

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 an 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)

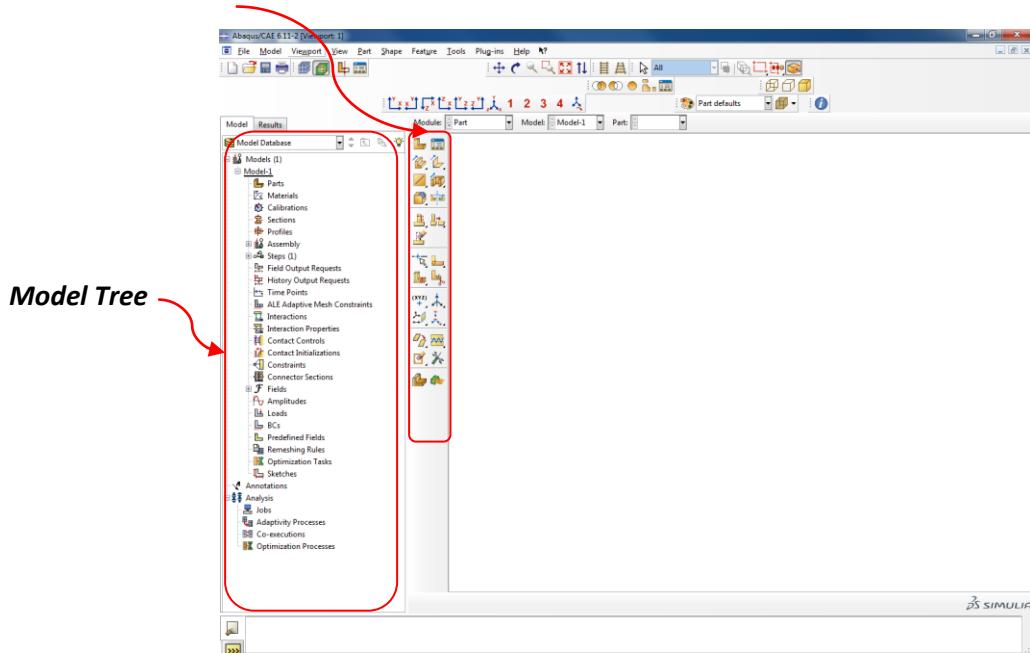


Figure 2. Abaqus/CAE Viewport

- 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.

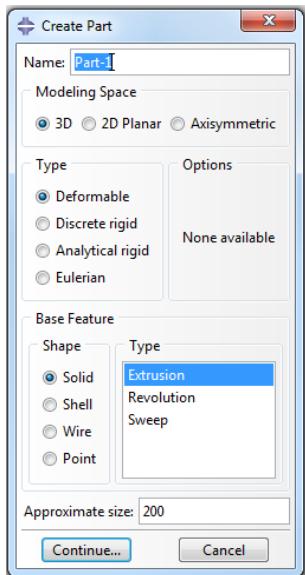


Figure 3a. Create Part Dialog Box

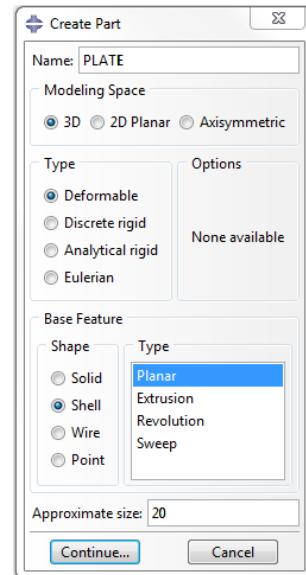


Figure 3b. Create Part Dialog Box (PLATE)

- 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)

Table 1. Points for Geometry

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

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.

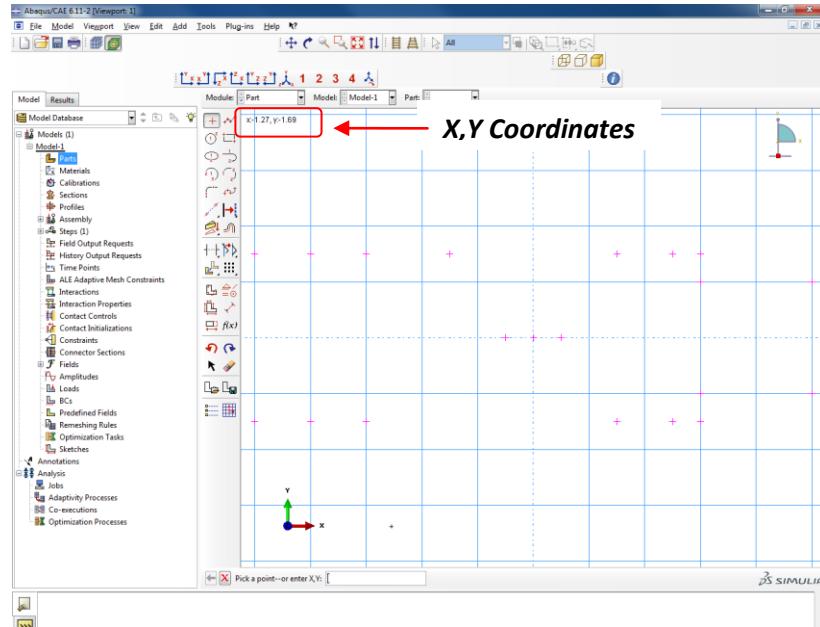


Figure 4. Geometry Points

- 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.

