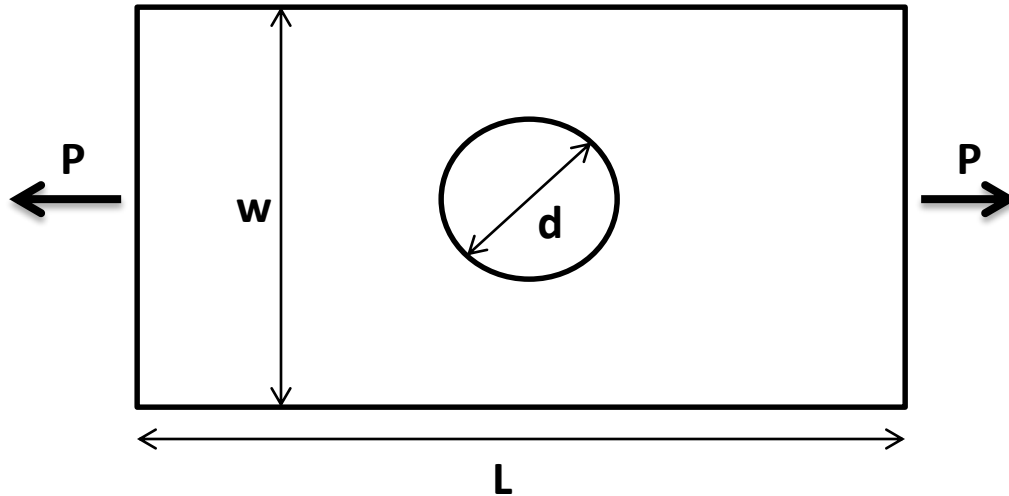


Finite Element Analysis (FEA) Tutorial

Project 2: 2D Plate with a Hole
Problem

Problem

- Analyze the following plate with hole using FEA tool ABAQUS



Dimensions:

$$t = 3 \text{ mm}$$

$$w = 50 \text{ mm}$$

$$d = 10 \text{ mm}$$

$$L = 100 \text{ mm}$$

$$P = 5 \text{ kN} = 5000 \text{ N}$$

Allowable Stress:

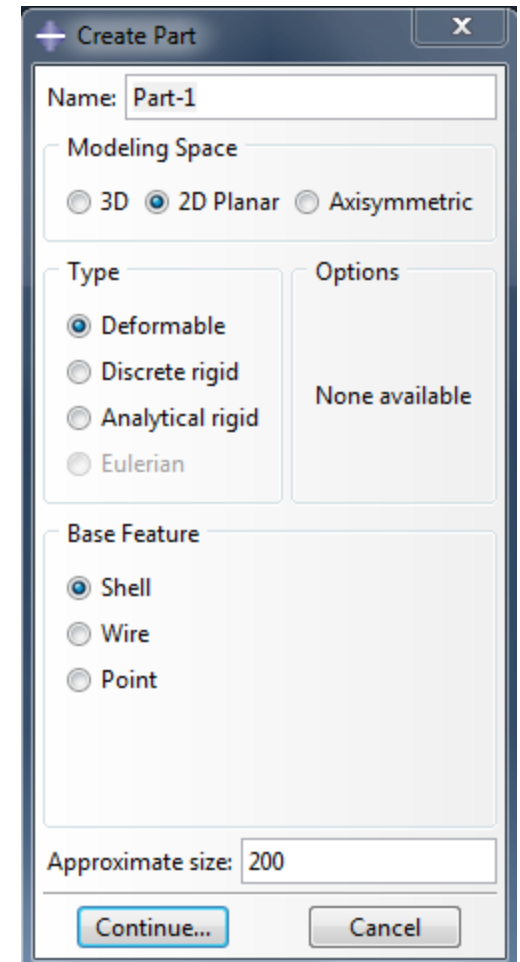
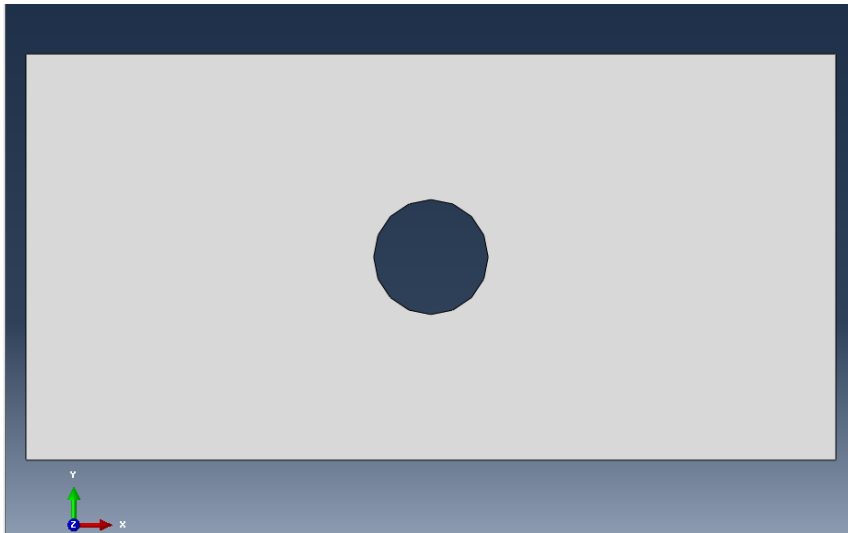
$$\sigma_{\text{allow}} = 120 \text{ MPa}$$

- Determine:
 - The stress concentration factor
 - The factor of safety
- Plot:
 - Deformed shape
 - Stress contour
 - Stress diagram along a line

Preprocessing

Creating the Geometry

- Open ABAQUS/CAE and create a new part: Plate
- Using rectangle tool, create a rectangle using two corner points (-50,-25) and (50,25)
- Create a circle using center at (0,0) and a perimeter point at (5,5); Click **Add Dimension** and put **5** as **Radius**



Creating the Materials

- Double click **Materials** under Model tree to create **Material-1**
 - Edit Material window will pop up
 - Click **Mechanical > Elasticity > Elastic**
 - Write down the properties:
 - Young's Modulus: 200E3
 - Poisson's Ratio: 0.25
 - Click **Ok**

Edit Material

Name: Material-1

Description:

Material Behaviors

Elastic

General Mechanical Thermal Electrical/Magnetic Other

Elastic

Type: Isotropic

☐ Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

☐ No compression

☐ No tension

Data

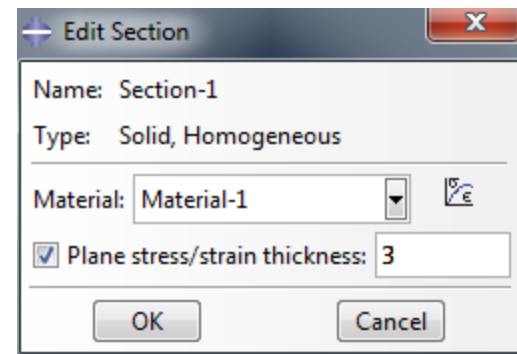
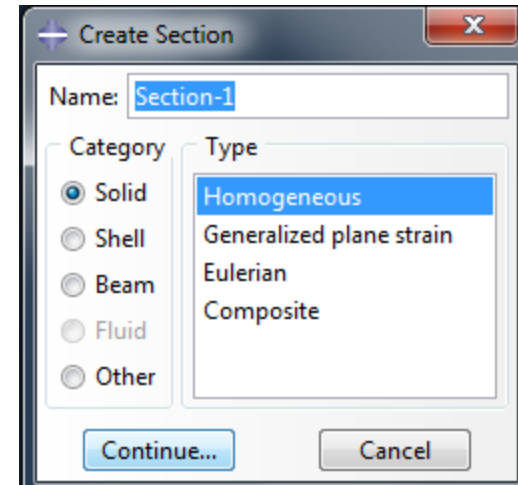
	Young's Modulus	Poisson's Ratio
1	200E3	0.25

OK Cancel

Note: Stress concentration factor is independent of material. It only depends on geometry. Choose any material.

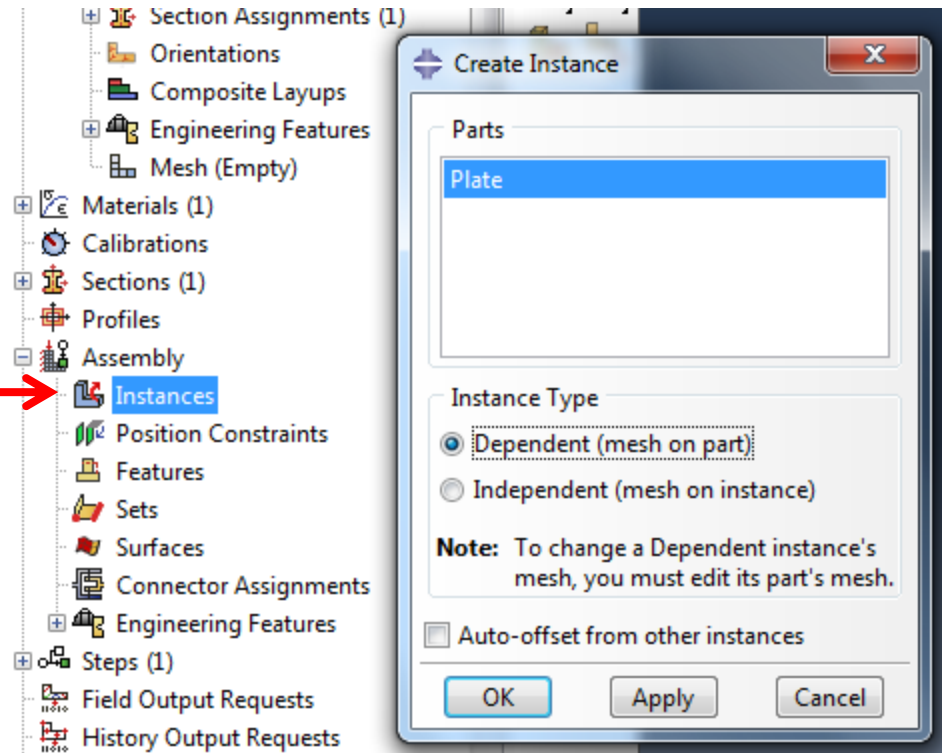
Creating the Section and Section Assignments

- Double click **Sections** under Model tree to create **Section-1**
 - **Create Section** window will pop up
 - **Name:** Section-1
 - **Category:** Solid
 - **Type:** Homogeneous
 - **Edit Section** window will pop up
 - Select **Material-1** from the Material drop down list
 - Put **3** in the **Plane stress/strain thickness** and click **Ok**
- Create a **Section Assignment** using **Section-1** (Ref. Truss Problem Tutorial)



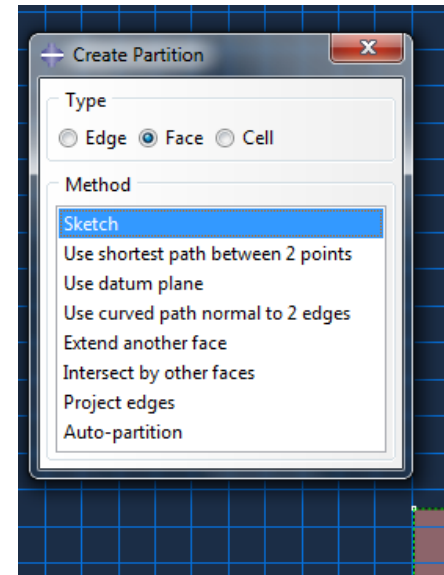
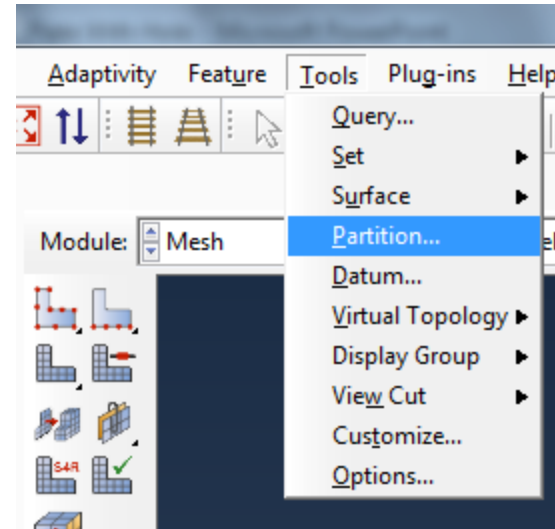
Creating an Assembly

- Under Model-1, go to **Assembly > Double click Instances** to create an instance
- **Create Instance** window will pop up
 - Click **Ok**
 - Tips: Plate will turn into blue



Creating Mesh (Partitioning)

- Before generating mesh, the geometry needs to be partitioned-
 - Mapped mesh can be applied
 - Less number of elements will be required
- Go to **Parts > Plate** and double click **Mesh (Empty)** to enter in the Mesh module
- Go to Tool>Partition
 - Create Partition window will pop up
 - Type: Face
 - Method: Sketch
 - Click 'x' to close the Create Partition window



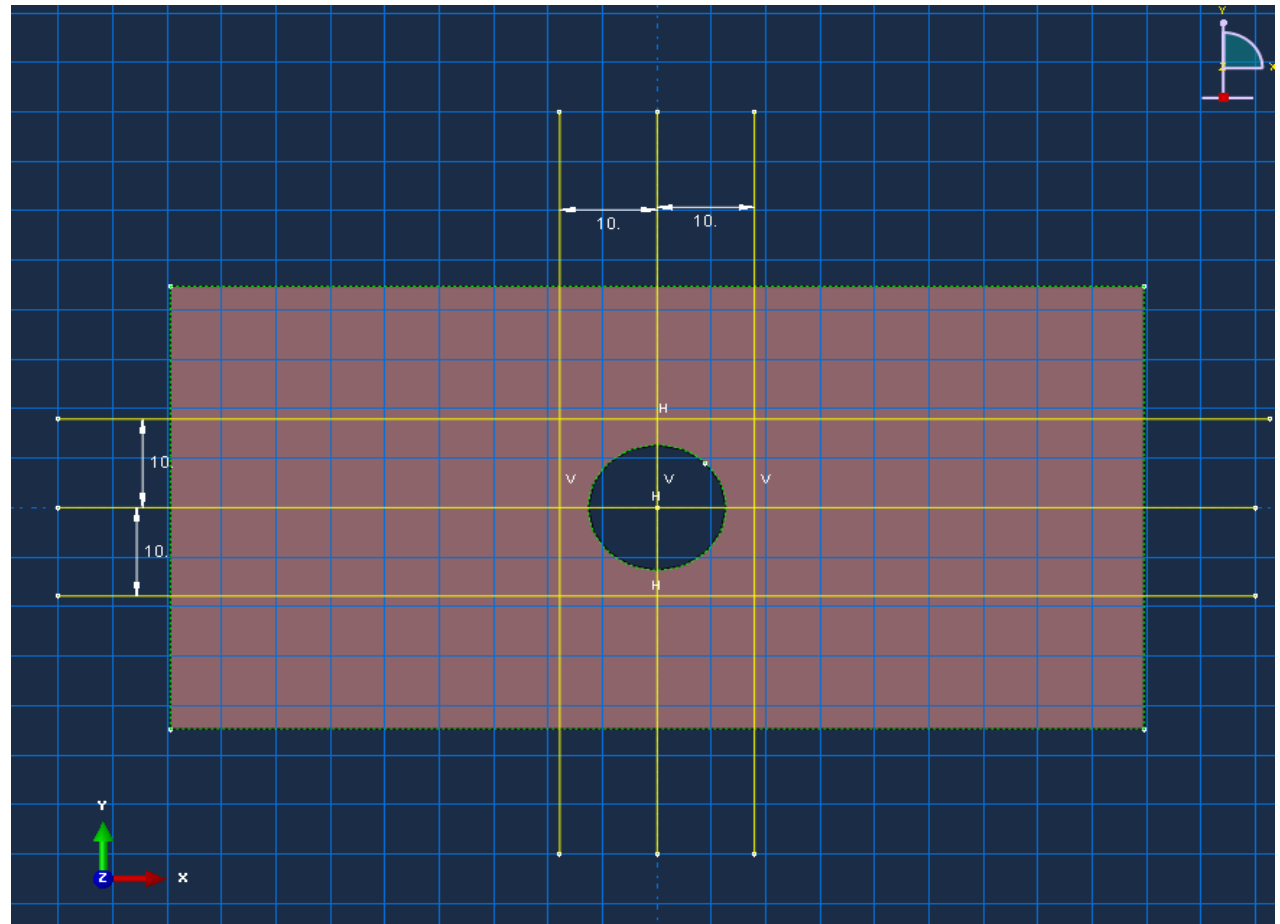
Creating Mesh (Partitioning)

