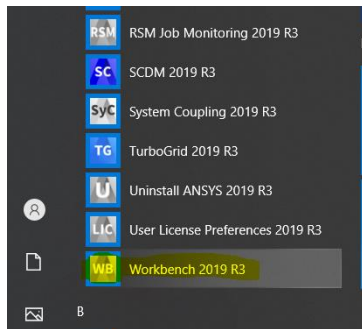
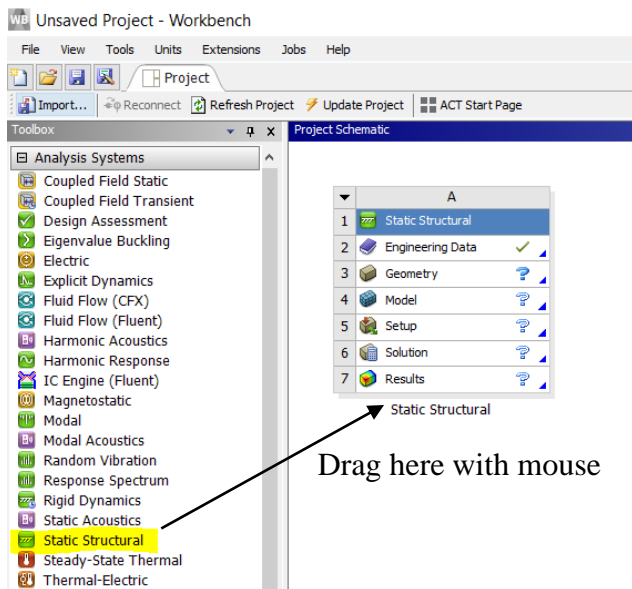


Hyperelastic Seal: Starting ANSYS Workbench

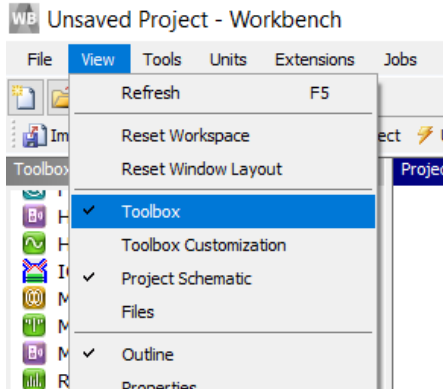
Make sure that ANSYS has been installed on computer. Go to your Windows Apps Start group and find the ANSYS app. Open it and scroll to the bottom and select Workbench at the bottom of the list. If you recently installed the latest version then you will have the latest version listed.



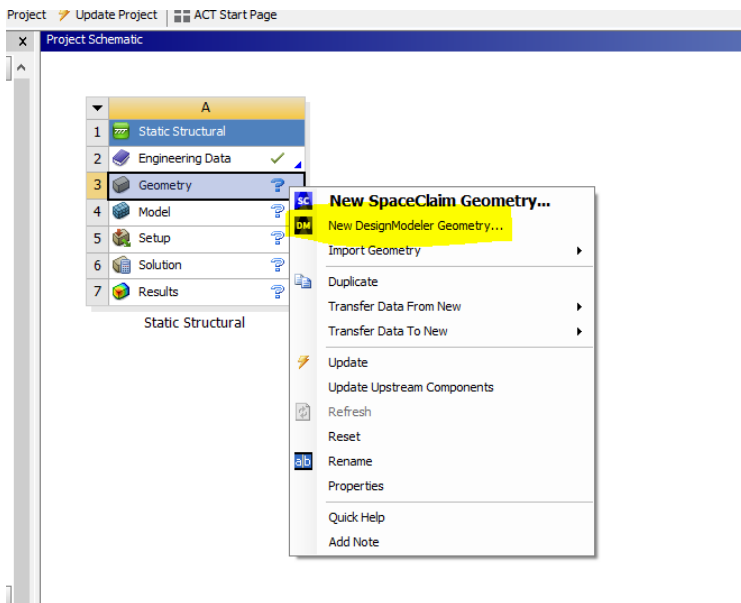
Workbench will open and on the left you should see the toolbox as shown in the figure below. Select “Static Structural” from the list, and holding the left mouse button drag the analysis to the right into the project schematic as shown below. Release the mouse button and you will see the project menu list.



NOTE: If you do not see the list on the left, go to the menu bar at top and select View. Make sure that Toolbox is checked. If you still don't see the list, then there was a problem with your installation and the license setup. You will have to run the installation process again.



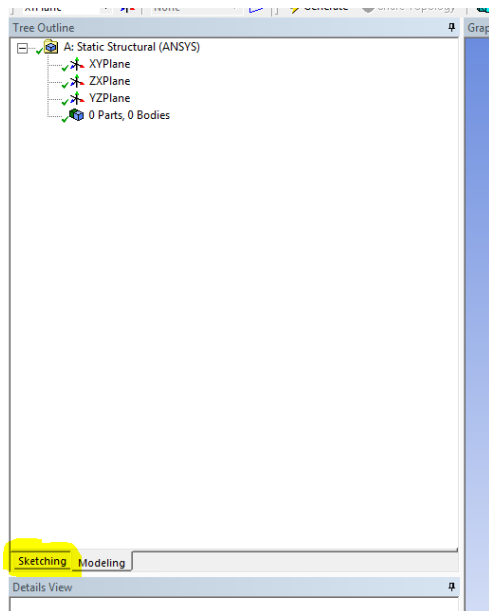
Right click on Geometry from the Static Structural menu list. Select “New DesignModeler Geometry”. Note that by default if you double click Geometry will open new SpaceClaim Geometry. We want to use DesignModeler Geometry for this exercise. When Design Modeler opens proceed with the geometry creation as outlined below.



