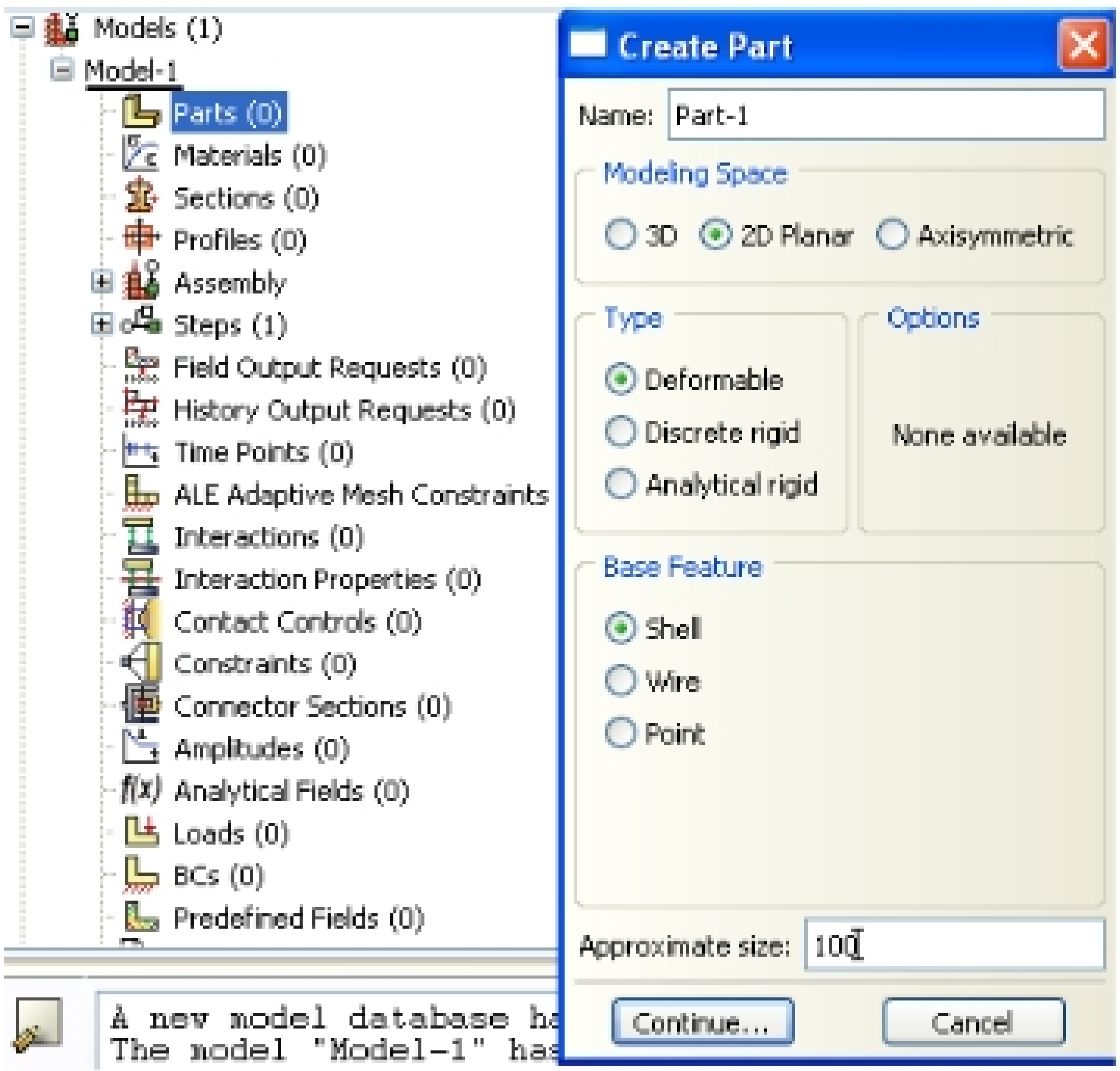
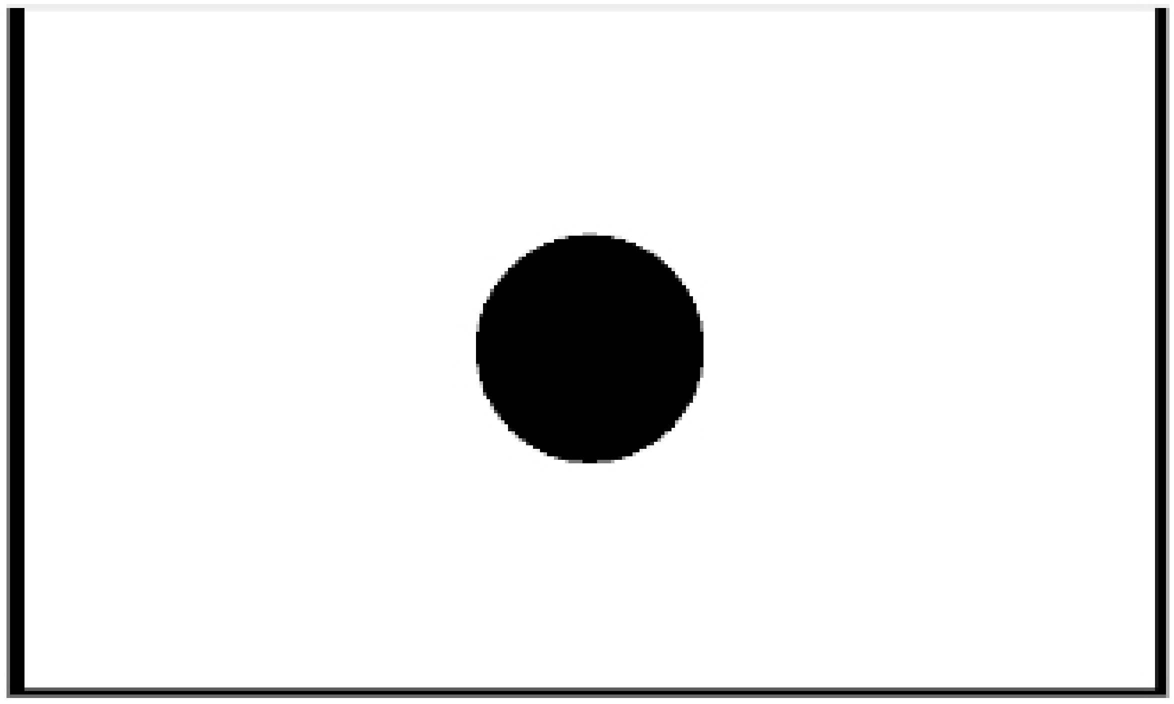
Abaqus Tutorial
Stress for Flat Plate with a Hole

