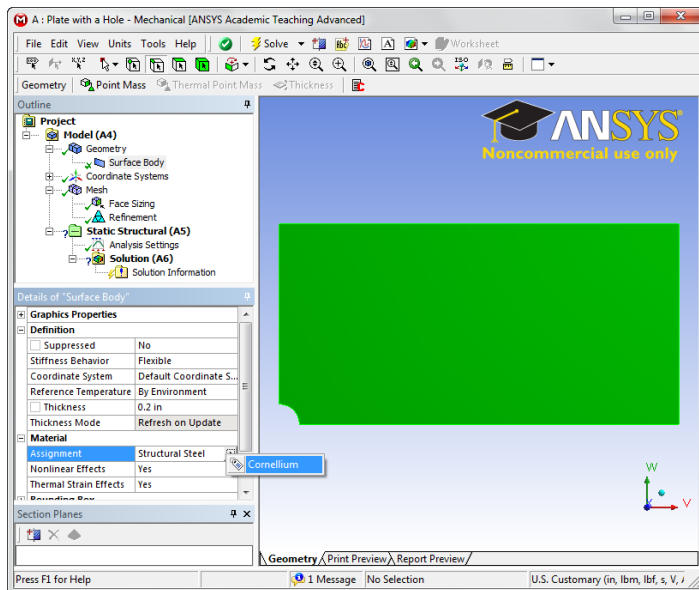# Plate With a Hole - Physics Setup

Author: Benjamin Mullen, Cornell University

Physics Setup

Specify Material

First, we will tell ANSYS which material we are using for the simulation. Expand **Geometry**, and click **Surface Body** in the *Outline* window. In the *Details* window, select **Material > Assignment > Cornellium**. The material has now been specified.
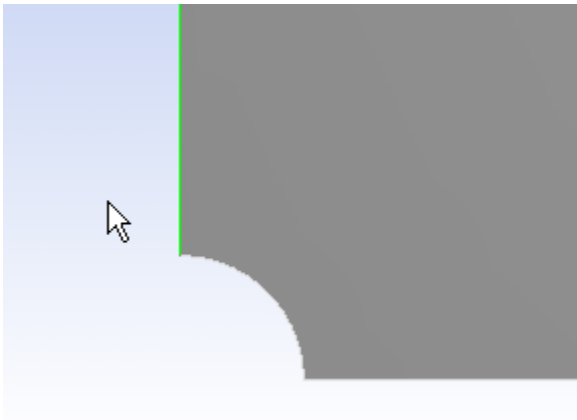

Click here to enlarge

Symmetry Conditions

First, let's start by declaring the symmetry conditions in the problem. Right click **Model > Insert > Symmetry**.
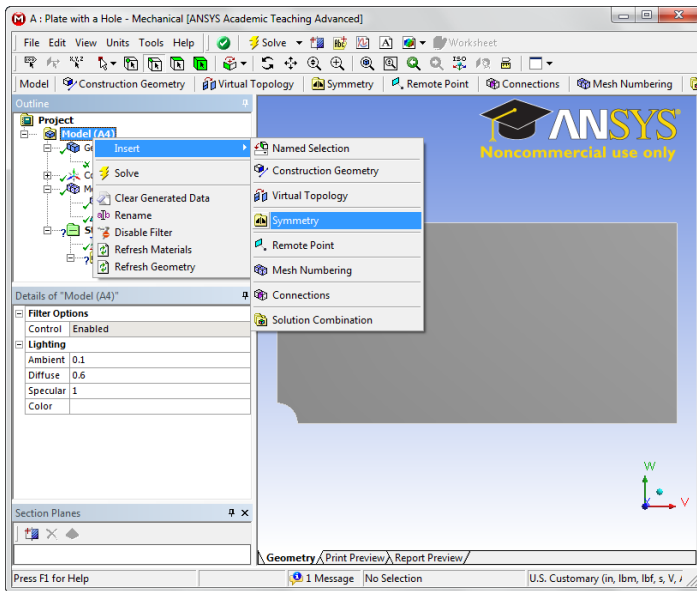
This will create a symmetry folder in the outline tree Now, right click **Symmetry > Insert > Symmetry Region**. Make sure the Edge Select Tool is highlighted and select the left edge above the hole.


Now look to the Details View window and select **Geometry > Apply**. This should create a red line with a tag on the model. Ensure that under **Symmetry Normal** the x-axis is selected. Repeat this process to create a symmetry region for the bottom edge to the right of the hole, but this time making sure that the **Symmetry Normal** parameter is the y-axis.

Forces

Now, we can specify the forces on the body. Click **Static Structural** in the outline tree. This will bring up the Physics Sub-Menu bar in the Menu Bar.
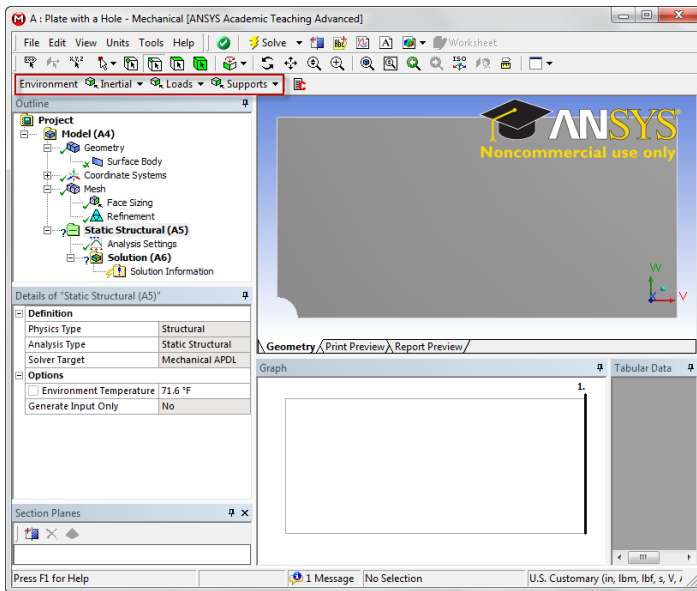
Click **_Loads > Pressure_** to specify a traction. Select the right edge of the geometry and apply it in the details view window. The pressure's magnitude from the problem specification is `-1e6 psi` (pressure in ANSYS defaults to compression, and we need tension, hence the negative sign). Now that the forces have been set, we need to set up the solution before we solve.

Go to Step 5: Numerical Solution