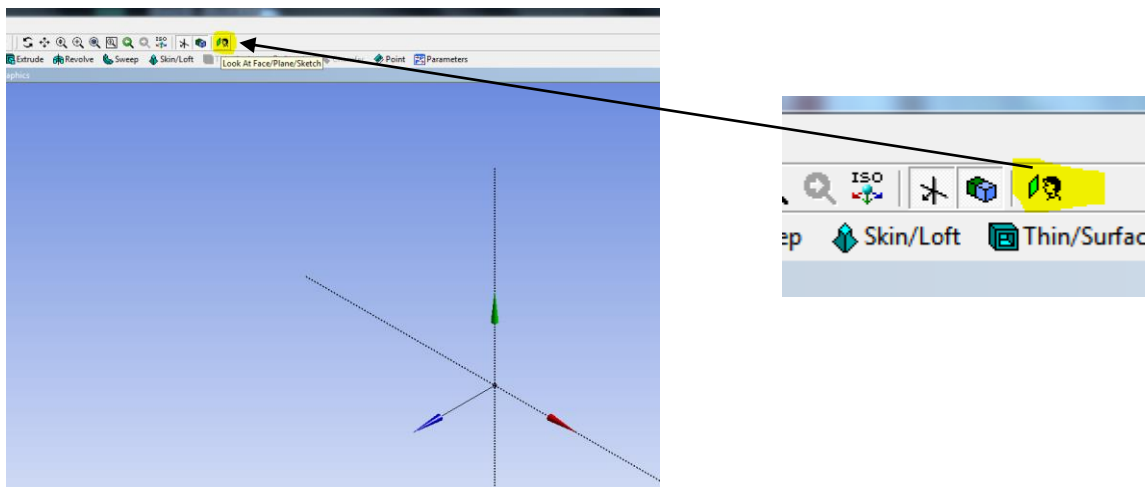
Hyperelastic Seal Tutorial: Geometry Creation

This tutorial will outline the steps for creating the seal in workbench. When complete go back to the hyperelastic seal tutorial that was provided to set up and solve the analysis.

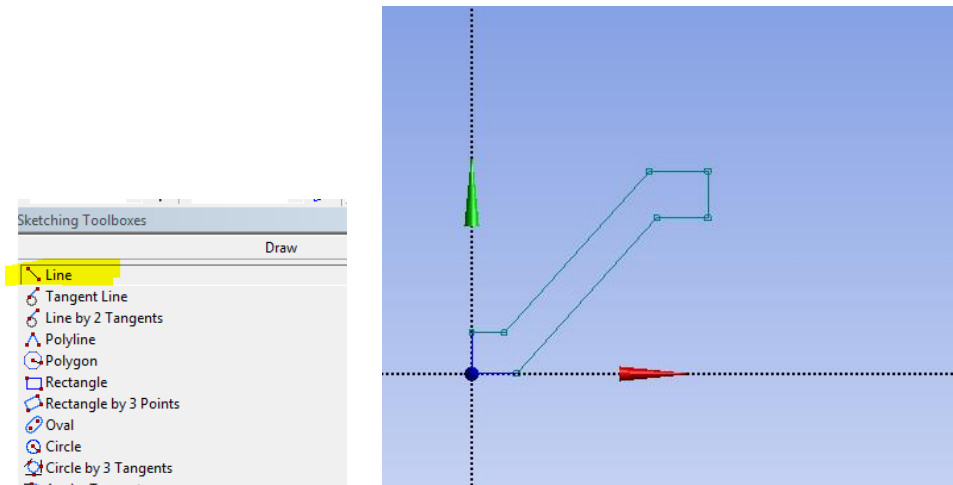
1. Set the units to Inch.
2. Select the Sketching tab in the lower left of the Tree Outline



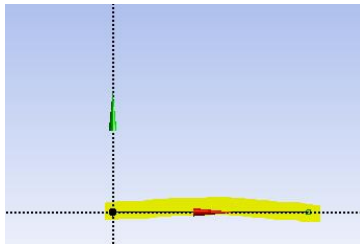
3. Change the plane view from isometric to 2D. Select the quick access button highlighted in the figure below. Different plane views can also be selected from the tree outline by selecting the desired plane from the list (see figure above).



