

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](#)

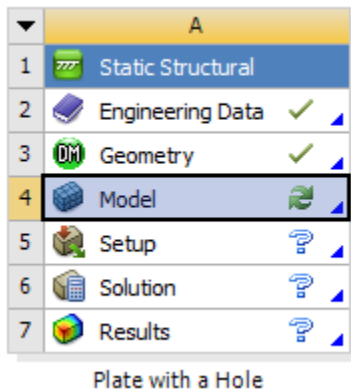
[Exercises](#)

[Comments](#)

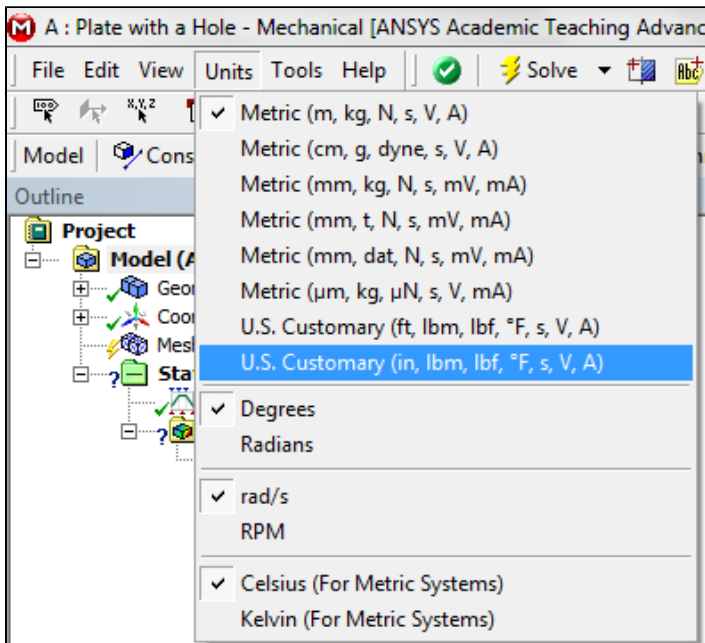
Mesh

Face Sizing

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




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

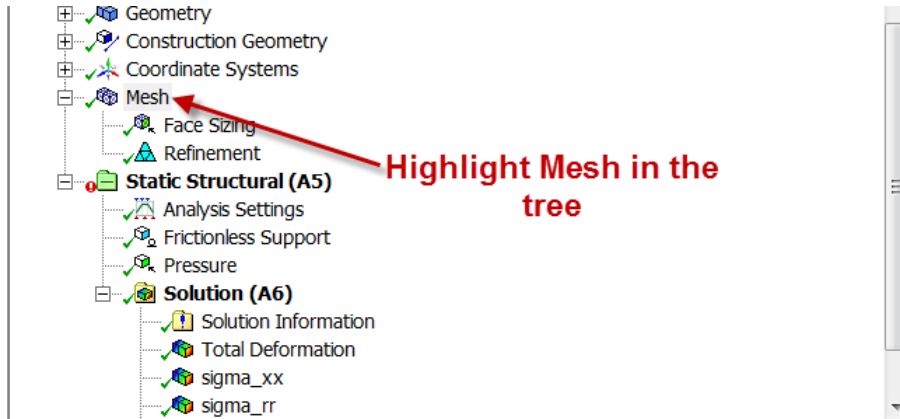


To begin the Mesh process, click **Mesh** in the outline window. This will bring up the Mesh Menu bar in the Menu bar.



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 element. Choose 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.



Details of "Mesh"

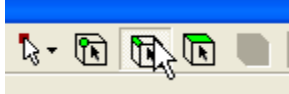
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Fun...	Off
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	0.39270 in

Details of "Face Sizing" - Sizing

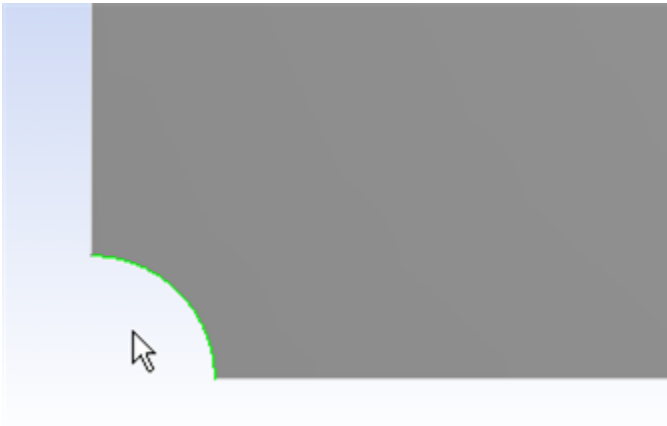
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Type	Element Size
Element Size	5.e-002 in
Behavior	Soft

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.

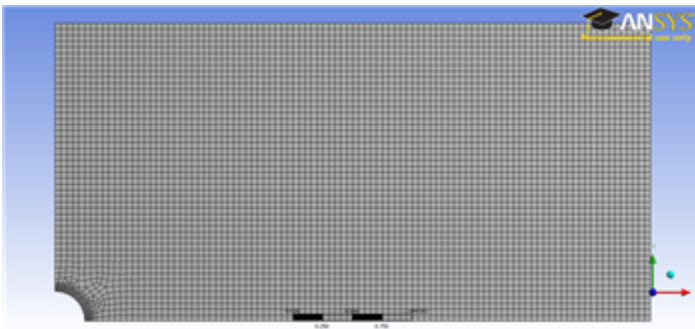


Now, select the hole's edge then click **Geometry > Apply**.



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  **Mesh** 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](#)