

# Plate With a Hole - Geometry

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

## Geometry






For users of ANSYS 15.0, please check [this link](#) for procedures for turning on the Auto Constraint feature before creating sketches in DesignModeler.

### Analysis Type

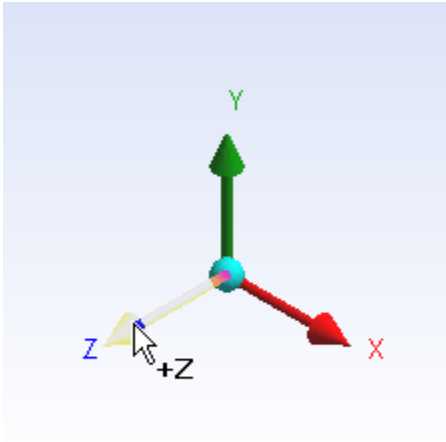
First, right click  Geometry  and click **Properties** to bring up the geometry properties menu. The default analysis type of ANSYS is 3D, but we are doing a 2 dimensional problem. Change the **Analysis Type** from **3D** to **2D**.

	A	B
1	Property	Value
2	General	
3	Component ID	Geometry
4	Directory Name	SYS
5	Geometry Source	
6	Geometry File Name	
7	Basic Geometry Options	
8	Solid Bodies	<input checked="" type="checkbox"/>
9	Surface Bodies	<input checked="" type="checkbox"/>
10	Line Bodies	<input type="checkbox"/>
11	Parameters	<input checked="" type="checkbox"/>
12	Parameter Key	DS
13	Attributes	<input type="checkbox"/>
14	Named Selections	<input type="checkbox"/>
15	Material Properties	<input type="checkbox"/>
16	Advanced Geometry Options	
17	Analysis Type	2D
18	Use Associativity	<input checked="" type="checkbox"/>
19	Import Coordinate Systems	<input type="checkbox"/>
20	Import Work Points	<input type="checkbox"/>
21	Reader Mode Saves Updated File	<input type="checkbox"/>
22	Import Using Instances	<input checked="" type="checkbox"/>
23	Smart CAD Update	<input type="checkbox"/>
24	Enclosure and Symmetry Processing	<input checked="" type="checkbox"/>
25	Mixed Import Resolution	None

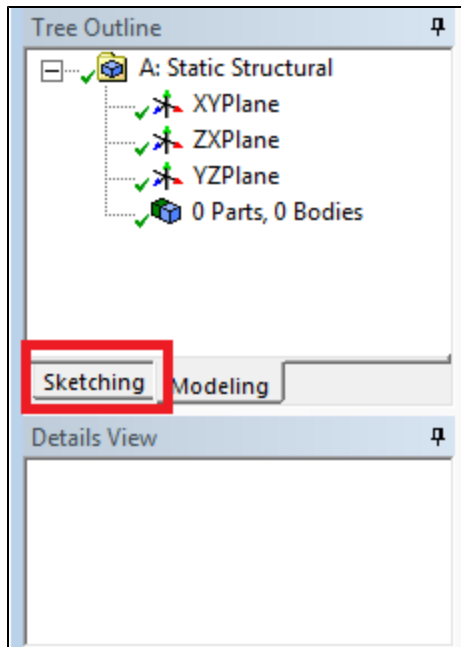
Next double click **3**  **Geometry**  . This will bring up the Design Modeler. It will prompt you to pick the standard units. Since all of the units in the problem specification were given in English units, we want to choose *inch*. When the *inch* radio button is selected, press **OK**

## Draw the Geometry


To begin sketching, we need to look at a plane to sketch on. Click on the Z-axis of the compass in the bottom right hand corner of the screen to look at the x-y plane.