Create part
In the model tree, double click on Parts icon
Create a **2D Planar**, **Deformable**, **Shell**. Set the **Approximate size** based on how big your part will be (I have put 100 here) and name it **Plate_with_hole**



Hit *Continue*
Sketch mode is now active. Use the rectangle tool and sketch a rectangle
Coordinates: (-50,-30) and second one (50,30)
Sketch a Circle in the middle of the rectangle.
center is in the middle of the rectangle and for perimeter take (10,0).
Make sure to choose *Done* to finish the sketch.



NOTE
The sketch mode is very similar to SolidWorks. In order to make a different shape cutout there are several tools available. The tools include points, lines, circle (defined by center and radius, square, ellipse (defined by center and two points on perimeter), arc (tangent to adjacent line), arc (defined by center and two points), arc (defined by 3 points that the arc will pass through), a spline, and a fillet tool for rounding edges. Using these tools as you would in solidworks, sketches can be made to fit almost any object.

Create Material
Create a *material* (double click on material in the module tree).
Choose **Mechanical → Elasticity → Elastic**
Set **Young's Modulus** to 70E9 and **Poisson's Ratio** to 0.3
Hit OK

Create Section
Create a section
In the box that appears Choose **Solid, Homogenous**
Click *Continue*
In the Material Drop down box, choose the Material that you created a minute ago.
Let Plane stress/strain thickness =1
Click *Ok*

Apply Section Assignments
In the Module Tree, expand **Parts**. Expand the part that consists of the plate with the hole. Double click on **Section Assignments**.
The program now asks you to select the regions to be assigned section.
Click on the flat plate and click *Done*.
In the box that appears, choose the Section that was created earlier.
Click *Ok*

Create Instance
Expand the **Assembly** option in the Module Tree.

Create an Instance by double clicking on **Instances**
In the box that appears, choose the part to instance, and whether or not to **mesh dependently** or **independently** (For this example, I am using a 'Dependent' mesh).
Click *Ok*

Create a Step
Create a Step by double clicking **Step** in the Module Tree.
In the box that appears, choose **General** for the procedure type, and **Static, General** in the list. You can also change the name of the step.
Click *Continue*
You can also add a description about the step and Click *Ok*.

Create Boundary Conditions (BCs)
Create a Boundary Condition by double clicking **BC's** in the Module Tree.
In the box that appears, choose the type of boundary condition and which step to apply it to. For this example I am choosing **Mechanical** and then **Symmetry/Antisymmetry/Encastre**
Select the regions for BCs and click done: For this example I am selecting the left edge.
Now, I am using **Encastre**.
Choose *Ok*
For verification, little arrows should have appeared on the edge that is now constrained.

Create a Load
Create a load by double clicking **load** in the Module Tree.
In the box that appears, choose the **category** to be **Mechanical** and the **Type** to be a **Surface Traction**.
Click *Continue*.
You must now select the surface for the load. Click the right edge as the surface.
Make sure to click *Done* after selecting the edge.
In the box that appears choose the **Traction** to be **General.**
A Direction vector must be made. Click the **EDIT** box.
In this example we will make the shell in tension, so choose the first point to be the center dot on the left edge, and then choose the second point to be the center dot on the right edge.
Specify the **Magnitude** to be 100.
Choose *Ok*.
Arrows should have come off the right edge with arrows pointing to the right as shown below: