution:
Nodal distribution:
Material: ALUMINUM
Thickness integration rule: Simpson Gauss
Thickness integration points: 3
Options: 🔶
OK

Figure 10. Edit Section Dialog Box (SHELL)

Assigning Sections

Now that the shell section has been created, it can be assigned to the geometry. In the model tree, click the + to the left of the Parts icon, this will further expand the model tree's options. Next, click the + to the left of the part called **PLATE**, further expanding the model tree (Figure 11).

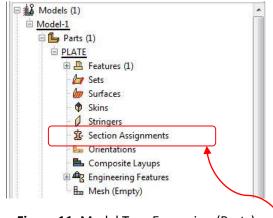


Figure 11. Model Tree Expansion (Parts)

- After the model tree has been expanded, double click **Section Assignments**. Using the cursor, draw a box around the whole part. If the section has been chosen correctly the part will change color from grey to red (Figure 12).
- Click **Done**.

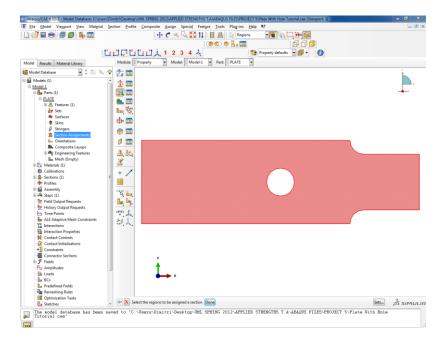


Figure 12. Selected Geometry

 The Edit Section Assignment dialog box will appear. Ensure that SHELL is selected under the Section option. Under the Shell offset option make sure the drop down definition is set to Middle surface. The Edit Section Assignment dialog box should look identical to that in Figure 13.

Region		
Region:	(Picked)	
Section		
Section:	SHELL 🝷 🅸	
	ist contains only sections pplicable to the selected regions.	
Туре:	Shell, Homogeneous	
Material:	ALUMINUM	
Thickne	55	
Assignm	ent: 💿 From section 💿 From g	eometry
Shell Of	fset	
Definitio	n: Middle surface y 🚨	
	OK Cancel	

Figure 13. Edit Section Assignment (SHELL)

• Click **OK**. The geometry should now turn to a green color.

Creating an Instance

• Before the part will be meshed, it can be brought into the assembly. To do this task, click the + to the left of **Assembly** in the model tree. The model tree will expand and should look identical to Figure 14.

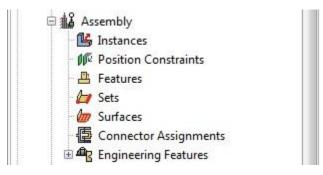


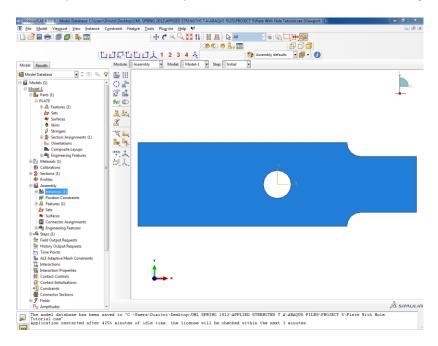
Figure 14. Model Tree Expansion (Assembly)

• Double click on the **Instances** icon in the expanded model tree. This feature will allow multiple parts to be brought into the assembly. The **Create Instance** dialog box will appear (Figure 15). Under the **Instance Type** Option make sure to click **Independent (mesh on instance)**.

Create Instance
Parts
PLATE
Instance Type
Dependent (mesh on part)
Independent (mesh on instance)
Note: To change a Dependent instance's mesh, you must edit its part's mesh.
Auto-offset from other instances
OK Apply Cancel

Figure 15. Create Instance Dialog Box

• Click **OK**. If this step was done correctly the model should turn a blue color (Figure 16).





Creating a Mesh

- To create a mesh for the model geometry, expand **Instances (1)** in the model tree and then expand the **PLATE-1** instance in the model tree and double click **Mesh (Empty)**. Be sure to mesh in the ASSEMBLY part of the model tree and not the PARTS section. If this selection is done correctly, then the geometry should change color to pink.
- The first step in creating the mesh is to partition the geometry. Click the **Partition Face: Sketch** icon in the module. You will be prompted to click an edge or axis, click one of the edges of the geometry and the sketching plane will appear.

• Using the **Create Lines: Connected** icon in the module create lines such that the geometry looks identical to that in Figure 17.