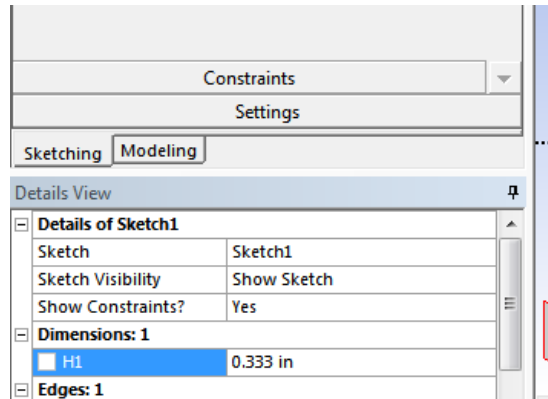
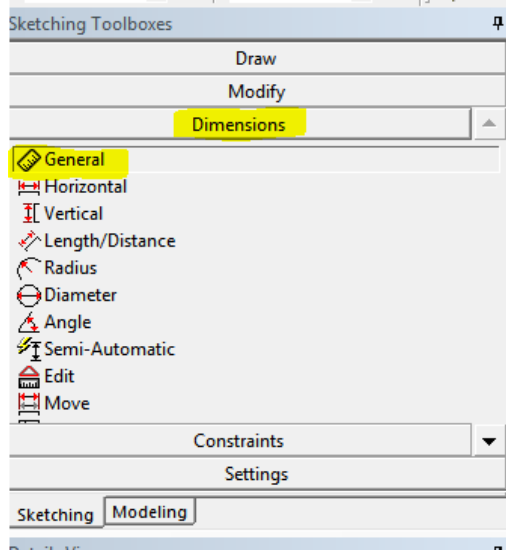
4. Sketch the basic geometry. Select Line from the draw tab. You can start by either drawing the full geometry as shown below or by drawing and dimensioning each line as you go. However, drawing the full geometry before dimensioning can result in the geometry distorting when the dimensions are updated. Once the geometry becomes highly distorted it may not be possible to recover and revert back to the original geometry. Therefore, it is better to set initial dimensions on at least 2 lines and then complete the geometry, as is described below.



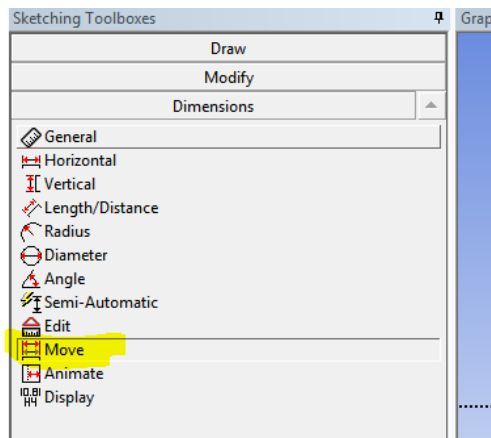
Begin by drawing a horizontal line from the origin. Select “Line” from the options in the Draw tab. Place the cursor at the origin and drag the line. Click the left mouse button to position the line.



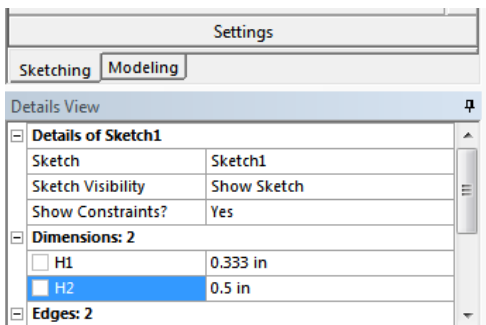
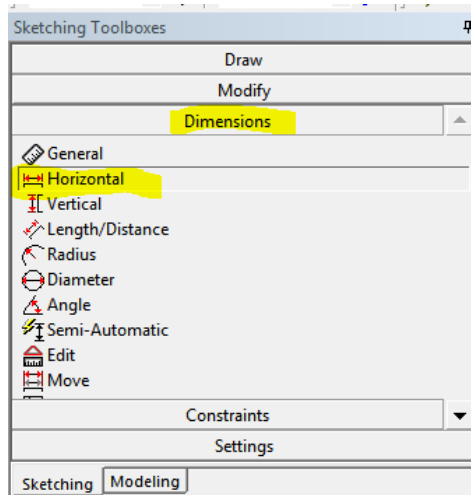
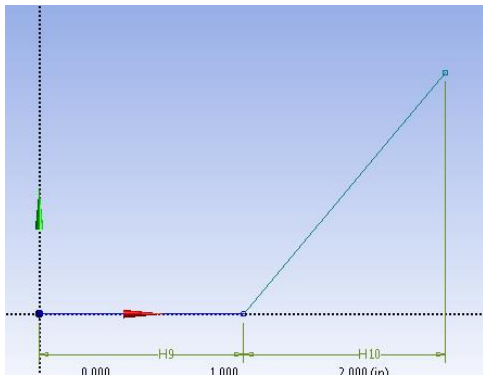
Select the Dimensions tab and then select General dimension. Select the line that was just drawn. Move the cursor to position the dimension on the drawing. The newly created dimension will appear in the details view. Click on the dimension and change it to 0.333. The dimension can be changed at any time by selecting it in the details view. The dimension can also be deleted by selecting the dimension in the details view and deleting it.



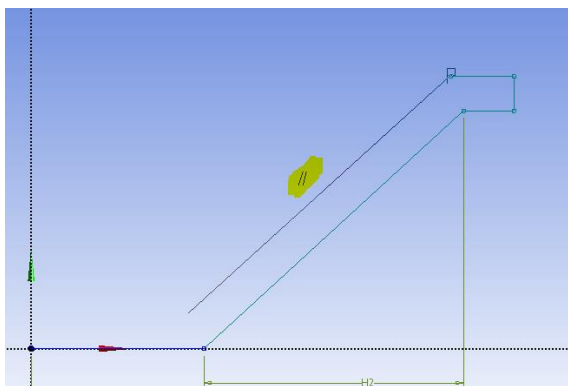
Tip: The position of the dimension can be moved by selecting “Move” from the list. Select Move then select the dimension in the drawing window and use the mouse to drag the dimension to the desired location.