- Create 1 vertical and 1 horizontal line through the center of the hole
- Create 2 arbitrary vertical and 2 arbitrary horizontal lines
- Using **Add Dimension**, set the lines 10 mm apart from the center lines
- Click the middle mouse button/**Done** to complete partitioning

Create Lines →

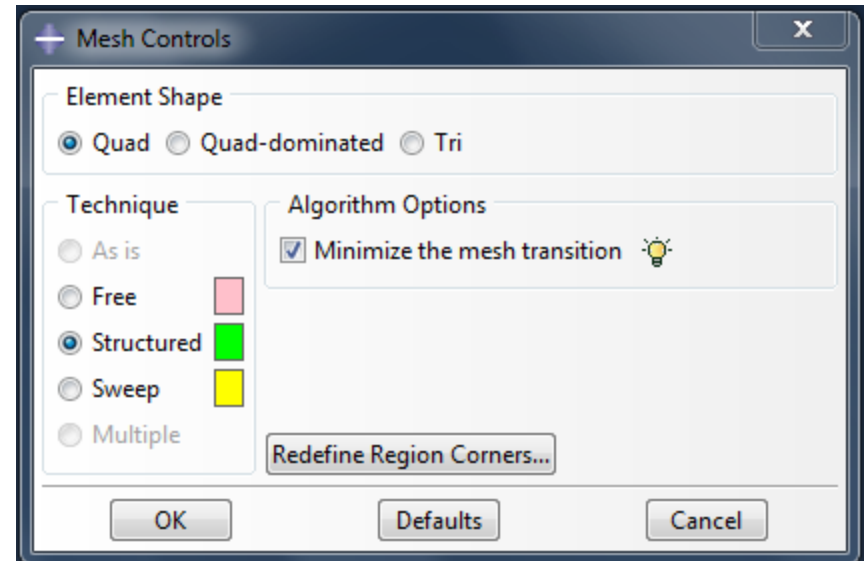
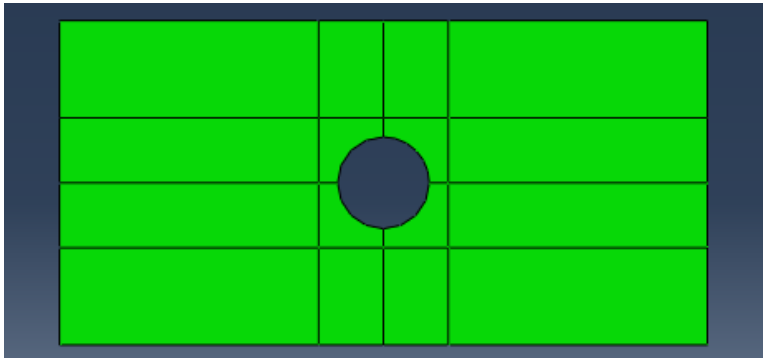
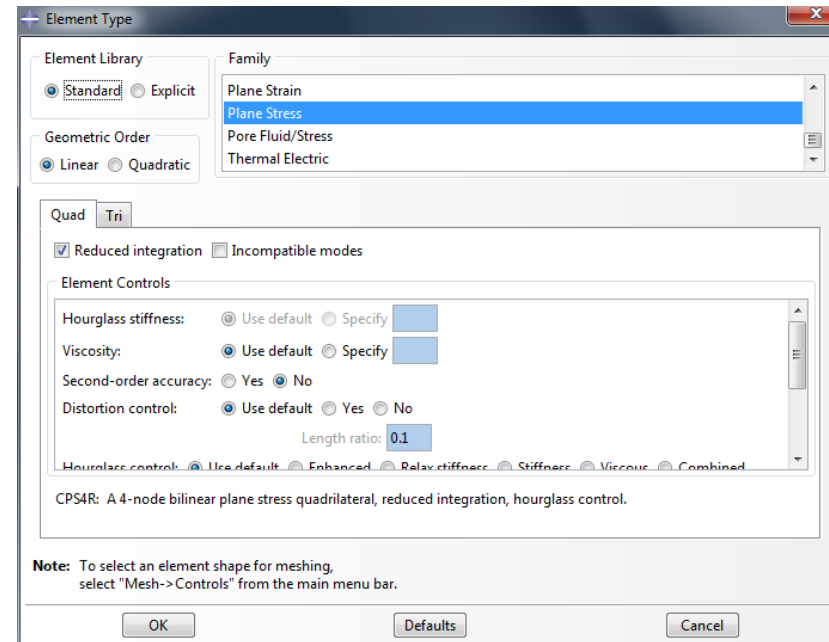


Add
Dimension →



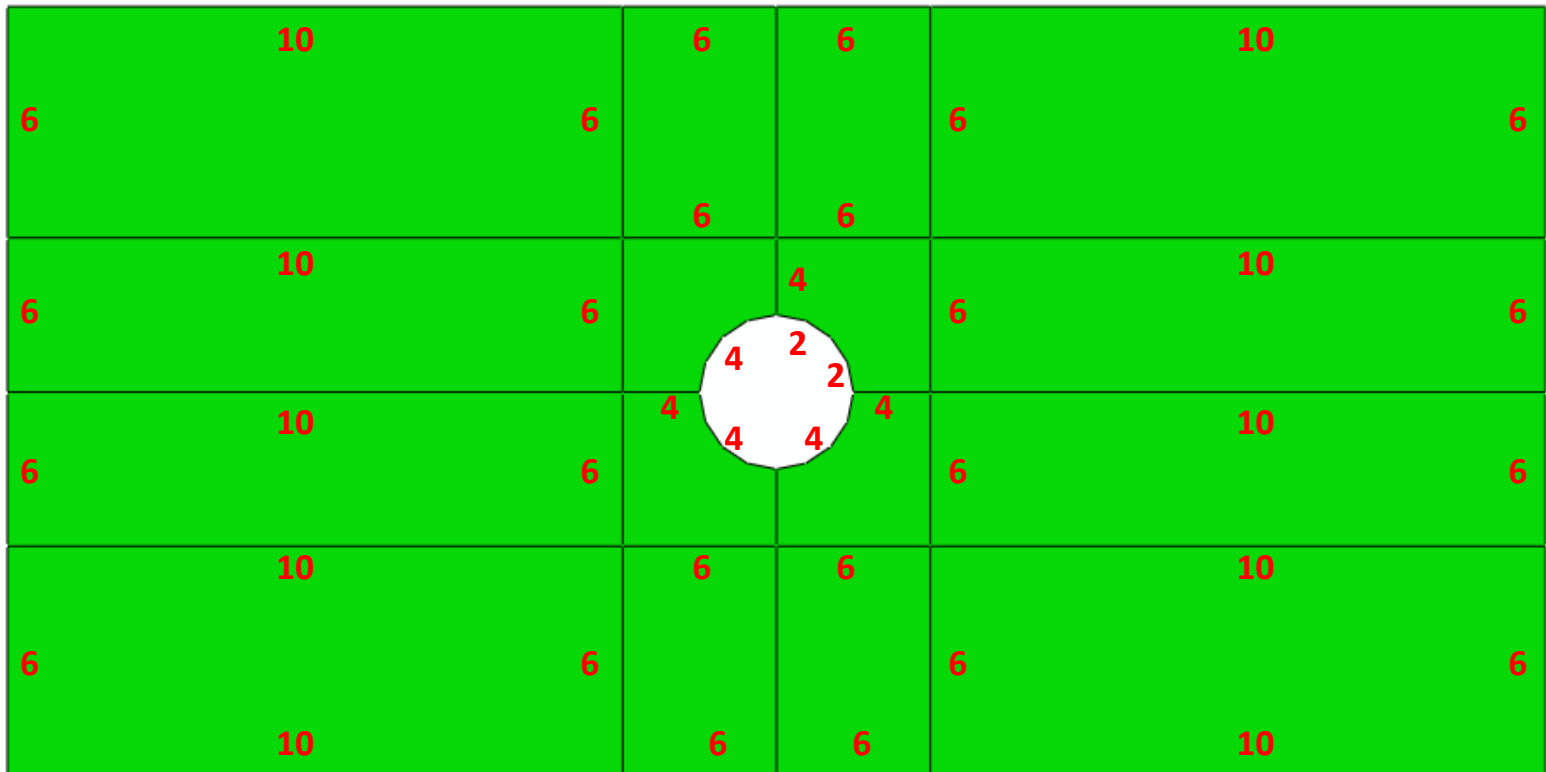
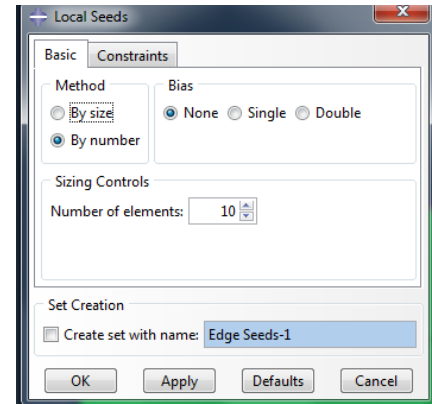
Creating Mesh (Meshing)