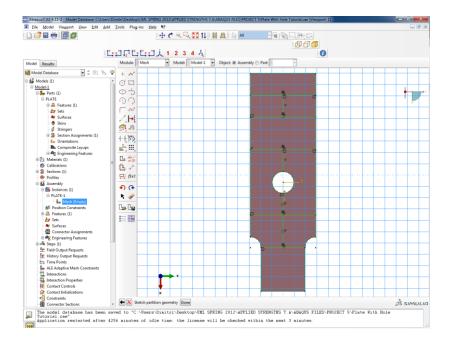


Figure 17. Partitioned Geometry

- Click **Done**. The sketching tool will automatically be exited.
- Click Done.
- The next step in creating a mesh is to seed the part. Since a refined mesh is desired around

holes and cracks each edge will be meshed accordingly. Click and the **Seed Edge** icon in the mesh module.

- The edges listed in Figure 18 are to be seeded with a bias. Table 2 lists the edge number (1-10), the bias ratio, and how many elements along the length of that edge. Note, when clicking the edge to be seeded, click the side of the edge where the bias should be denser. A red arrow will point where the mesh will be more refined.
- When clicking an edge listed in Table 2, the Local Seeds dialog box will appear. Under the Basic tab, change the Method to By number and the Bias to Single. To change direction of the biased seeding click Flip bias. When the bias information for an edge has been entered click OK, and move on to the next edge listed in Table 2.

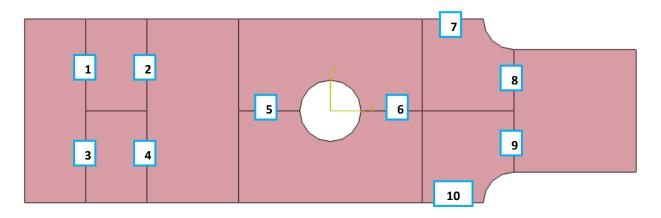
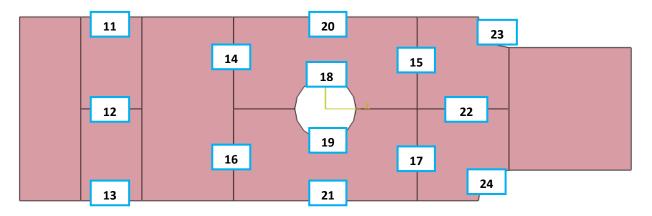


Figure 18. Selected Biased Edges

Edge Number	Bias Ratio	# of Elements
1	3	6
2	3	6
3	3	6
4	3	6
5	3	4
6	3	4
7	3	4
8	3	4
9	3	4
10	3	4

Table 2. Biased Edges	Table 2.	Biased	Edges
-----------------------	----------	--------	-------

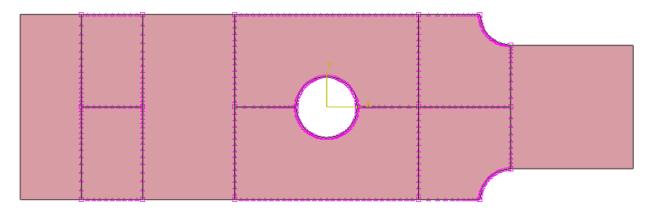
Now that all of the biased edges have been seeded, the edges shown in Figure 19 will be seeded with equal spacing of the nodes. Under the Basic tab in the Local Seeds dialog box, change the Bias option to None. Only the number of elements is to be entered for this edge seeding. Seed the edges listed in Figure 19 with their respective values shown in Table 3.

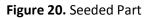




Edge Number	Bias Ratio	# of Elements
11	0	4
12	0	4
13	0	4
14	0	6
15	0	6
16	0	6
17	0	6
18	0	24
19	0	24
20	0	12
21	0	12
22	0	6
23	0	12
24	0	12

• If the part has been seeded correctly, the viewport should look similar to that in Figure 20.





• The next step in creating the mesh is to assign mesh controls. Click the **Assign Mesh Controls** icon in the model tree. Using the cursor draw a box around the whole geometry. If this is

done correctly the color of the part will turn from pink to red.

- Click Done.
- The Mesh Controls dialog box will immediately appear. Under the Element Shape option click Quad. Under the Technique option click Structured. You may be prompted by an Abaqus dialog box, click Yes.
- Click OK.
- All of the regions of the model should turn a green color except for the two regions that define the central hole (Figure 21).