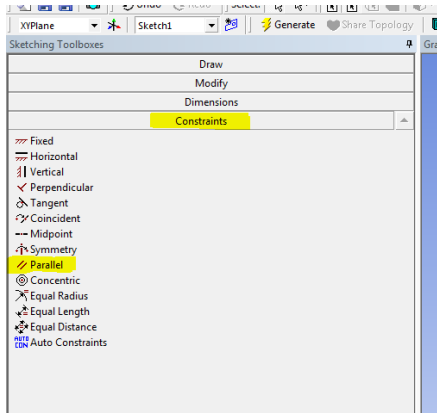
Draw the sloped line. Select “Horizontal” from the dimension tab to set the dimension. Holding the control key select the two endpoints of the line. Move the cursor to position the dimension. In the details view set the dimension to 0.5. Note that your dimension number (ie. H2 or H3 etc) may differ from what is shown here. That is fine.



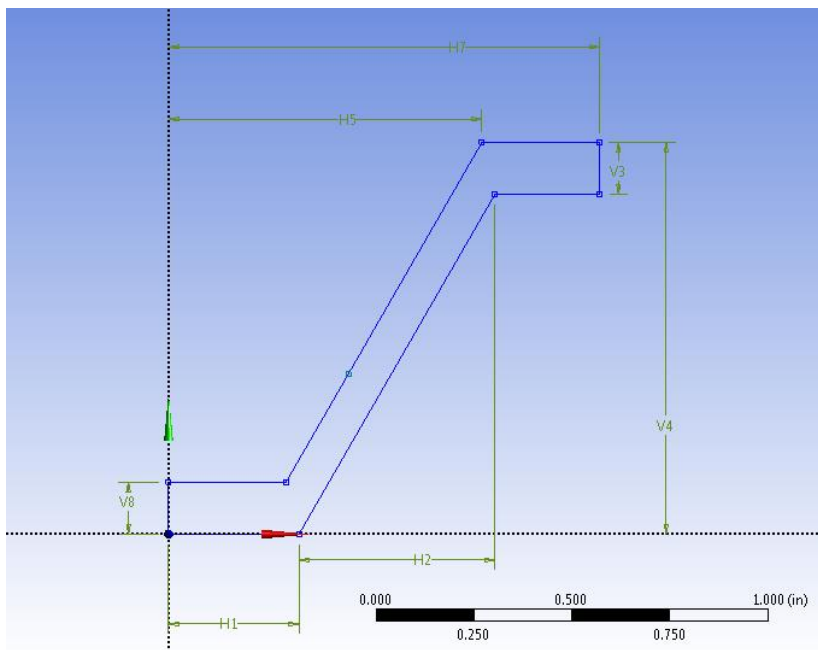
With the dimensions set on the 2 lines the rest of the geometry can be drawn. The remaining dimensions will be set after the entire geometry has been sketched. Note that as you move the cursor to sketch the lines constraint marker will appear. An “H” will appear to make the line horizontal and a “V” for vertical constraint. The constraint will force the line to be either horizontal or vertical even if the lines are not sketched horizontal or vertical. Make sure that the two sloped lines have a parallel constraint. The parallel marker will appear while sketching the line.



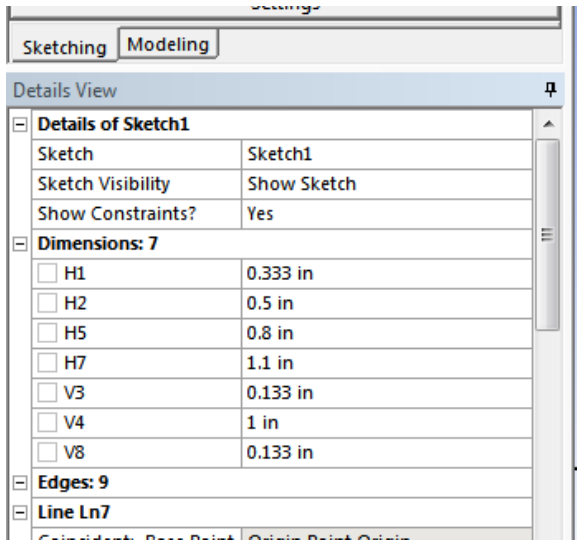
Tip: Constraints can also be set in the constraints tab after the geometry has been sketched.



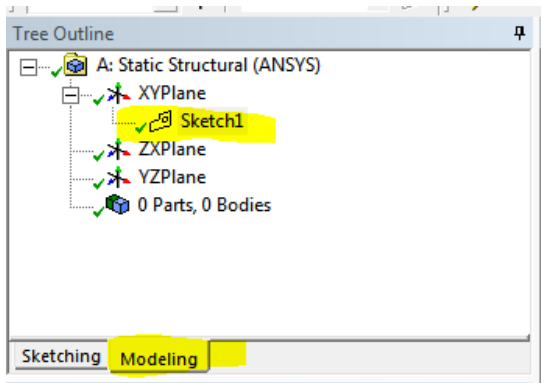
Be sure to close off the geometry (no open sections with missing lines). Set the remaining dimensions as described above. To set the dimension to the axis, select either vertical or horizontal as appropriate from the list. Select the point you wish to dimension. Hold the control key and select the x for a vertical dimension or y axis for horizontal dimension. Position the dimension in the drawing.



