## Plate With a Hole - Mesh

Author: Benjamin Mullen, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up

2. Geometry

3. Mesh

4. Physics Setup

5. Numerical Solution

6. Numerical Results

7. Verification & Validation

Exercises

Comments

## Mesh

## **Face Sizing**

Now, double click *Model* in the project outline to bring up the Mechanical window.

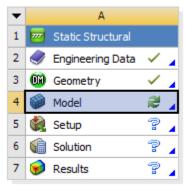
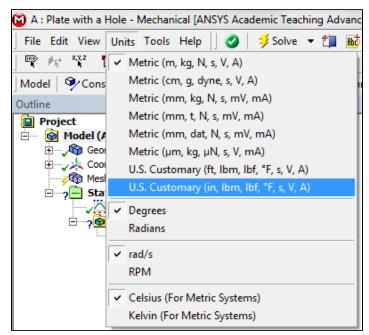


Plate with a Hole

Go to Units > U.S. Customary (in. Ibm, Ibf, F, s, V, A) to make sure the proper units are selected.



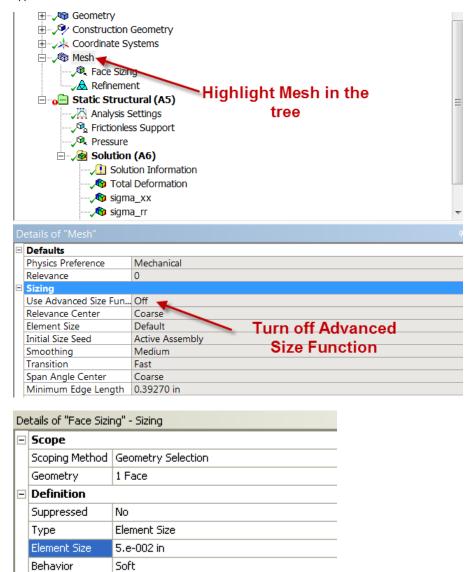
To begin the Mesh process, click Mesh in the outline window. This will bring up the Mesh Menu bar in the Menu bar.



We want to control the size of the elements in the mesh for this problem; to accomplish this, click Mesh Control > Sizing. We now need to pick the

geometry we are going to mesh. Make sure the Face Selection Filter is selected then click the face of the geometry to select it. In the *Details* window click *Geometry > Apply*. Now, we can set some of the details of our mesh. Select *Element Size > Default*, this will allow you to change the size of the elements to be .05 in.

Turn off the Advanced Size Function in the details window of "Mesh". If we leave the Advanced Size Function on, ANSYS will override the face sizing we applied.

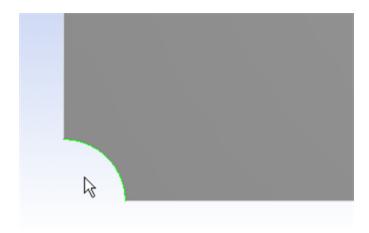


## **Edge Refinement**

Now, we want to refine the mesh by the hole, where we expect a stress concentration. Go to *Mesh Control > Refinement*. This will open the Refinement menu if the details view window. To select the hole as the geometry for refinement, make sure the edge select tool is selected from the menu toolbar.

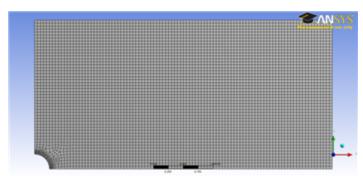


Now, select the hole's edge then click Geometry > Apply.



In the details window, change the Refinement parameter from 1 to 3, this will give us the finest mesh at the hole which will improve accuracy of the simulation.

Now that we have our mesh setup, click *Mesh* > *Generate Mesh*. This will create the mesh to our specifications. Click to display it. It should look something like this:



Click here to enlarge the image

Now that the mesh has been created, we are ready to specify the boundary conditions of the problem.

Go to Step 4: Physics Setup

Go to all ANSYS Learning Modules