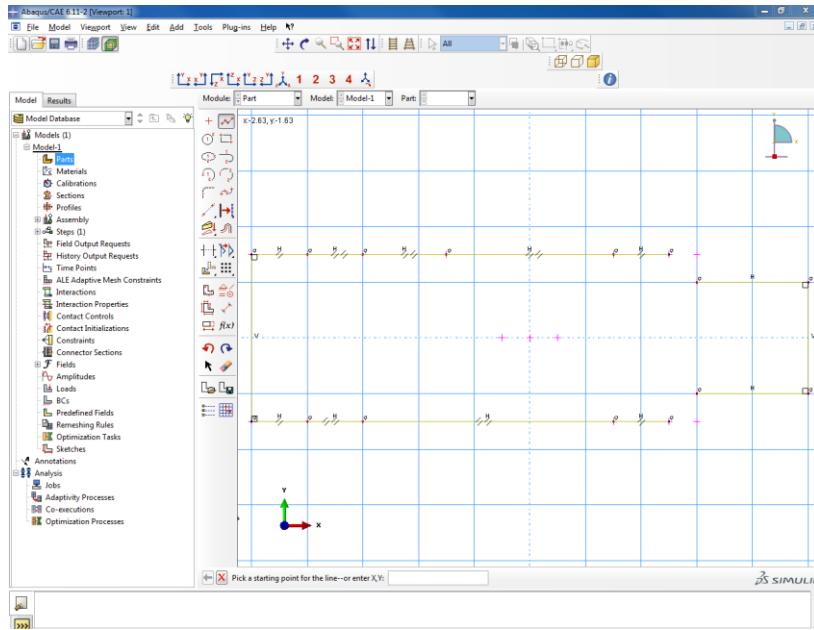


Figure 5. Geometry Lines

- 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** 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.

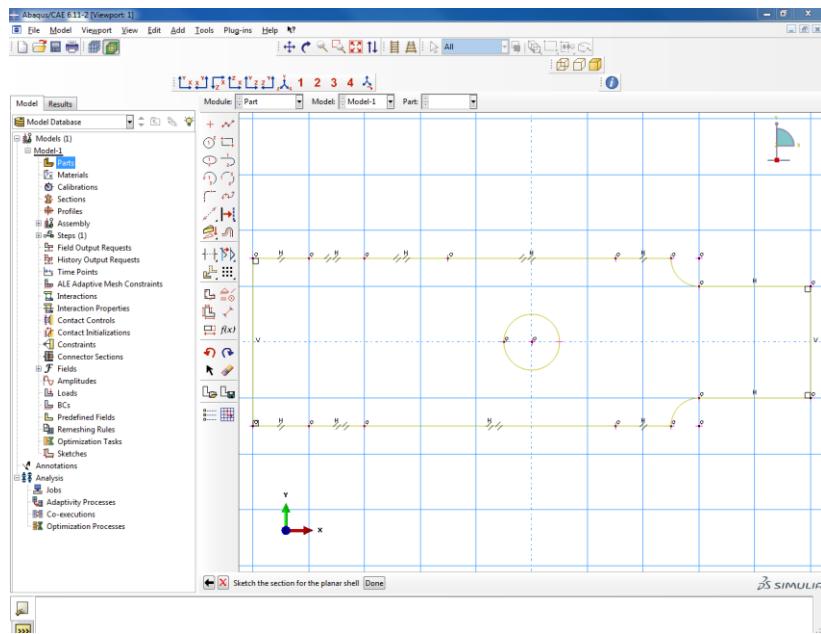


Figure 6. Geometry Complete

- 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).

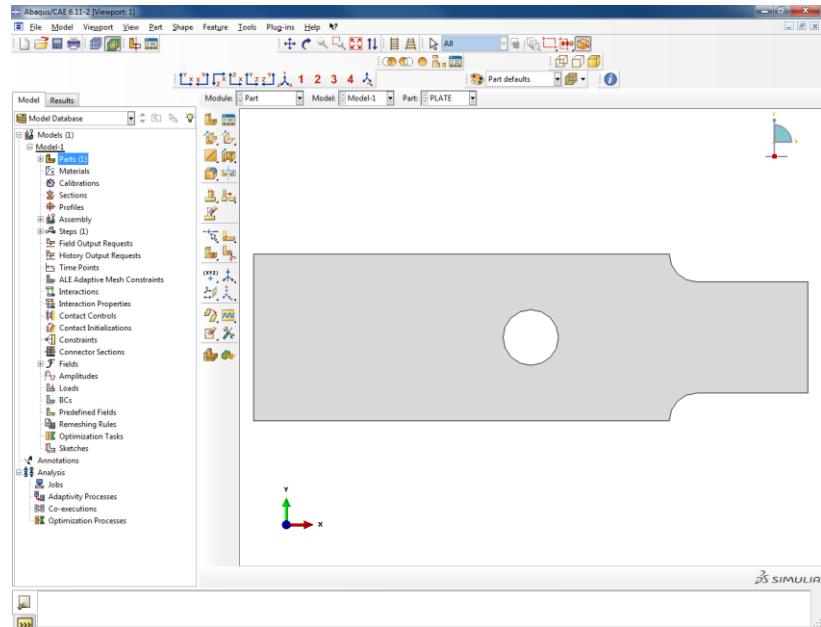


Figure 7. Plate with Hole (Part Module)

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**.

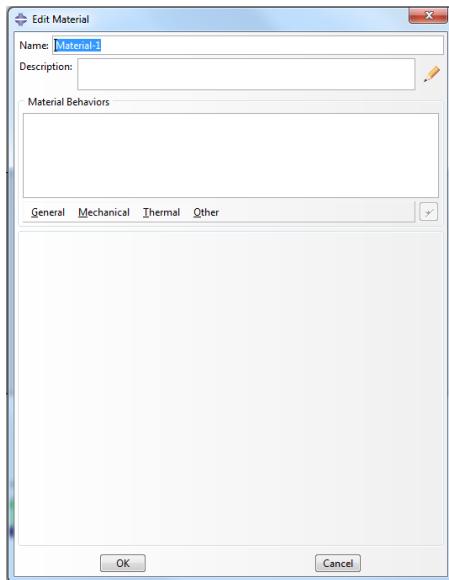


Figure 8a. Edit Material Dialog Box

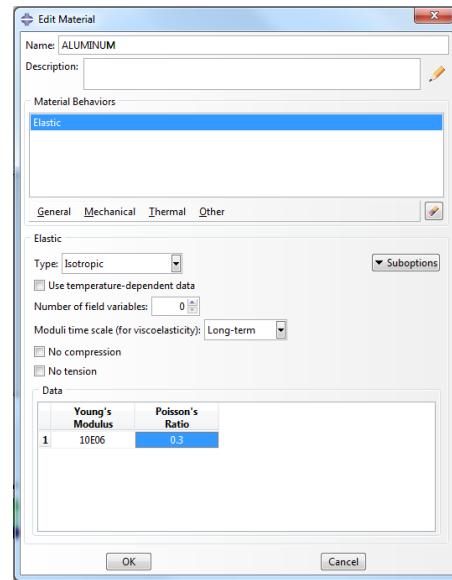


Figure 8b. Edit Material Dialog Box (ALUMINUM)