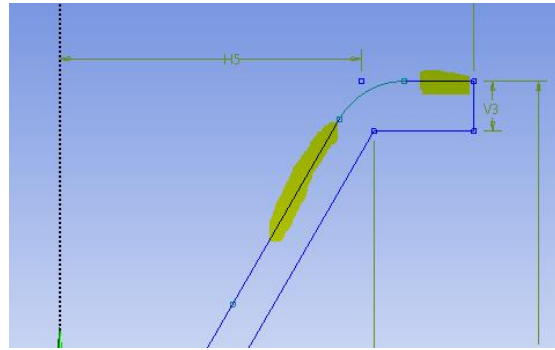
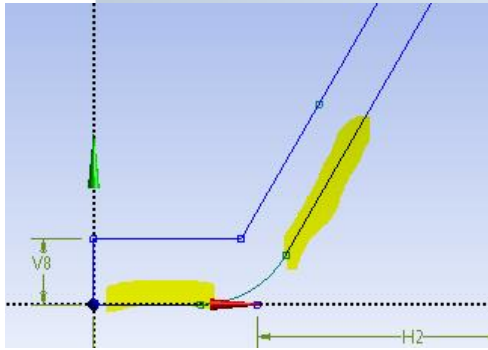
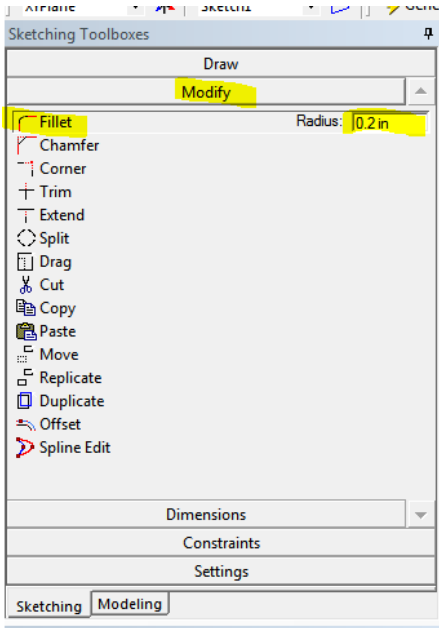
Set the dimensions. Note that your dimension labels (H3, V4, etc.) may differ. That is OK. Refer to the drawing for the dimensions and set the appropriate value in the details view. If after dimensioning you see that any there are open edges simply draw a line to connect the open endpoints. The geometry must be enclosed (no open edges)



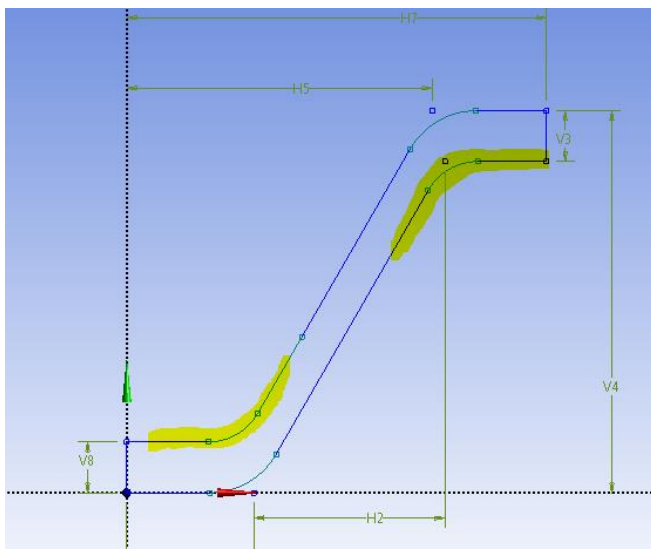
Tip: If your drawing becomes completely distorted when change the dimension, try starting over by deleting (or use undo option) the sketch.. It may take several tries and some practice until you get a feel for the scale/dimension of your sketch. To clear and start over go back to the modeling tab. Expand the XY plane to show Sketch1 (if not already expanded). Select it from the tree and delete it.



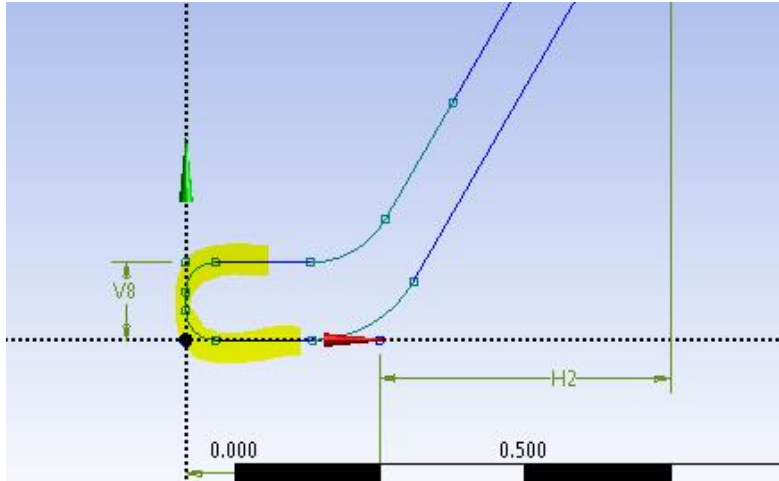
5. Add fillets and set the radii. Select the Modify tab and select Fillet. The outside radius will be set to 0.2. Select the 2 highlighted lines and the fillet will be created. Repeat for the top line.



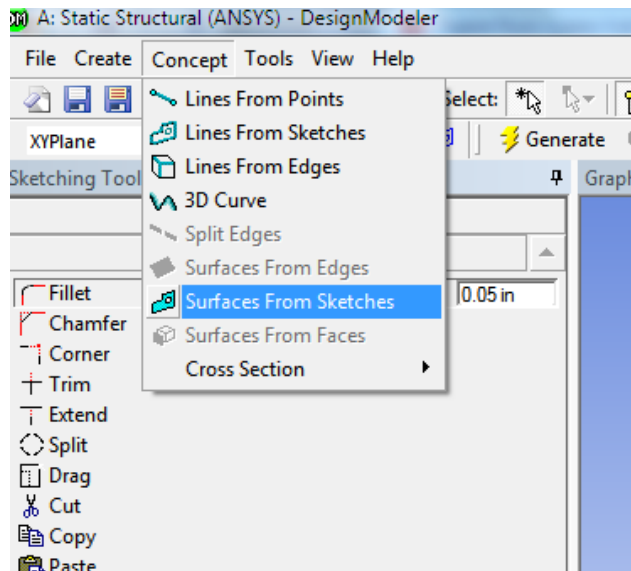
Change the fillet radius to 0.15 to set the inside radius. Make sure to press enter after modifying the value to lock in the change. Select the lines to create the fillet.



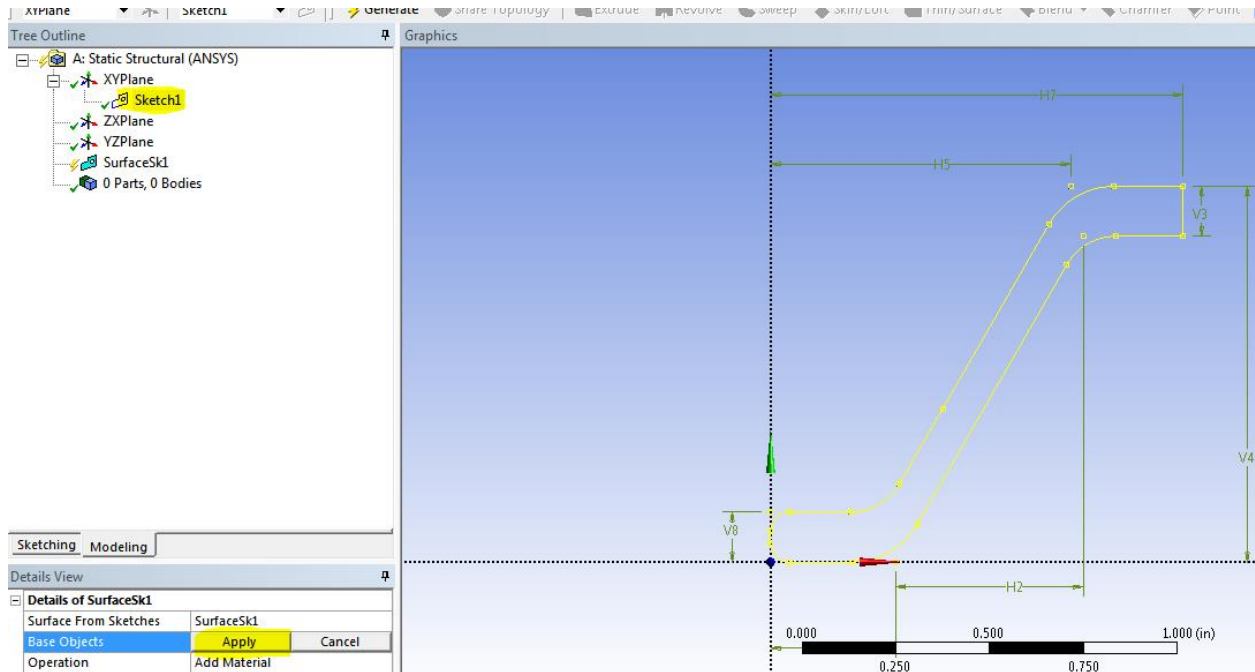
Change the radius to 0.05 and add the radius the end



6. Create Surfaces from Sketches. Select Concept from the menu and “Surfaces from Sketches”.



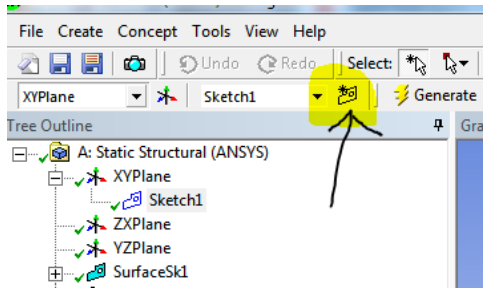
Select Sketch 1 from the tree. The sketch in the drawing window will highlight and change to yellow. Select Apply in the details view.



