

AIM Plate with Hole - Geometry

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results
6. Verification & Validation

Geometry

Since the problem given to us represents a 2D state of stress, there are several simplifications that must be made due to the fact that AIM model will be 3D. Since the geometry is symmetric, when we draw the shape we want to create a one-quarter model. By taking advantage of symmetry, we can simplify the problem and also ensure that the model is not over constrained.

Draw Geometry

In order to use the units given to us in the problem, press the **Project** button in the top left corner and select **Units > U.S. Engineering**. Click on the **Z-axis** on the compass in the bottom left corner of the screen to look at only the XY-plane. Right click in the empty white space and choose **Select New Sketch Plane**, then click on the grid that appears, so that the plane we are sketching on will be on the XY-plane. Choose the rectangle drawing tool in the **Sketch** subgroup of the **Design** tab. Select the origin, then drag up and to the right until the rectangle is the correct size, or enter the dimensions directly into the highlighted boxes as pictured below. The height should be 5 inches while the length is 10 inches.

After the rectangle has been drawn, make a circle at the origin with a radius of 0.25 inches.

Using the **Trim Away** tool in the **Sketch** subgroup of the **Design** tab, remove the outside of the circle and the ends of the box to look like the picture below.

Rotate the model slightly out of the XY-plane, then extrude the face using the **Pull** tool in the **Edit** section of the toolbar. The problem specifies that the thickness of the plate must be 0.2 inches.

Go to Step 3: Mesh

Go to all ANSYS AIM Learning Modules