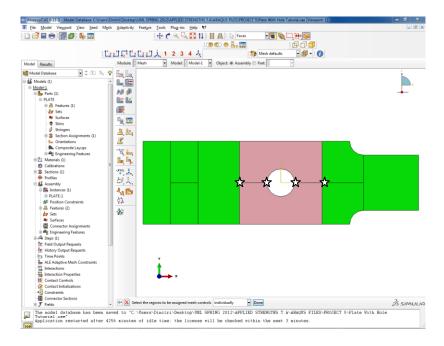
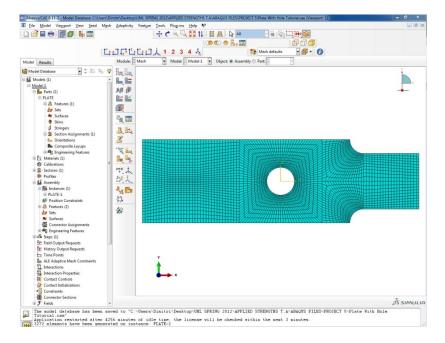


Figure 21. Mesh Regions

- While the **Assign Mesh Controls** option is still selected hold shift on the computer keyboard and click the two sections that define the central hole. If this is done correctly they will turn from a pink to a red color.
- Click Done.
- The Mesh Controls dialog box will immediately appear. Under the Element Shape option click Quad. Under the Technique option click Structured. You will be prompted by an Abaqus dialog box, click Yes. Click the Redefine Region Corners... option in the Mesh Controls dialog box. At the bottom of the viewport, click Select New.
- While holding shift of the keyboard click the four nodes that join the two regions along the x-axis (Denoted by the stars in Figure 21).
- You will be prompted to do this procedure twice, for both the top and bottom partition of the model, follow the same procedure and select the same four nodes for both partitions.
- Click **Done**. Click **OK**.
- The part is ready to be meshed. Click the **Mesh Part Instance** Icon In the module.
- Click Yes. If the mesh has been generated properly, the part should look similar to Figure 22.





Creating a Crack

• Now that the geometry has been meshed, a crack will be created on the part. In the model tree, expand the **Engineering Features** by clicking the **+** sign. If this has been done correctly, the model tree should look similar to that in Figure 23.

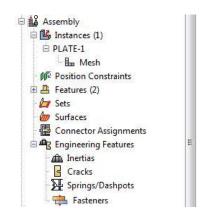


Figure 23. Model Tree Expansion (Crack)

 Double click Engineering Features in the model tree and top toolbar of the screen will change. At the top toolbar click Special and in the dropdown menu hover the cursor over <u>Crack</u> and click <u>Assign Seam...</u> (Figure 24).

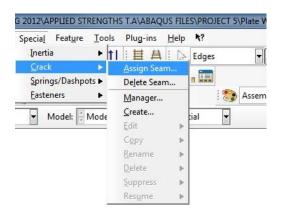


Figure 24. Assign Seam Dropdown Menu

- Click the horizontal line created in the partition to assign the line to be modeled as a seam. (This line is the central horizontal line located at the left side of the part).
- Click Done.
- Click **Done**.

Creating a Step

- A Step is where the user defines the type of loading, e.g. Static or Dynamic, and defines the boundary conditions, e.g. support constraints and forces.
- In the model tree, double click the Steps (1) icon. The Create Step dialog box will appear (Figure 25a). Create a Name for the step called LOADING STEP. Under Procedure type choose General > Static, General. The Create Step dialog box should look identical to Figure 25b.

Create	e Step		23
Name: S	itep-1		
Insert nev	w step after		
Initial			
Procedur	e type: Ge	neral	-
Dynamic	, Explicit		*
Dynamic	, Temp-dis	p, Explicit	
Geostati	c		_
	nsfer		
Heat trai			
Heat trai Mass dif			=
			E
Mass dif	fusion		Ш
Mass dif Soils	fusion eneral		H

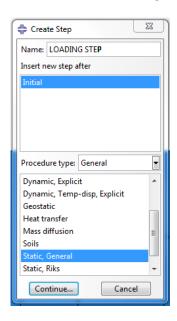


Figure 25a. Create Step Dialog Box

Figure 25b. Create Step Dialog Box (LOADING STEP)

• Click Continue..., and the Edit Step dialog box will immediately appear (Figure 26).

🜩 Edit Step	23
Name: LOADING STEP	
Type: Static, General	
Basic Incrementation Other	
Description:	
Time period: 1	
Nlgeom: © On of large displacements and affects subsequent steps.)	
Automatic stabilization: None	
Include adiabatic heating effects	
OK	

Figure 26. Edit Step Dialog Box

• Click **OK** to accept the default values for the various options.