- 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...**

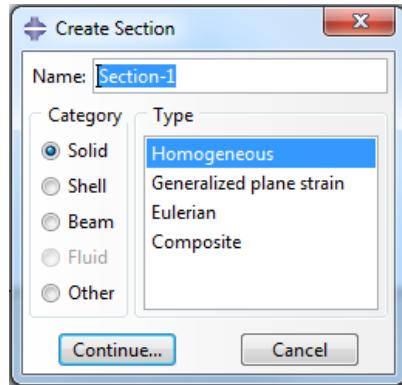


Figure 9a. Create Section Dialog Box

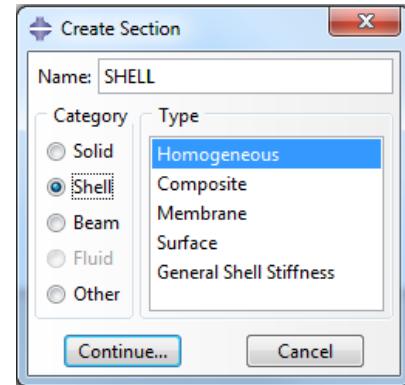


Figure 9b. Create Section Dialog Box (SHELL)

- 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**.

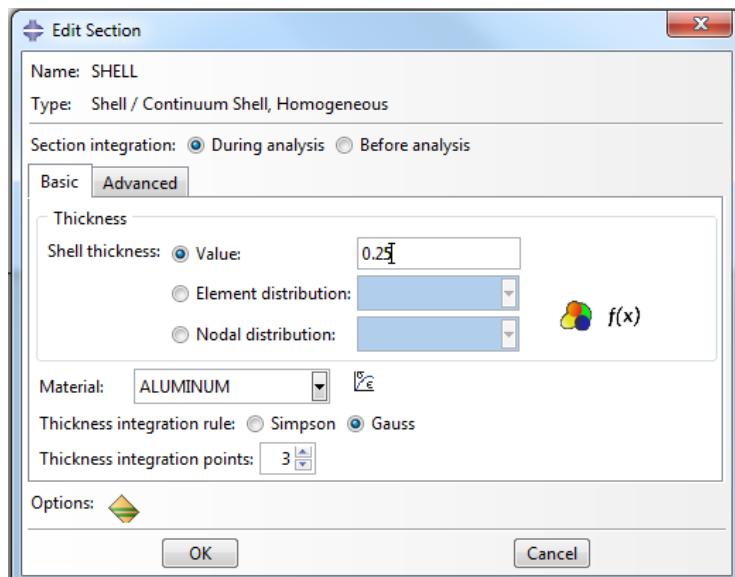


Figure 10. Edit Section Dialog Box (SHELL)

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).

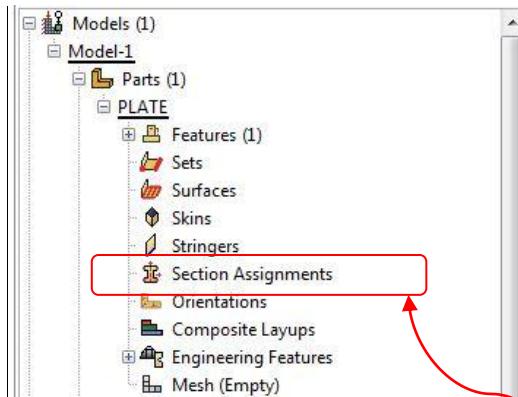


Figure 11. Model Tree Expansion (Parts)

- 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**.

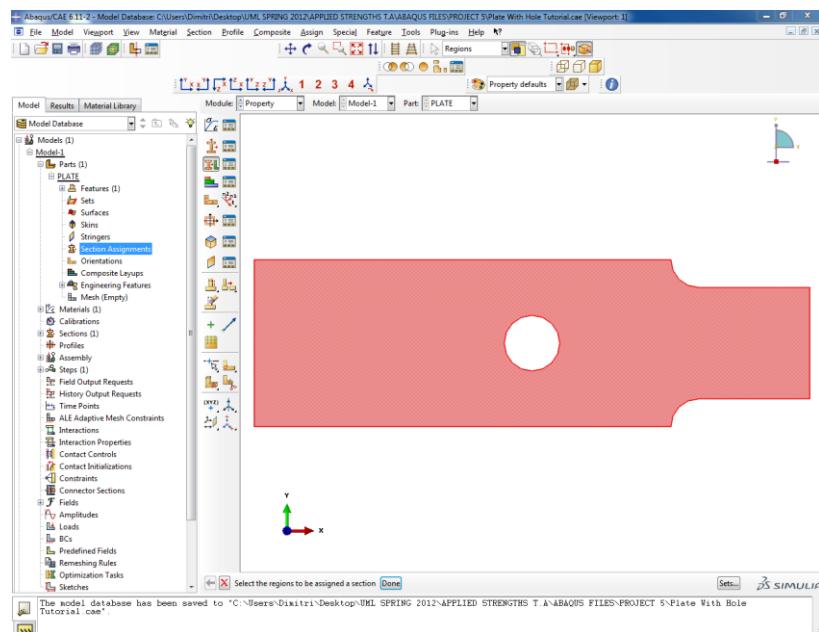


Figure 12. Selected Geometry

- 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.