- Select **Mesh > Element Type** from the menu bar
 - Select the whole plate by holding the left mouse button, dragging and releasing the left mouse button
 - Tips: Zoom out to make the plate smaller
 - Hit **Done**
- **Element Type** window will pop up
 - From the **Family** select **Plate Stress** and hit **Ok**
- Select **Mesh > Control** from the menu bar
 - Select the whole plate; **Mesh Control** window will pop up
 - **Element Shape: Quad**
 - **Technique: Structured**



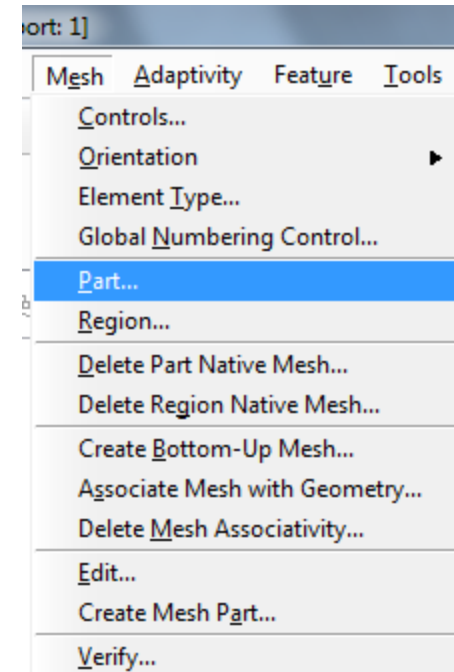
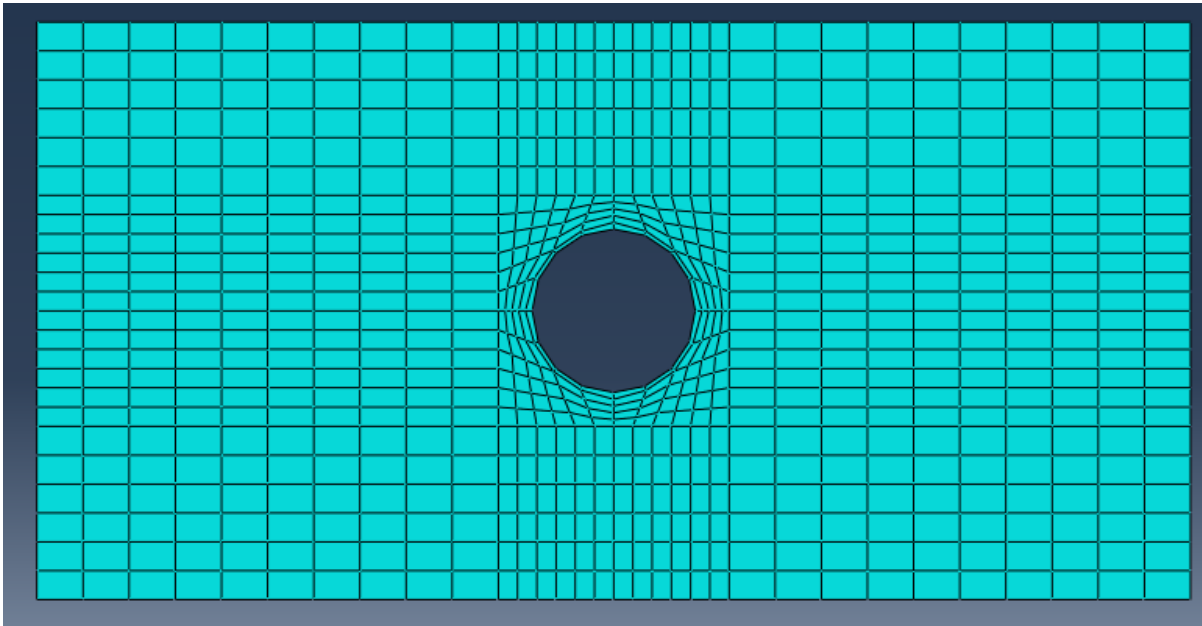
Creating Mesh (Meshing)

- Select **Seed > Edge** from the menu bar
 - Select the top left horizontal line section
 - Click middle mouse button
 - **Local Seeds** window will pop up
 - **Method:** By number
 - **Sizing Controls** → **Number of elements:** 10
- Using the same method, complete seeding all the lines as following
- **Important Tips:** Hold the **Shift** button to select multiple lines for seeding them altogether



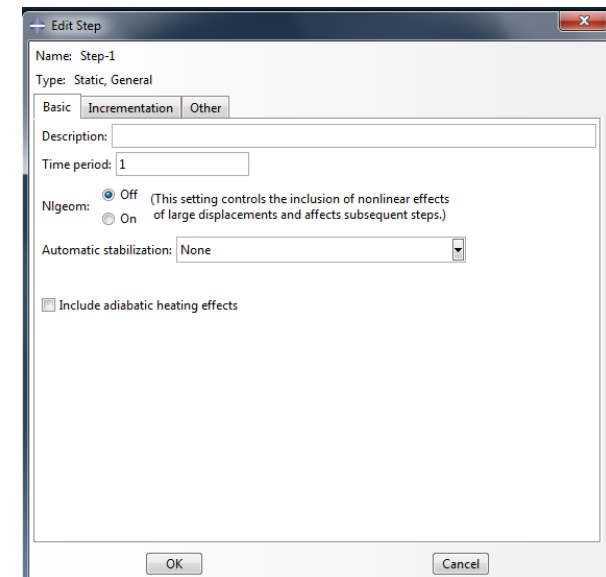
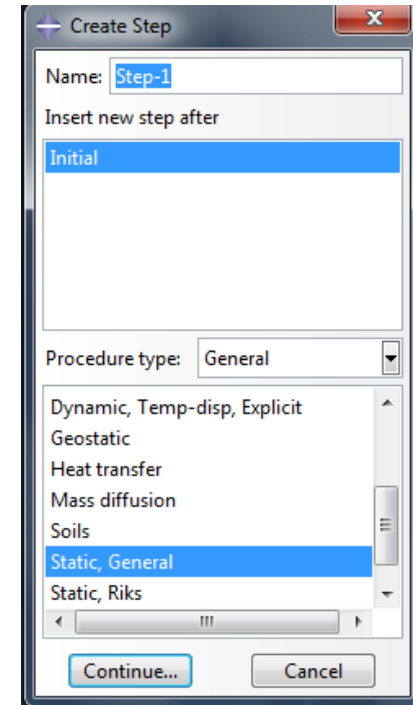
Creating Mesh (Meshing)