Apply Constraint Boundary Conditions

- Boundary conditions will be defined which will simulate a fixed (also known as "clamped") beam at one end with a tip load.
- Double click BCs in the model tree and the Create Boundary Condition dialog box will appear (Figure 27a). Create a Name for the boundary condition called FIXED, and under the Step drop down menu make sure to choose Initial. Under the Category option choose Mechanical, and choose Symmetry/Antisymmetry/Encastre under the Types for Selected Step option. The Create Boundary Condition dialog box should look identical to that in Figure 27b.

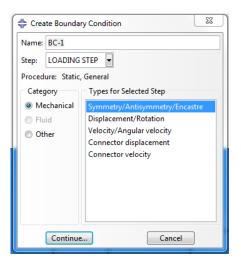


Figure 27a. Create Boundary Condition

💠 Crea	te Bounda	ry Condition			
Name:	FIXED				
Step:	Initial				
Proced	ure:				
Categ	lory	Types for Selected Step			
Me	chanical id	Symmetry/Antisymmetry/Encastre Displacement/Rotation			
© Oti	her	Velocity/Angular velocity Acceleration/Angular acceleration Connector displacement Connector velocity Connector acceleration			
Continue Cancel					

Figure 27b. Create Boundary Condition (FIXED)

- Click Continue...
- Click on the left vertical edge of the part. If the edge has been selected correctly it will turn a red color.
- Click Done.
- The Edit Boundary Condition dialog box will immediately appear. Click ENCASTRE (U1=U2=U3=UR1=UR2=UR3=0). The Edit Boundary Condition dialog box should look identical to that in Figure 28.

🕂 Edit Boundary Condition 🛛 🛛 🔀
Name: FIXED
Type: Symmetry/Antisymmetry/Encastre
Step: Initial
Region: (Picked)
CSYS: (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)
PINNED (U1 = U2 = U3 = 0)
ENCASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0)
OK Cancel

Figure 28. Edit Boundary Condition Dialog Box

• Click OK.

Applying an Axial Load to the Structure

 An axial load of 500 lbs will be applied on the right side of the structure. Double click Loads in the model tree and the create load dialog box will appear (Figure 29a). Create a Name for the load called AXIAL. Ensure that the Step option is set to LOADING STEP and that the Category is set to Mechanical. For the Types for Selected step option choose Shell edge load. The Create Load dialog box should look similar to Figure 29b.

Create Load	🚔 Create Load
Name: Load-1	Name: AXIAL
Step: Initial	Step: LOADING STEP
Procedure:	Procedure: Static, General
	Category Types for Selected Step
	Mechanical Concentrated force
	Thermal Moment
	Acoustic Pressure
Loads may only be created in an analysis step.	○ Fluid
Discourse in the different store	© Electrical Dine pressure
Please select a different step.	Mass diffusion Body force
	O Other Line load
	Gravity
	Bolt load 👻
Continue Cancel	Continue Cancel

Figure 29a. Create Load

Figure 29b. Create Load (AXIAL)

- Click Continue...
- Click the right vertical edge of the part.
- Click **Done**. The **Edit Load** dialog box will immediately appear (Figure 30a). Under **Magnitude** enter **-500**, under the **Traction is defined per unit** dropdown menu choose **undeformed area**. The **Edit Load** dialog box should look similar to Figure 30b.

💠 Edit Lo	bad 🛛 🕅
Name:	AXIAL
Type:	Shell edge load
Step:	LOADING STEP (Static, General)
Region:	(Picked)
Distributi	on: Uniform 💌 f(x)
Traction:	Normal
Magnitud	de:
Amplitus	le: (Ramp)
Traction	is defined per unit deformed area 💌
Follow ro	tation
	OK Cancel

Figure 30a. Edit Load Dialog Box

Edit Load
Name: AXIAL
Type: Shell edge load
Step: LOADING STEP (Static, General)
Region: (Picked)
Distribution: Uniform f (x)
Traction: Normal
Magnitude: -500
Amplitude: (Ramp)
Traction is defined per unit undeformed area 🔽
Follow rotation
OK Cancel

Figure 30b. Edit Load Dialog Box (AXIAL)

• Click **OK**. If this has been done correctly, small purple arrows will appear on that edge in the direction of the loading.