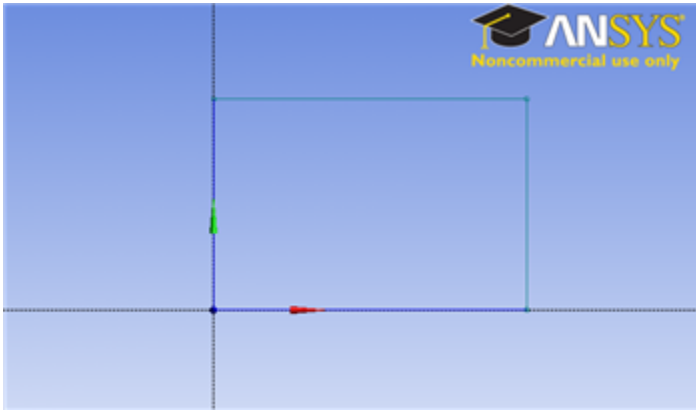



Now, look to the sketching toolboxes window and click the sketching tab; this will bring up the sketching menu.

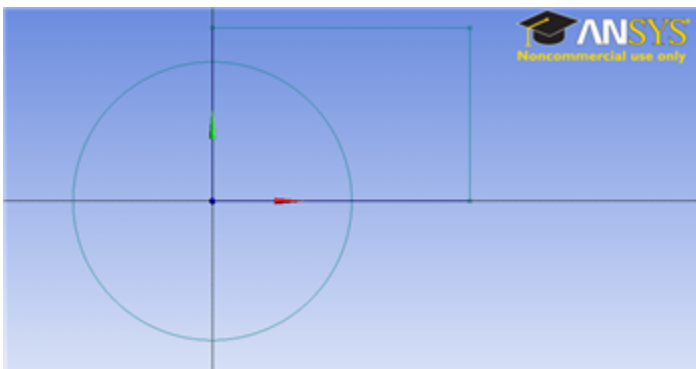


Before we sketch the geometry, let's note something about the problem specification. The geometry itself has two planes of symmetry: it is symmetric about the x-plane and y-plane. This means we can model 1/4 of the geometry, and use symmetry constraints to represent the full geometry in ANSYS. If we model a quarter of the geometry, we can make the problem less complex and save some computational time.

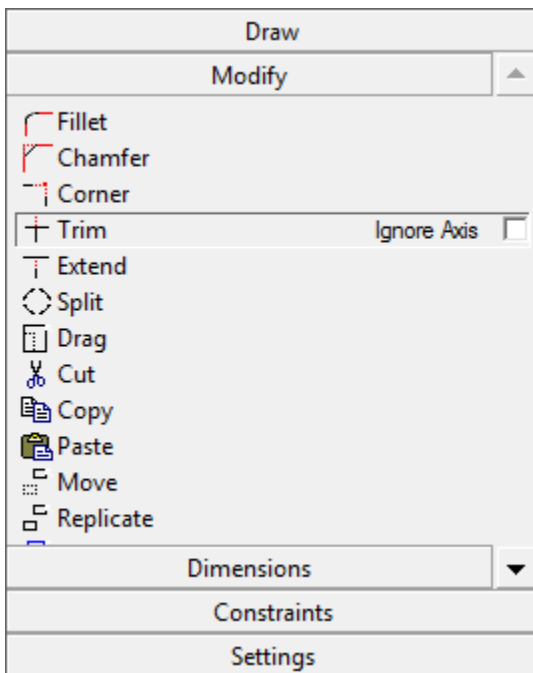
Okay! Let's start sketching. First, click  **Rectangle** in the sketching tool bar. This tool defines a rectangle by two points. Place the first point at the origin (Watch for the P- symbol which shows you are placing the point at the origin point), and the other point somewhere in the first quadrant.



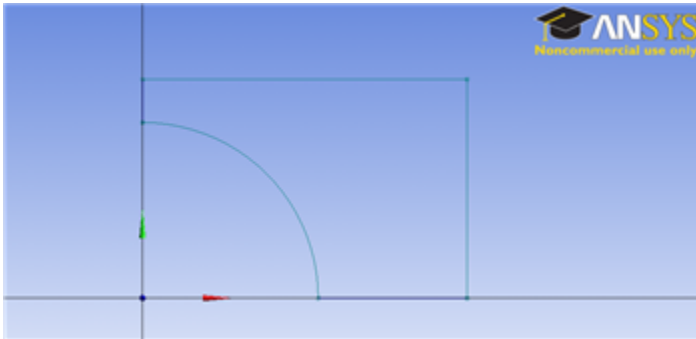
Now, click  **Circle**. This tool allows to define a circle by clicking once to define its center point, then click a distance away from the center point to define a radius. Define the circle so its center point is at the origin, define the radius by clicking somewhere inside the rectangle.



We almost have a geometry, but we first need to get rid of the superfluous lines. In the sketching toolboxes window, click **Modify > Trim**.

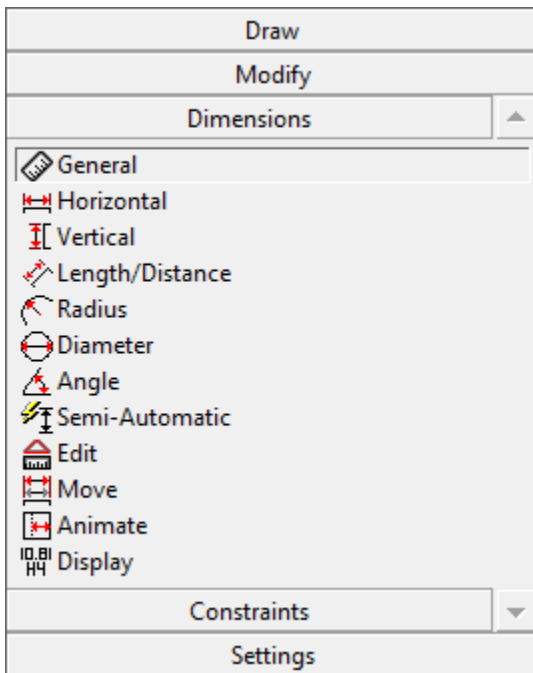


Now, trim the segments that are 1. outside of the 1st quadrant, and 2. between the circle and the origin. You should end up with something similar to the following figure.

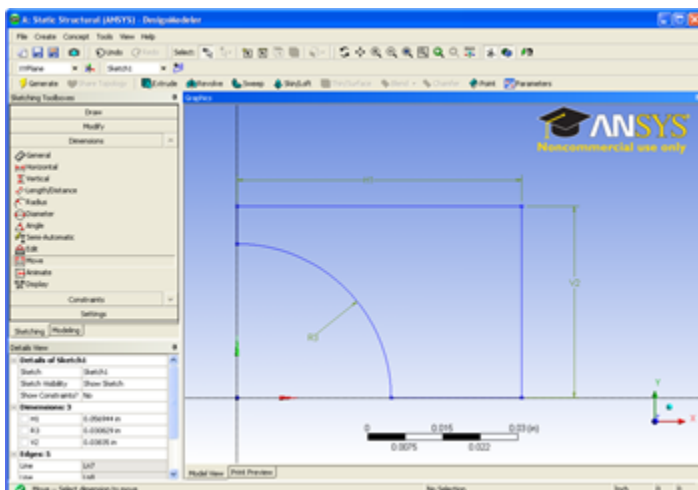


## Dimensions

Now, we have to dimension the drawing to the problem specification. (Remember! We are only drawing 1/4 of the geometry, so we need to take this into account when dimensioning the figure in ANSYS). In the sketching toolboxes window, click **Dimensions > General**