- Go to **Mesh>Part**
 - Click **Ok** to generate mesh



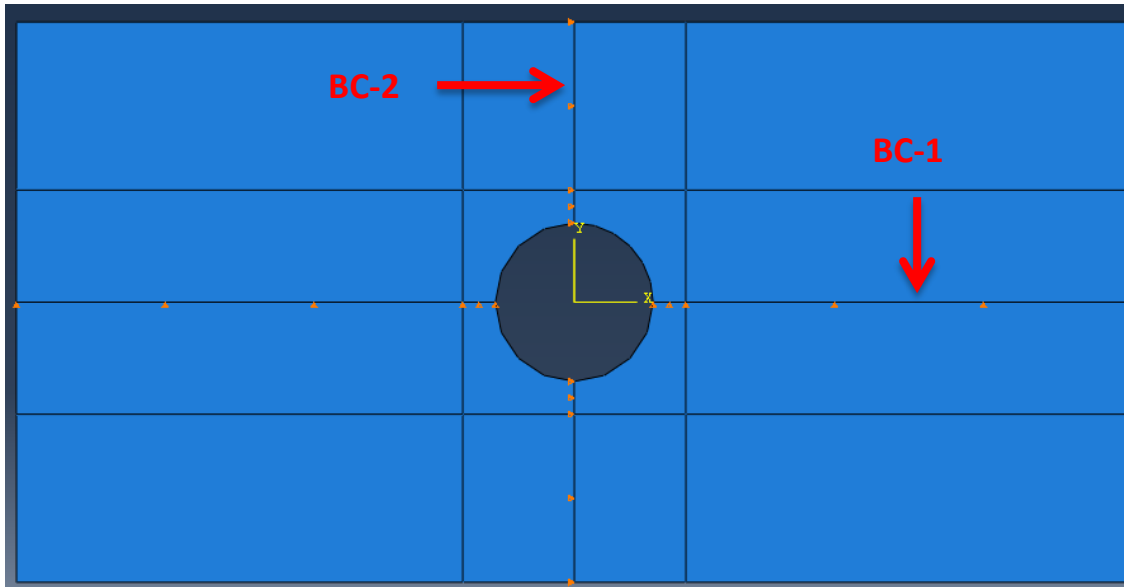
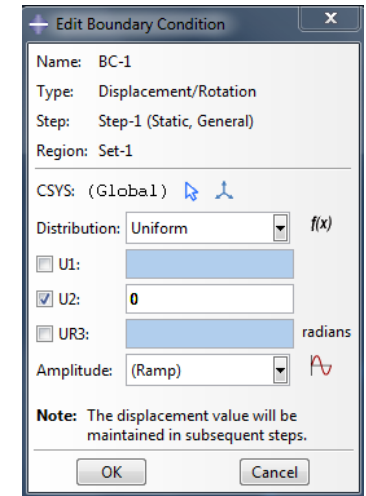
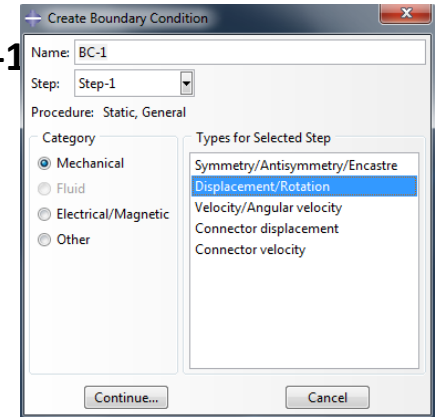
Creating a Step

- Double click **Steps** under Model tree and **Create Step** window will pop up
 - Keep the properties in default
 - Name: Step-1
 - Procedure Type: General
 - Static, General
 - Hit Continue
- **Edit Step** window will pop up
 - Hit **Ok**



Creating Boundary Conditions (BCs)

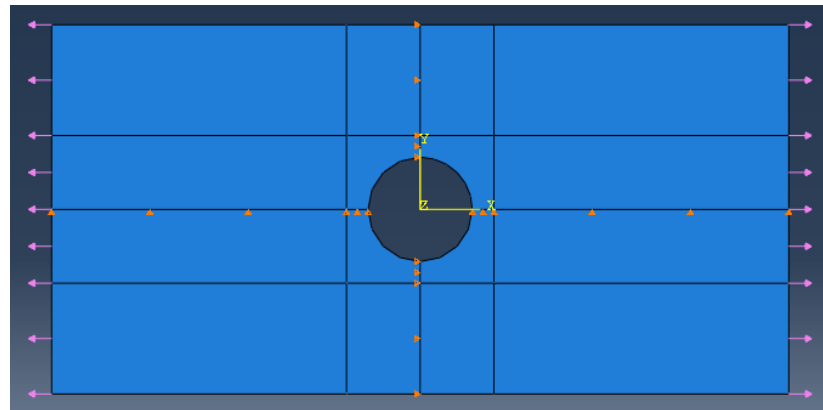
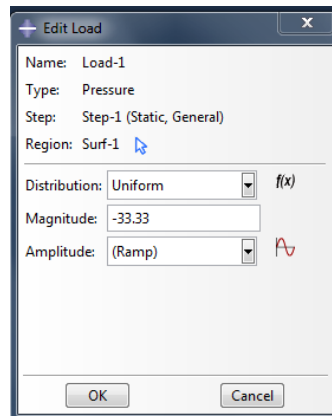
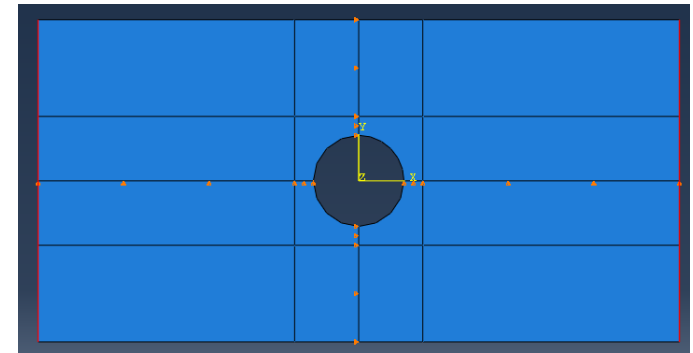
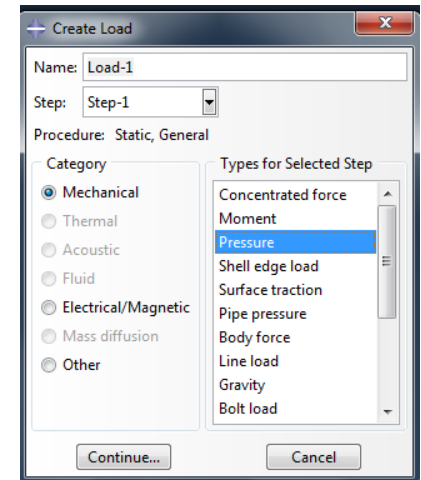
- Go to **Steps > Step-1** and double click **BCs** to create boundary condition **BC-1**
- Create Boundary Condition** window will pop up
 - Name:** BC-1
 - Steps:** Step-1
 - Category:** Mechanical
 - Types for Selected Step:** Displacement/Rotation
- Hit **Continue**, select the horizontal centerline and hit **Done**
- Edit Boundary Condition** window will pop up
 - Click the check box **U2** and put **0** in the right side box
- Create **BC-2** by selecting vertical centerline
 - U1 = 0, U2 = 0**