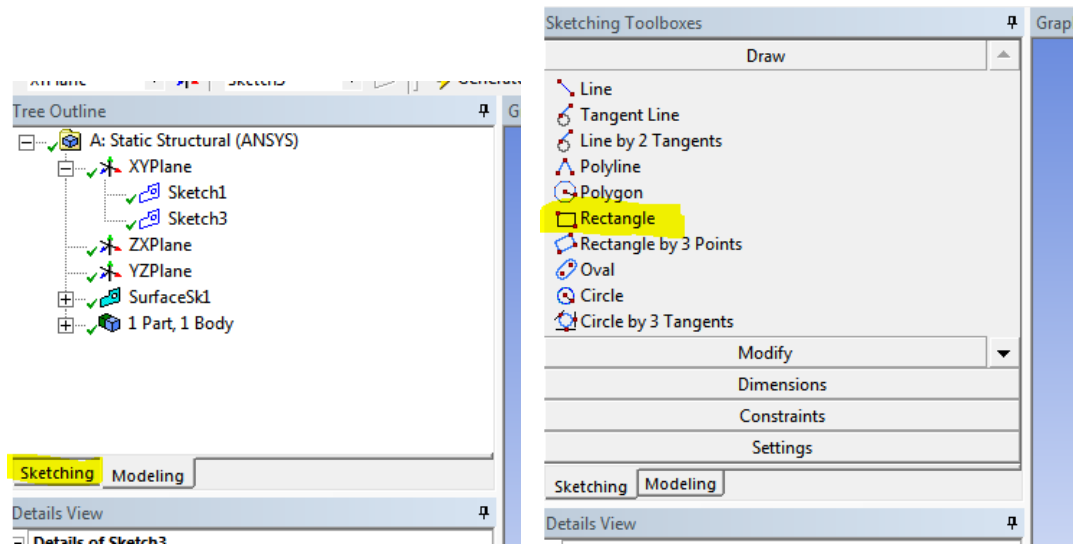
Generate. The sketch will become gray when generation is complete.



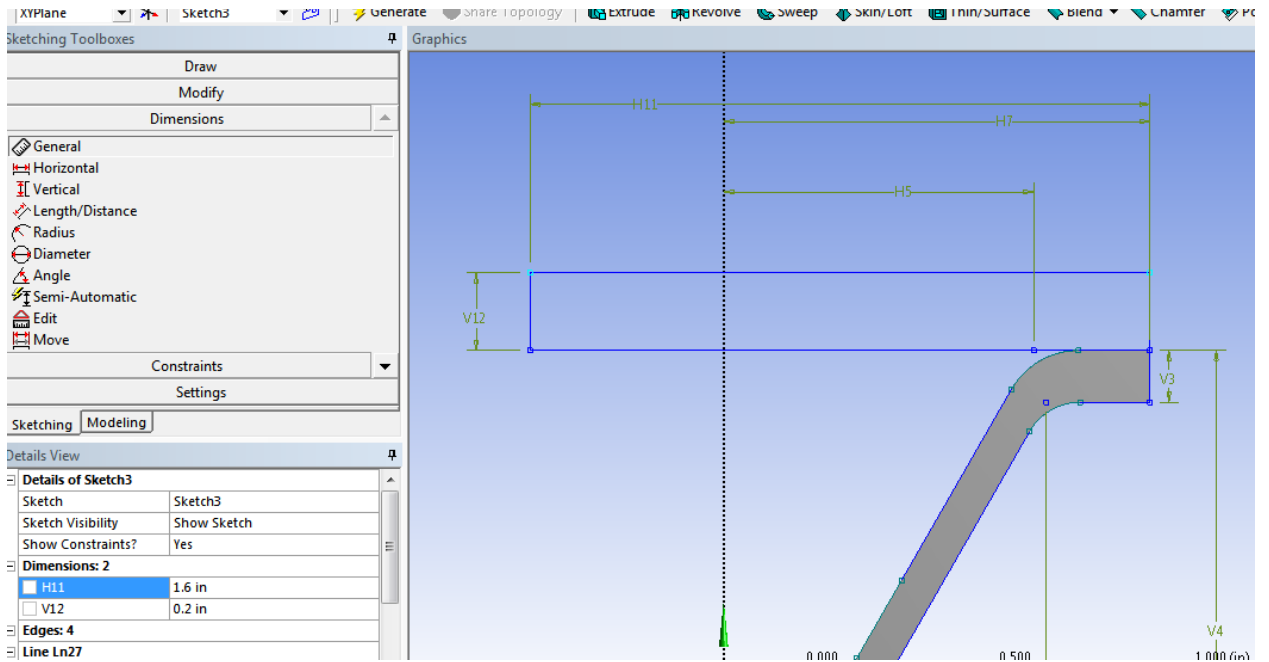
7. Create a new sketch to create the top plate