Creating a Job

• To create a job for this model, double click the **Jobs** icon in the model tree. Up to this point, you have been preprocessing the model. A job will take the input file created by the preprocessor and process the model, i.e. perform the analysis. In the **Create Job** dialog box, create a **Name**

for this job called **AXIAL**. Blank spaces are not allowed in a job name. Thus the use of the underline in the name. The Create Job dialog box should look identical to that in Figure 31.

Figure 31. Create Job Dialog Box (AXIAL)

- Click Continue...
- The Edit Job dialog box will immediately appear (Figure 32).

≑ Edit Job			X	3
Name: AXIAL				
Model: Mode	I-1			
Analysis produ	ict: Abaqus/Sta	ndard		
Description:				
Submission	General Mer	nory Parallelization	Precision	
Job Type -				
Full analy	/sis			
Recover	(Explicit)			
Restart				
Run Mode				
Background	nd 🔘 Queue:	▼ Hos Typ	st name: pe:	
- Submit Tim	e			
Immedia	tely			
🔘 Wait:	hrs. min.			
O At:		:		
	OK		Cancel	

Figure 32. Edit Job Dialog Box (AXIAL)

• Accept the default values and click **OK**.

Setting the Work Directory

• To ensure that the input files write to the correct folder, setting the work directory must be accomplished. At the top of the screen, click **File** and in the dropdown menu click **Set** <u>W</u>ork **Directory...** (Figure 33).

<u>File</u> <u>M</u> odel	Vie <u>w</u> port	View	Job	Adap
New				
<u>O</u> pen			Ctr	l+O
Network O	DB Connect	10		
Close ODB				
Set Work D	rectory			
Save			Ctr	l+S
Save <u>A</u> s				
Sa <u>v</u> e Optio	ns			
Import				•
Export				•
Run Script.				
Macro Mar	ager			
Print			Ctr	I+P
Abaqus PD	E			
1/TRUSS	TUTORIAL	CAE_FI	VAL.ca	e
2 D://011	911/10_BEA	MS_STA	CKED.	db
3 D://011	911/10_BEA	MS_SIDE	.odb	
4 D://10b	eam_SIDE.c	ae		
Exit			Ctr	+0

Figure 33. Set Work Directory

• The **Set Work Directory** screen will immediately appear (Figure 34). Click **Select...** and use standard Windows practice to select (and possibly create) a subdirectory.

- Set Work Directory				
Current work directory:				
C:\Users\Dimitri\Desktop\UML SPRING 2012				
New work directory:				
Note: In file selection dialog boxes, you can click the work directory icon to jump to the current work directory.				
OK Cancel				

Figure 34. Set Work Directory (FOLDERS)

- Click OK.
- Click OK.

Writing the Input File (.inp)

- To write the input file of the job that was created, first click the + next to the **Jobs (1)** icon in the model tree.
- Right click the job called **AXIAL** and click the **Write Input** option. This choice will write an input file (.inp) of this model to the work directory.
- It may be helpful to go to the folder on the computer to which the work directory is set to ensure that the input file was written there.

Model Analysis (ABAQUS Command)

Method #1

- Go to the Start Menu and open Abaqus Command
- Abaqus is set to a default directory (Example K:\>). To change directories in the Abaqus Command type the directory of choice followed by a colon (C:) then hit Enter.
- To access a specific directory within that drive type **cd** followed by the specific folder name in that directory (e.g., **cd APPLIED STRENGTHS T.A**) then hit **Enter**.
- Now that the correct directory has been sourced in the command window type **abaqus inter j=AXIAL** and then hit enter.
- If the job has completed successfully the Abaqus prompt should look similar to Figure 35.

C:\ProgramData\Microsoft\Windows\Start Menu\Programs\Abaqus 6.11-2\Abaqus Command.Ink	23
K:\>C:	<u>^</u>
C:\>cd users	=
C:∖Users≻cd Dimitri	
C:\Users\Dimitri>cd Desktop	
C:\Users\Dimitri\Desktop>cd UML SPRING 2012	
C:\Users\Dimitri\Desktop\UML SPRING 2012>abaqus inter j=AXIAL Abaqus JOB AXIAL Abaqus 6.11-2	
Begin Analysis Input File Processor 3/5/2012 11:26:50 AM	
Run pre.exe Abagus License Manager checked out the following licenses: Abagus/Standard checked out 5 tokens. K26 out of 50 licenses remain available). 43/5/2012 11:26:52 AM	
End Analysis Input File Processor Begin Abaqus/Standard Analysis 3/5/2012 11:26:52 AM Bun standard.exe	
Abaqus License Manager checked out the following licenses: Abaqus/Standard checked out 5 tokens. (26 out of 50 licenses remain available). 3/5/2012 11:26:55 AM End Abaqus/Standard Analysis	
Abaqus JOB AXIAL COMPLETÉD C:\Users\Dimitri\Desktop\UML SPRING 2012>	_