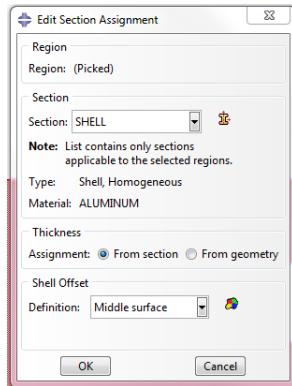


Figure 13. Edit Section Assignment (SHELL)

- 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.

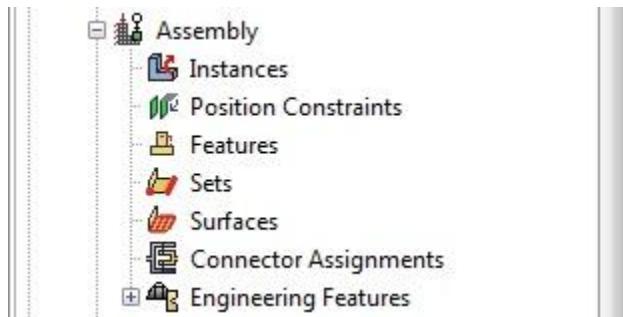


Figure 14. Model Tree Expansion (Assembly)

- 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)**.

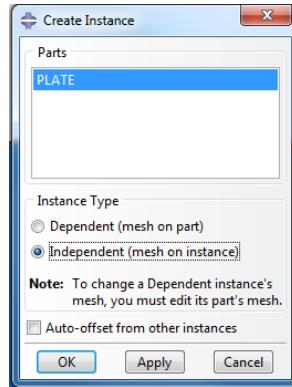


Figure 15. Create Instance Dialog Box

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

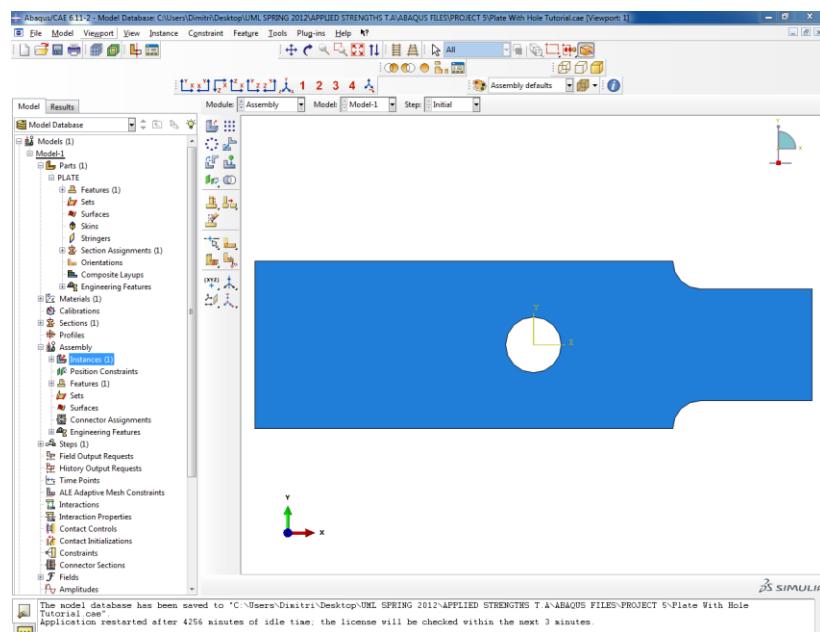


Figure 16. Instance Part

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.

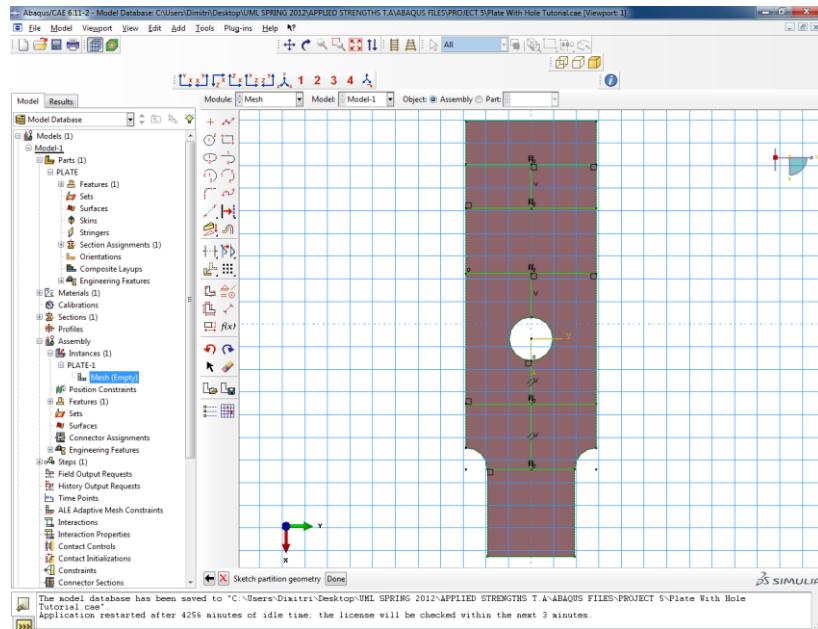


Figure 17. Partitioned Geometry

- 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.