Creating Loads

- Go to **Steps > Step-1** and double click **Loads** to create load
- **Create Load** window will pop up
 - **Name:** Load-1
 - **Steps:** Step-1
 - **Category:** Mechanical
 - **Types for Selected Step:** Pressure
- Hit **Continue**, select the left and right vertical lines at the edges, and hit **Done**
- **Edit Load** window will pop up. Put,
 - **Magnitude** = -33.33
 - Hit **Ok**

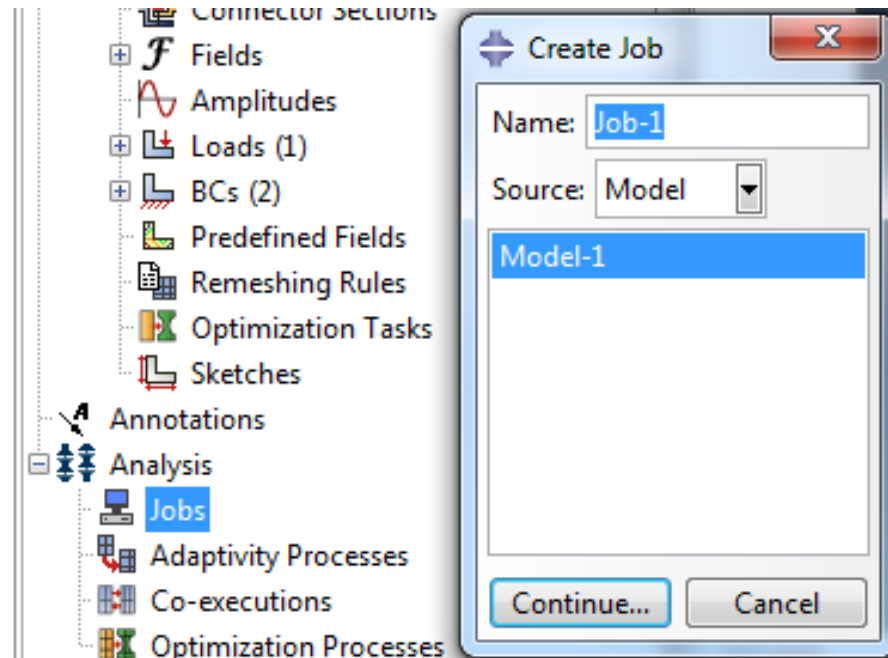
$$\text{Pressure} = P/(w*t) = 5000/(50*3) = 33.33 \text{ MPa}$$



Solving

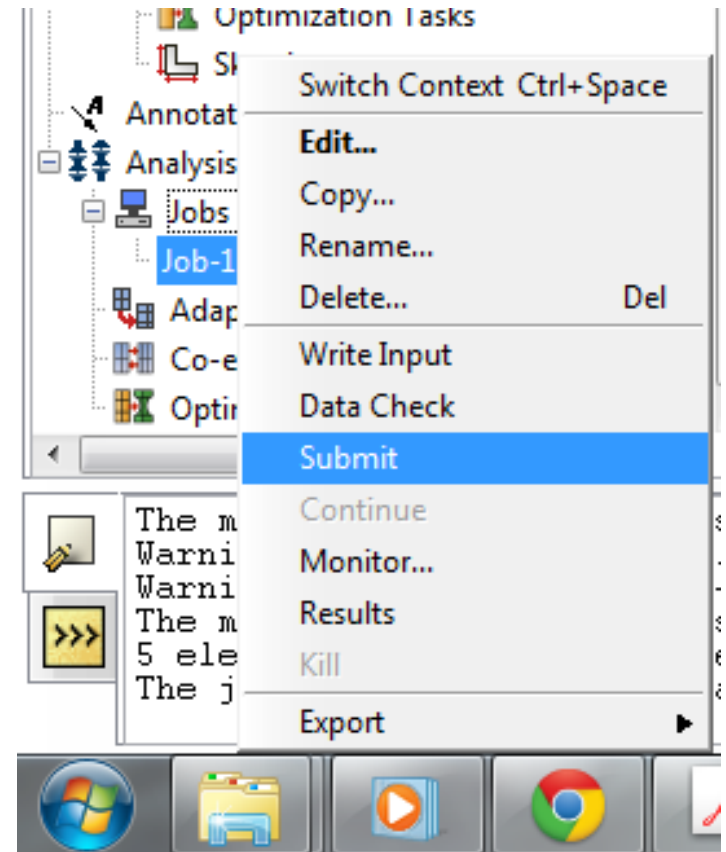
Create a Job

- Under **Analysis**, double click **Jobs** to create a job
- **Create Job** window will pop up
 - Name: **Job-1**
 - Hit **Continue**
 - Hit **Ok**



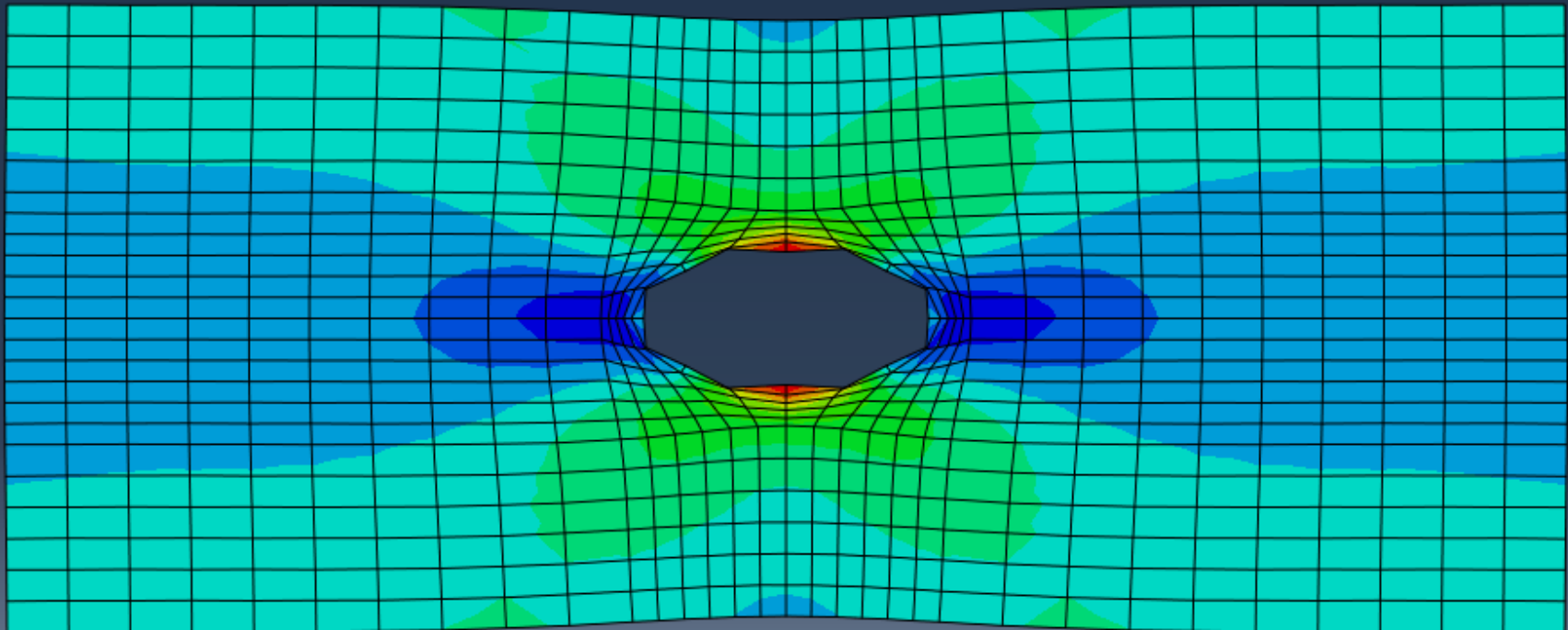
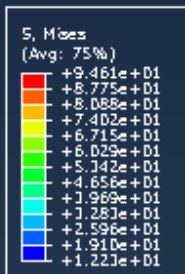
Submit the Job