Figure 35. Abaqus Command Prompt (COMPLETED)

Method #2

- An alternative method for submitting an *.inp file for processing by Abawus can be accomplished with Abaqus/CAE.
- Right click the job called **AXIAL** and click the **Submit** option.
- If you see a warning (Figure 36) :



Figure 36. Abaqus Warning

Click **OK**. The intent of this warning is to prevent the user from accidentally overwriting a previously completed analysis with the same name.

• The model will now be submitted for analysis by Abaqus and the progress can be viewed in the status window at the bottom of the screen (Figure 37).

2	The job input file "AXIAL inp" has been submitted for analysis. Job AXIAL: Analysis Input File Processor completed successfully.			
Job AXIAL: Abaqus/Standard completed successfully. Job AXIAL completed successfully.				

Figure 37. Status Window

Postprocessing using Abaqus/CAE

- After the analysis has successfully completed in the Abaqus Command window using Method #1 or using Method #2, return to view the Abaqus/CAE viewport.
- Because the last step of creating the model was to create a job/write (and possibly submit) an input file, the **AXIAL (Completed)** job should still be highlighted in Abaqus/CAE model tree. **Right click** the **AXIAL (Completed)** job and then click **Results**.
- If this selection was done correctly, the model should turn to a green color and the geometry will have rotated to an isometric view (Figure 38).

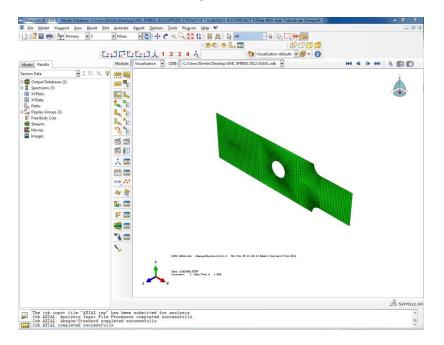


Figure 38. Analysis Results Isometric View

- To rotate the model back into the X Y plane for viewing, click <u>View</u> in the toolbar at the top of the screen. Next, Click **Toolbars** and make sure the option **Views** has a check mark to the left of it. If not, then click it.
- The **Views** toolbar will appear (Figure 39), and the **Apply Front View** button can be clicked to view the model in the X Y plane.



Figure 39. Views Toolbar

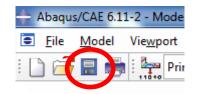
• To view the deformed shape of the model, click the **Plot Contours on Deformed Shape** icon in the **Visualization** module. The model should look similar to that in Figure 40.

Abaqus/CAE 6.11-2 - Model Database: C:\	Users\Dimitr\Desktop\UML SPRING 2012\APPLIED STRENGTHS T A\ABAQUS FILES\PROJECT 5\Plate With Hole Tutorial.cae [Viewport: 1]	= 0 ×
	It Plot Animate Report Options Icols Plug-ins Help % ?	
🗋 🚰 🚍 👘 🏪 Primary 💽 S	• Mises • 🐨 • 🕈 • • • • • • • • • • • • • • • • •	
	🎦 : : : : : : : : : : : : : : : : : : :	
Model Results	Module: Visualization 🔹 ODB: C/Users/Dimitri/Desktop/UML SPRING 2012/AXIAL.odb 🝷	► >> § 🕼 🕼
Session Data 🖉 🌲 🗈		X
 E Output Diablase (i) Spectrum (7) Spectrum (7) Notat Notat Notat Sologia (1) Sologia (1) Strama Strama Strama Images 		
	Source Control Title Title = 1.050 Non-rev Ver. 5. North Title = 1.050 Non-rev Ver. 5. North Source Factor + 8.002a+82	
	Z Z X	
		35 SIMULI
The job input file "AXIAL	.inp" has been submitted for analysis. File Frocessor completed successfully. d completed successfully.	

Figure 40. Deformed Shape (AXIAL)

Conclusion

• Save the file by doing either **File > Save** or clicking the disk icon.



- Close Abaqus/CAE: File > Exit or Ctrl+Q
- This completes the Finite Element Analysis of a Plate with Stress Concentrations Tutorial.