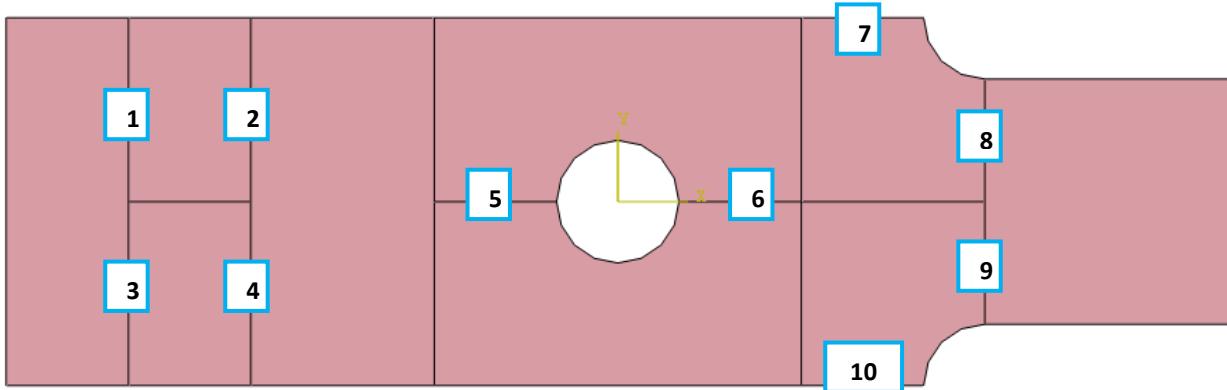


Figure 18. Selected Biased Edges

Table 2. 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

- 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.

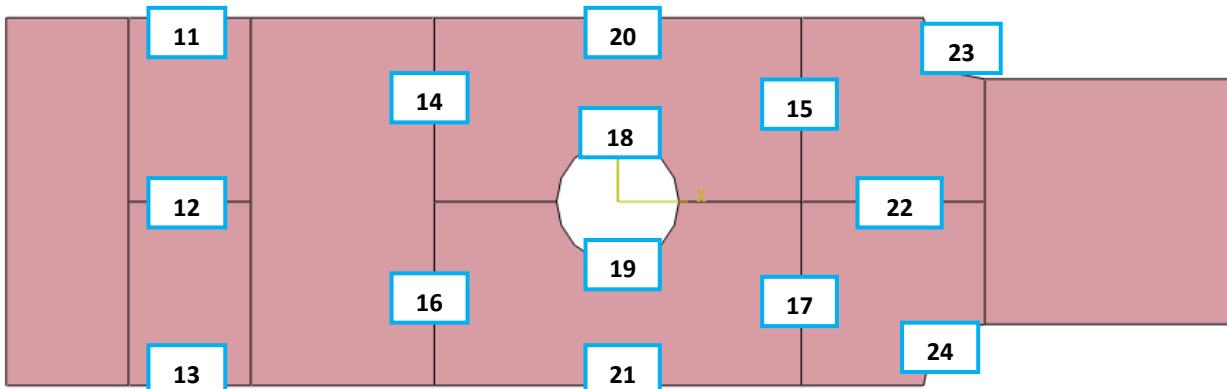


Figure 19. Selected Equal Edges

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.

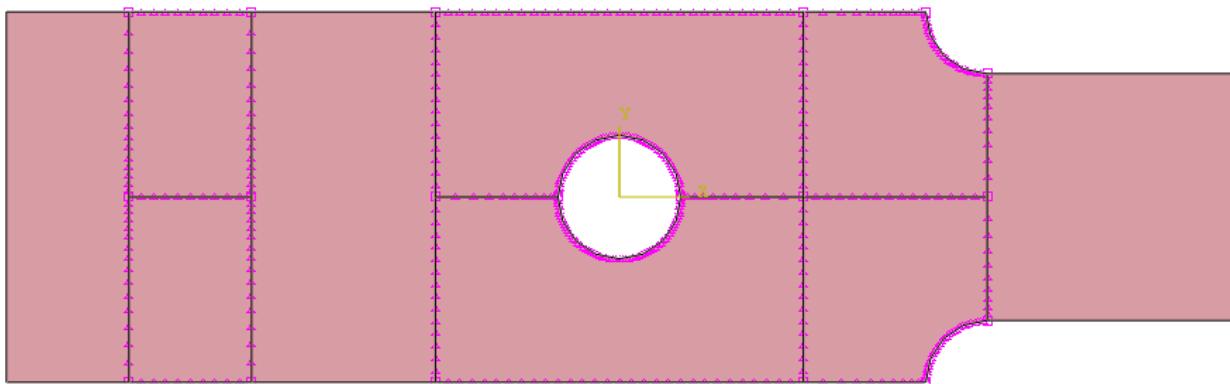


Figure 20. Seeded Part

- 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).

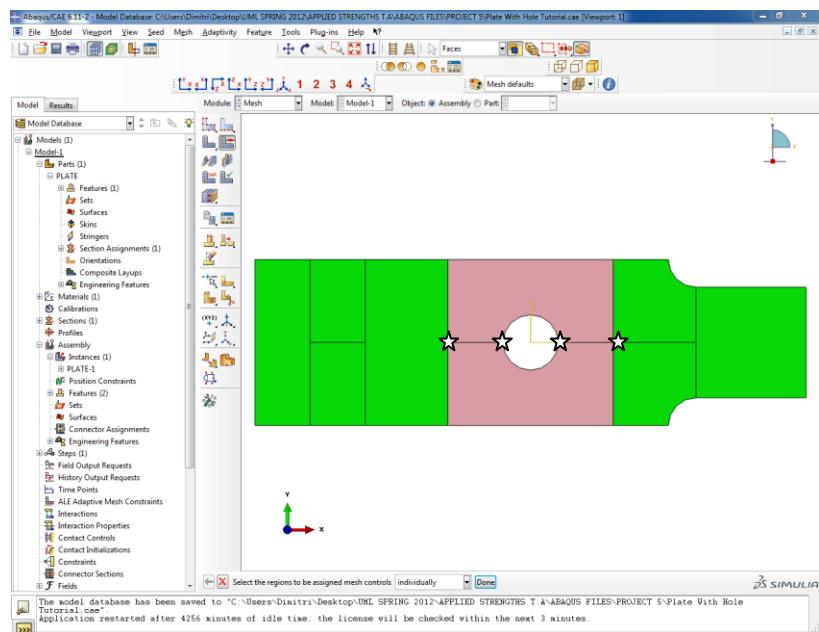


Figure 21. Mesh Regions

- 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.