- In order to conduct the analysis, the job needs to be submit for solving
- Right click **Job-1** under Jobs and click **Submit**
- The following solver status of the job will appear right next to **Job-1** in a parenthesis
 - Submitted
 - Running
 - Completed



Postprocessing

Deformed Shape and Stress Contour

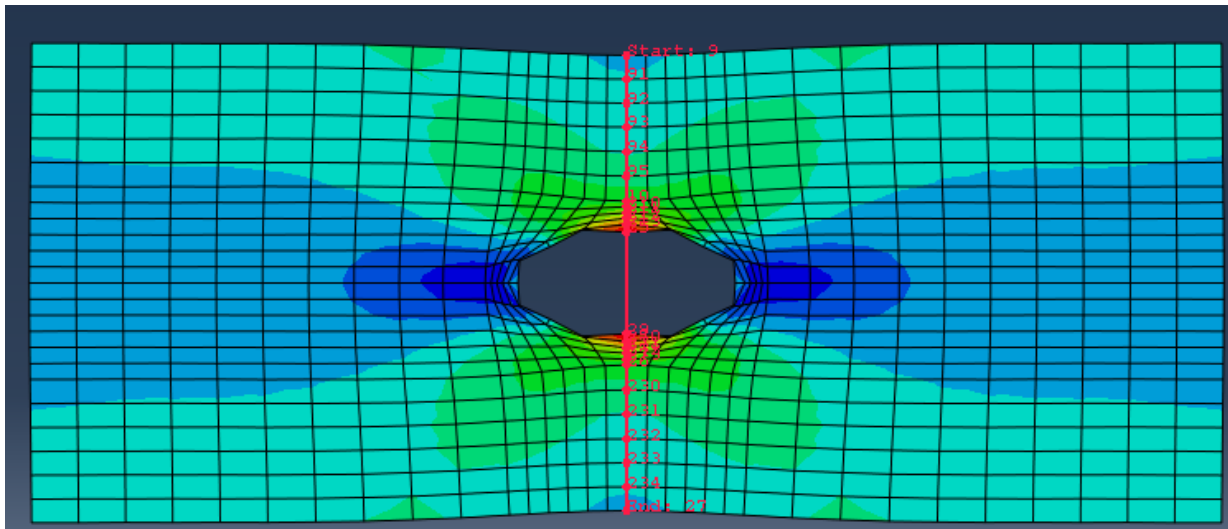
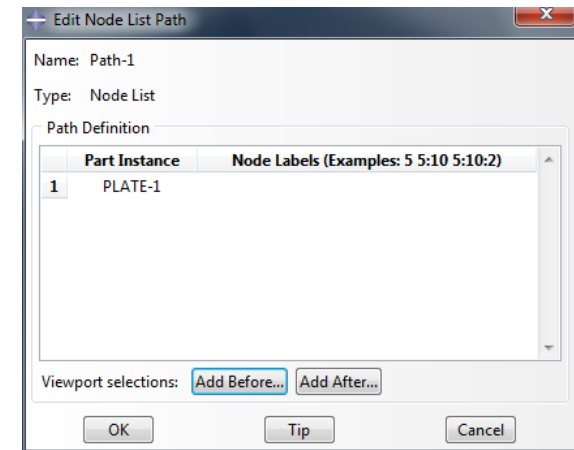
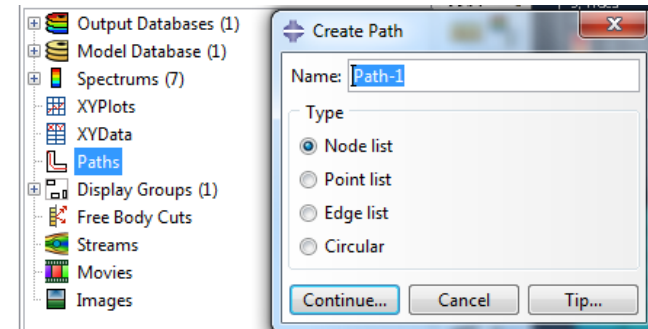


ODB: Job-1.odb Abaqus/Standard Student Edition 6.12-2 Tue Sep 09 23:24:15 Mountain Daylight Time 2014

Step: Step-1
Increment: 1; Step Time = 1.000
Primary Var: S, Mises
Deformed Var: U, Deformation Scale Factor: +1.078e+03

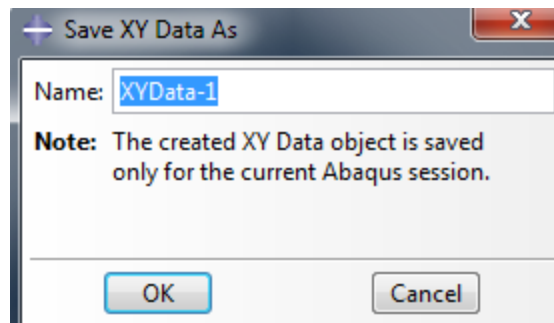
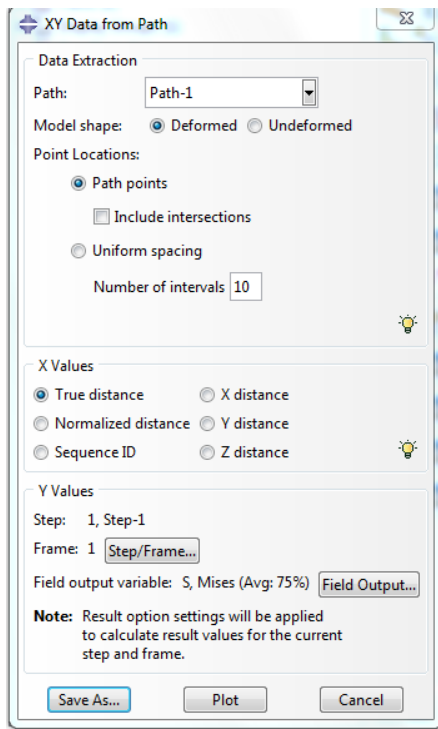
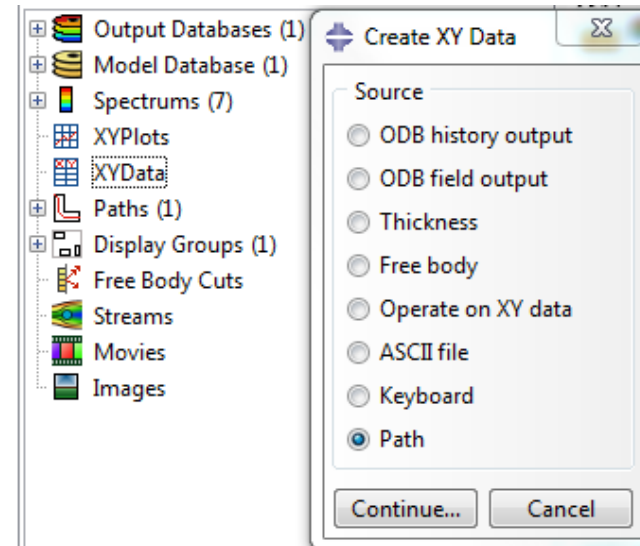
Stress along a line (Create a Path)

- From the Results tree, double click **Paths**
 - **Create Path** window will pop up, hit **Continue**
- **Edit Node List Path** window will pop up, hit **Add Before...**
- Click all the nodes of the vertical centerline from top to bottom (maintain order)
 - Hit **Done** and **Ok**