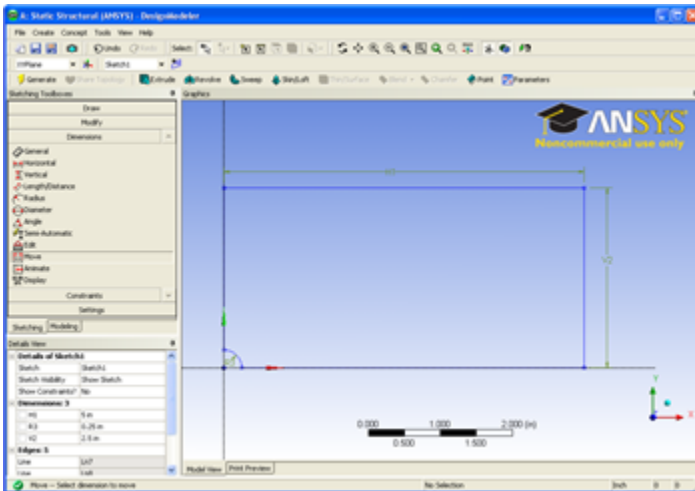
This tool will allow you to define dimensions that you can specify. We need to specify the rectangle's length and width, and the circle's radius. Use the tool to define the height of the rectangle (the right edge of the geometry), the length of the rectangle (the top edge of the rectangle), and the radius of the circle. You should end up with the following window.



To specify the dimensions, look to the Details View Window.

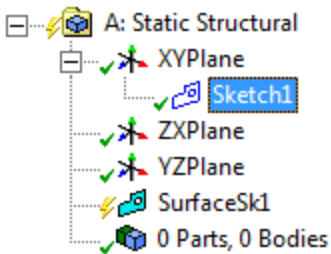
[-] <b>Details of Sketch1</b>	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
[-] <b>Dimensions: 3</b>	
<input type="checkbox"/> H1	5 in
<input type="checkbox"/> R3	0.25 in
<input type="checkbox"/> V2	2.5 in
[-] <b>Edges: 5</b>	
Line	Ln7
Line	Ln8
Line	Ln9
Line	Ln10
Circular Arc	Cr11

Change the H (for horizontal) dimension to 5 inches, the V (for vertical) dimension to 2.5 inches, and the R (for radius) dimension to .25 inches. Now we have the geometry specified in the problem statement sketched in ANSYS.



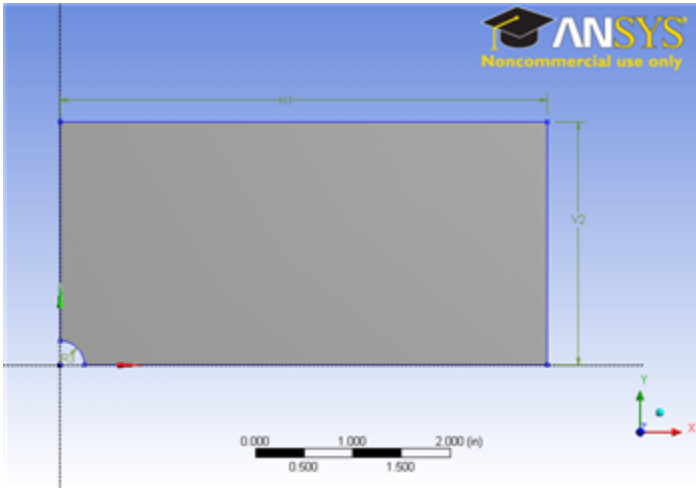
## Create a Surface from the Sketch

Next, we need to tell ANSYS what type of geometry we are modeling. For this problem, we will create a surface and give it a thickness. In the menu bar, select **Concept > Surfaces from Sketches**. To select the sketch, look to the outline window, and expand **XY plane > Sketch 1**.



In the details window pane, select **Base Objects > Apply**. Now, we need to specify a thickness. Specify the thickness as .1 inches, as from the problem

statement. Now in the menu toolbar, click  **Generate**. This should generate the geometry.



Close the Design Modeler (don't worry, the geometry will be saved in the project automatically). Now we are ready to mesh the geometry.

[Go to Step 3: Mesh](#)

[Go to all ANSYS Learning Modules](#)