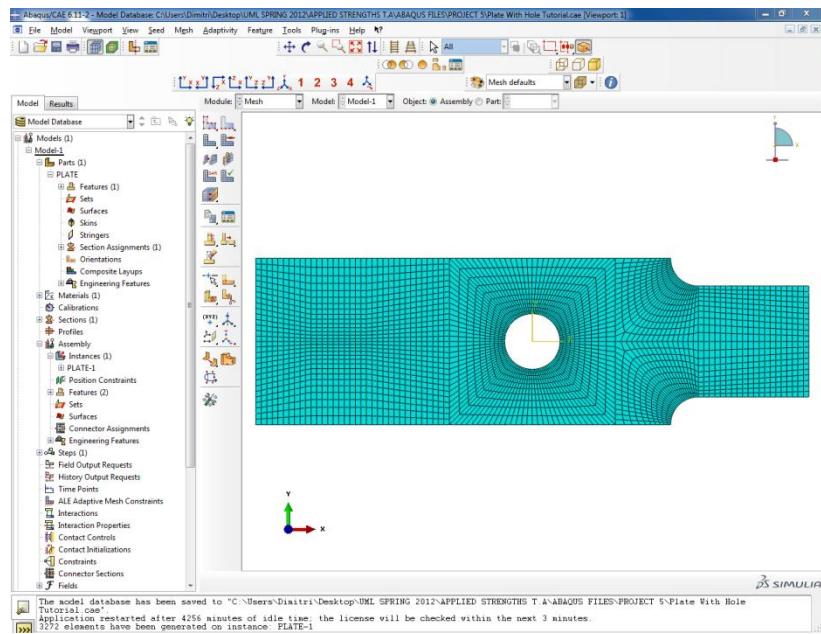


Figure 22. Meshed Part

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.

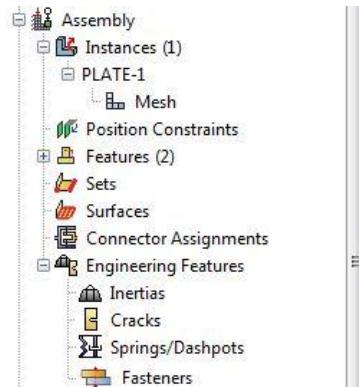


Figure 23. Model Tree Expansion (Crack)

- 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 **Crack** and click **Assign Seam...** (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.

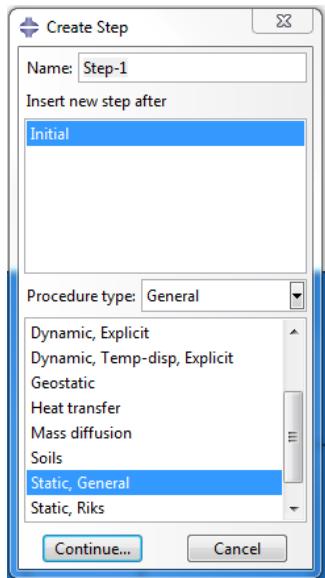


Figure 25a. Create Step Dialog Box

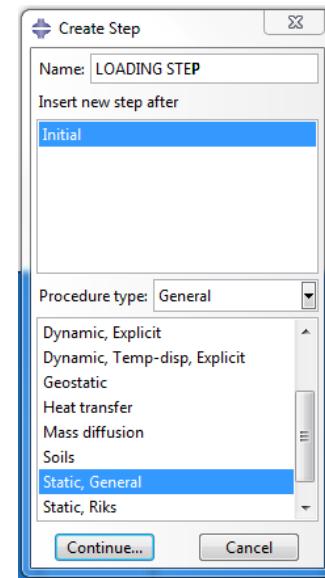


Figure 25b. Create Step Dialog Box (LOADING STEP)

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

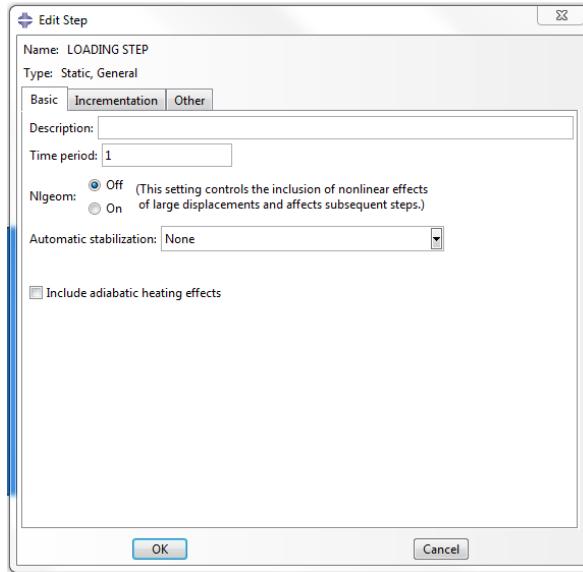


Figure 26. Edit Step Dialog Box

- 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.

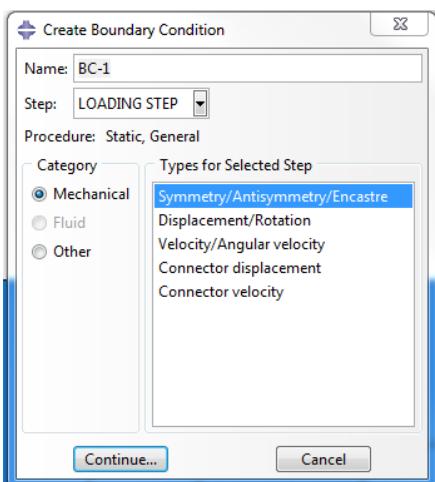


Figure 27a. Create Boundary Condition

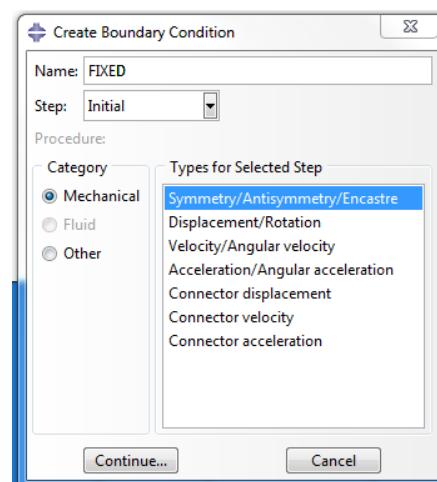


Figure 27b. Create Boundary Condition (FIXED)

- 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.

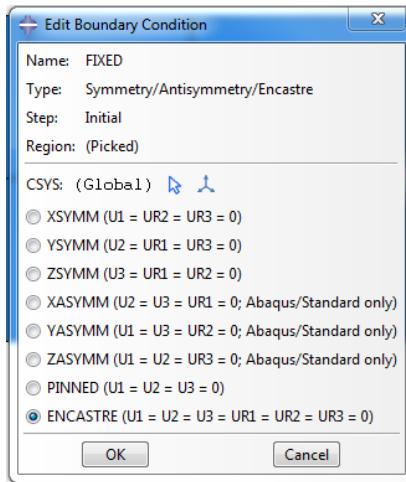


Figure 28. Edit Boundary Condition Dialog Box

- 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.

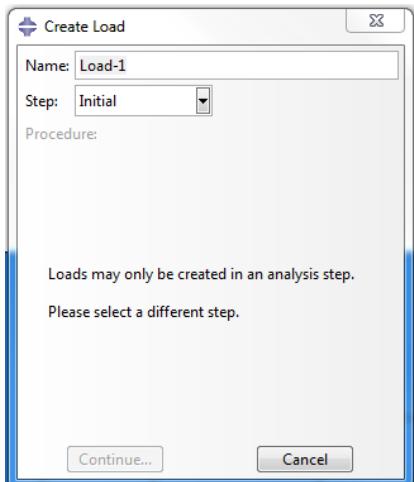


Figure 29a. Create Load

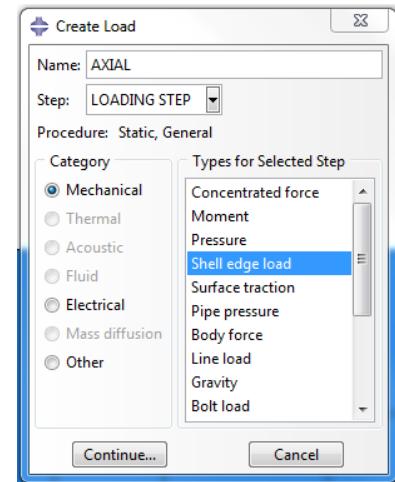


Figure 29b. Create Load (AXIAL)

- 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.

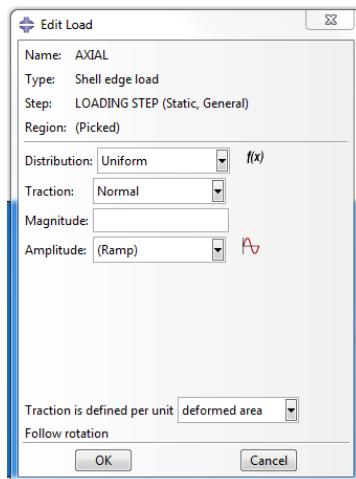


Figure 30a. Edit Load Dialog Box

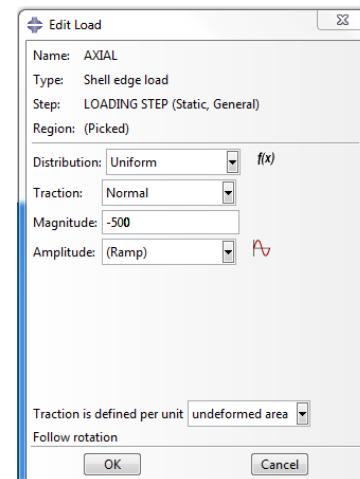


Figure 30b. Edit Load Dialog Box (AXIAL)

- 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.

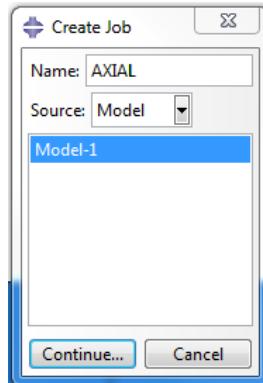


Figure 31. Create Job Dialog Box (AXIAL)

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

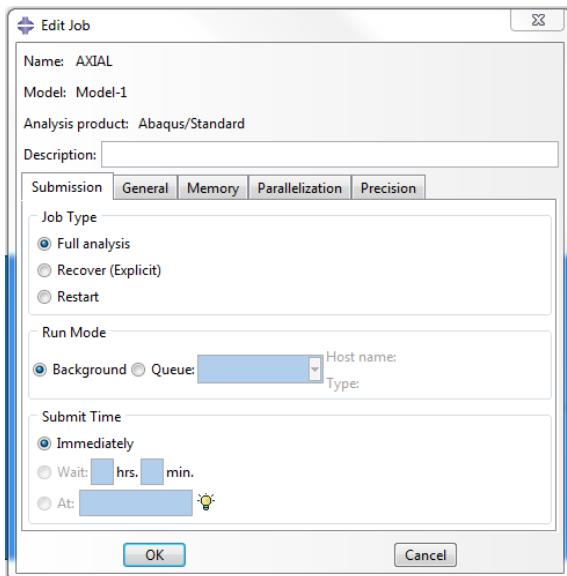


Figure 32. Edit Job Dialog Box (AXIAL)

- 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 Work Directory...** (Figure 33).

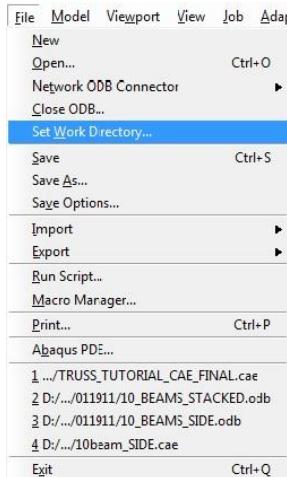


Figure 33. Set Work Directory

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

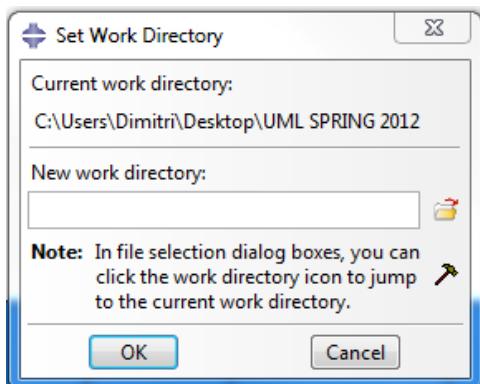


Figure 34. Set Work Directory (FOLDERS)

- 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.

```
K:\>C:
C:\>cd users
C:\Users>cd Dimitri
C:\Users\Dimi>cd Desktop
C:\Users\Dimi>cd UML SPRING 2012
C:\Users\Dimi>cd 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
Abaqus License Manager checked out the following licenses:
Abaqus/Standard checked out 5 tokens.
<26 out of 50 licenses remain available>
3/5/2012 11:26:52 AM
End Analysis Input File Processor
Begin Abaqus/Standard Analysis
3/5/2012 11:26:52 AM
Run 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 COMPLETED
C:\Users\Dimi>
```

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) :

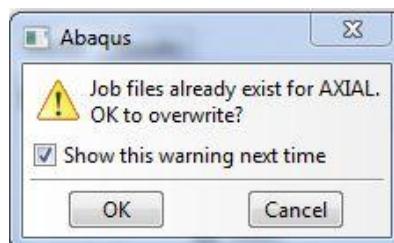


Figure 36. Abaqus Warning

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).

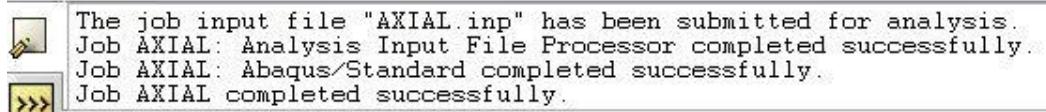


Figure 37. Status Window

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).

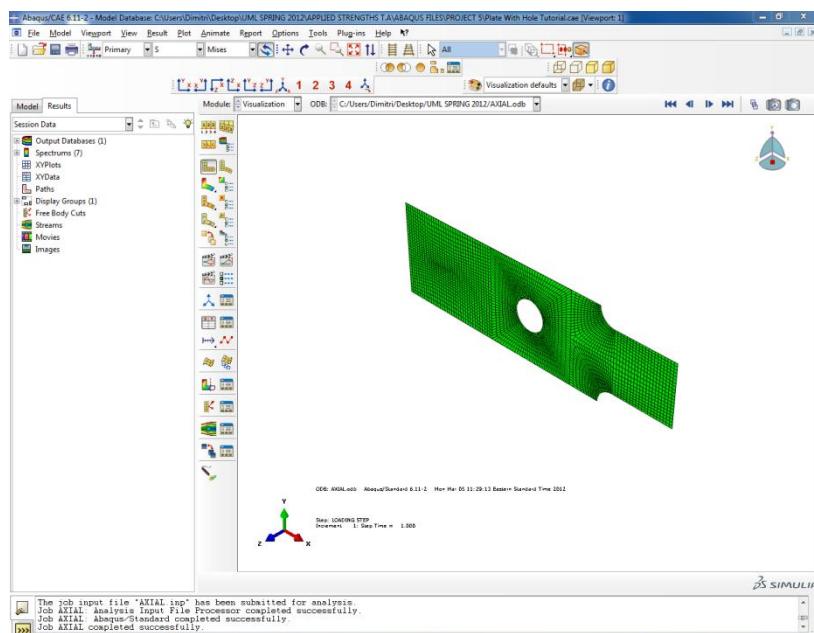


Figure 38. Analysis Results Isometric View

- To rotate the model back into the X Y plane for viewing, click **View** 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.

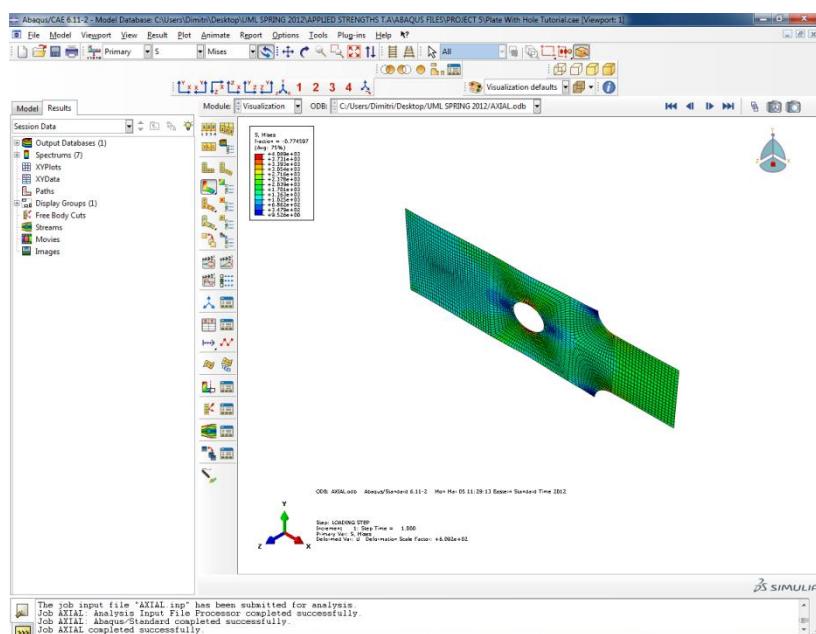
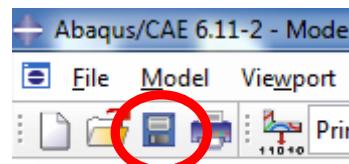


Figure 40. Deformed Shape (AXIAL)

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.