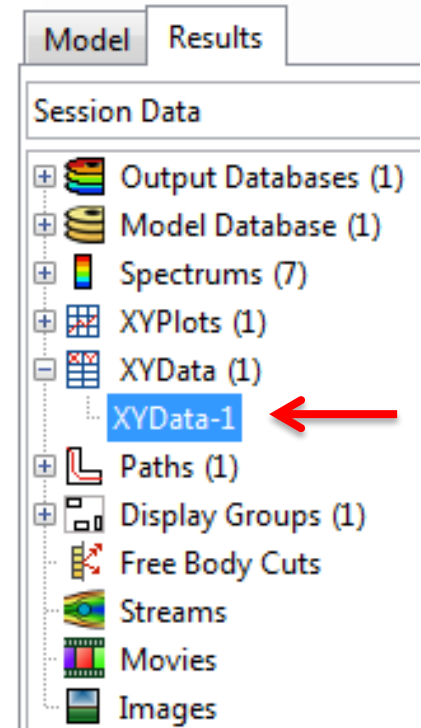
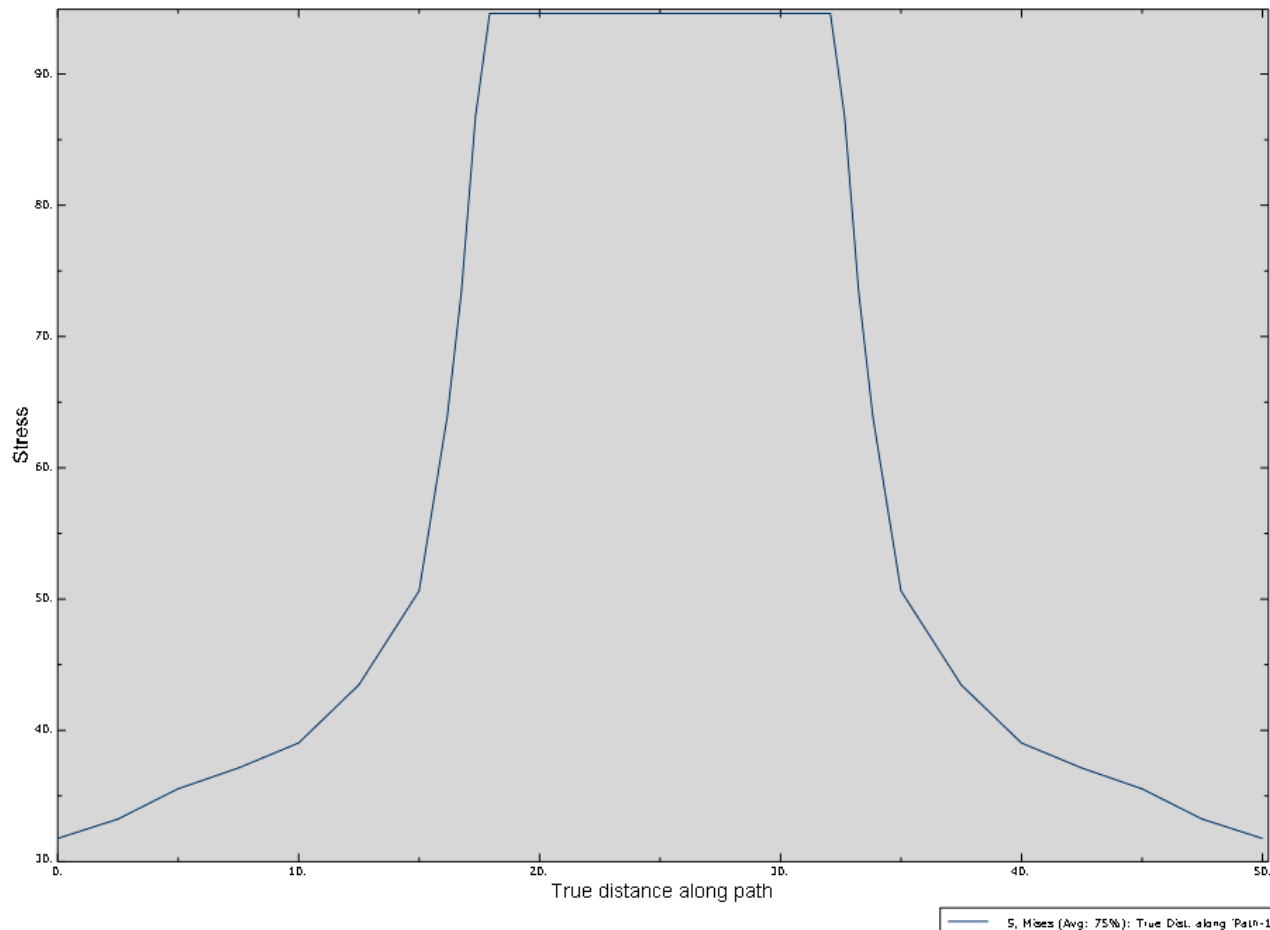
Stress along a line (Create XY Data)

- From the Results tree, double click **XYData**
 - **Create XY Data** window will pop up
 - Select **Path**, and hit **Continue**
- **XY Data from Path** window will pop up, hit **Save As**
- **Save XY Data As** window will pop up
 - Hit **Ok**
- Click 'x' or **Cancel** button to close the window



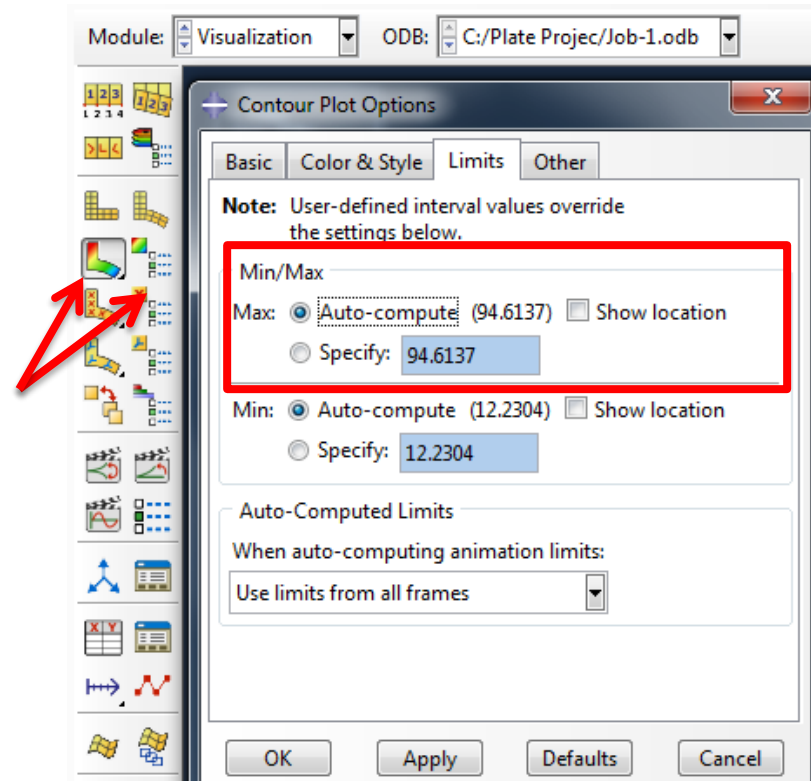
Stress along a line

- Click '+' button of the **XY Data** from Results tree
 - Double click the **XYData-1**
- **File>Print>Destination: File** to save the Stress vs. True distance along path curve



Maximum Stress

- Maximum Stress value is required to obtain to calculate stress concentration factor and factor of safety
- Click the **Deformed Shape** button
- Click the **Contour Options** button
 - Go to **Limit** Tab
 - The Maximum stress value can be obtained from here
 - Check the **Show location** to see the maximum/minimum stress location in the stress contour plot



FEA using Quarter Model

- Follow the same steps to conduct a finite element analysis for the quarter model
- Tips:
 - ✓ Create circle at the bottom left corner of the rectangle and use Auto-trim button
 - ✓ Seed the geometry as following
 - ✓ Apply $-33.33/2 = -16.665$ MPa pressure for Loading