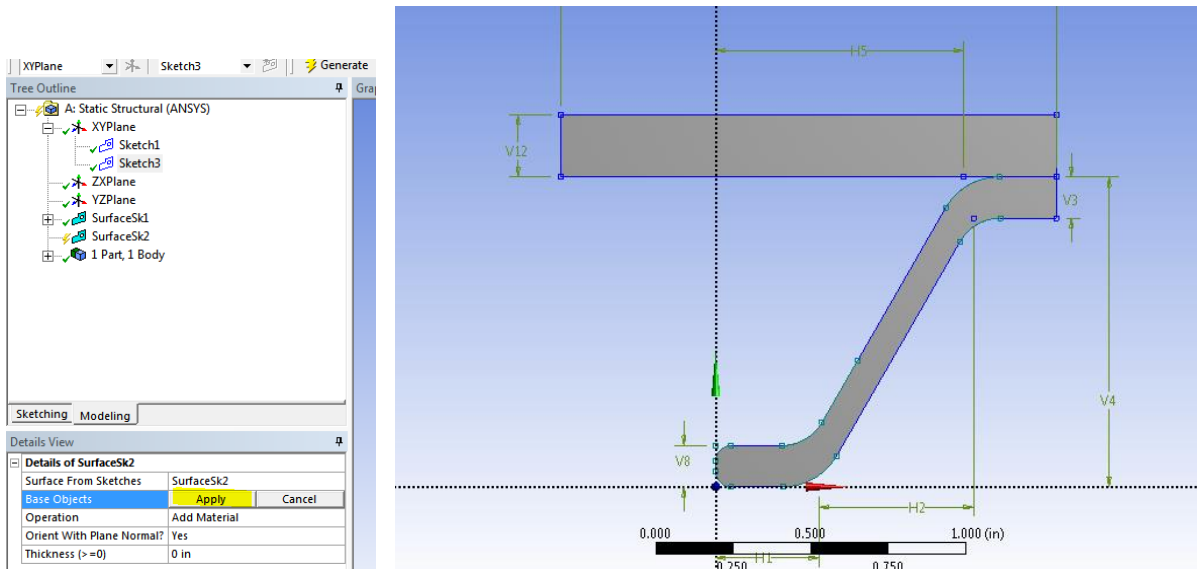
Select Sketching tab below the tree outline then select Rectangle from the Draw tab. Tip: If you cannot see all the drawing options, enlarge the window by pulling down on the line just above the Details view. Drag the cursor over the line and then expand the window until all the options are visible.



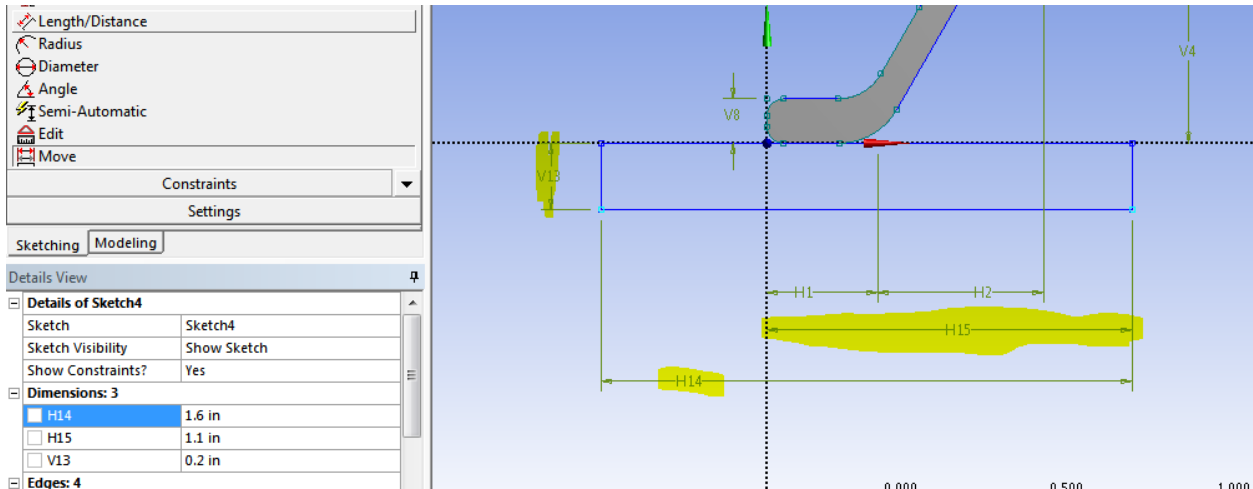
Align the lower right corner of the rectangle to the edge of the corner of the seal. Drag the mouse to create the rectangle as shown. The previously created dimensions can be moved (Dimensions Tab > Move) if they get in the way of the rectangle. Set the dimensions using the “General” option in the Dimension tab as previously described. The dimensions are 0.2 in thick by 1.6 inches long.



8. Create Surfaces from Sketches. Repeat step 6. Select the new sketch from the tree outline (your sketch number may differ, i.e. Sketch2). Apply and generate.

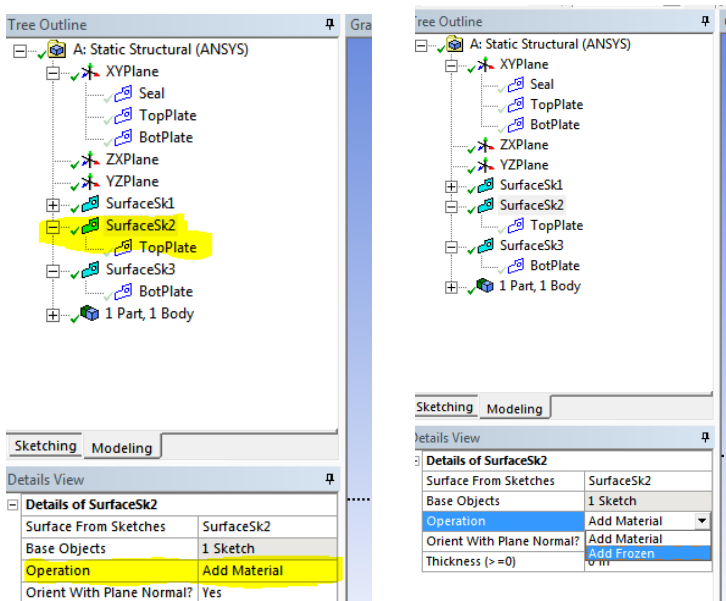


9. Create the bottom block. Align the top of the block with the bottom of the seal. The procedure is the same. Repeat the above steps. Set the dimensions.

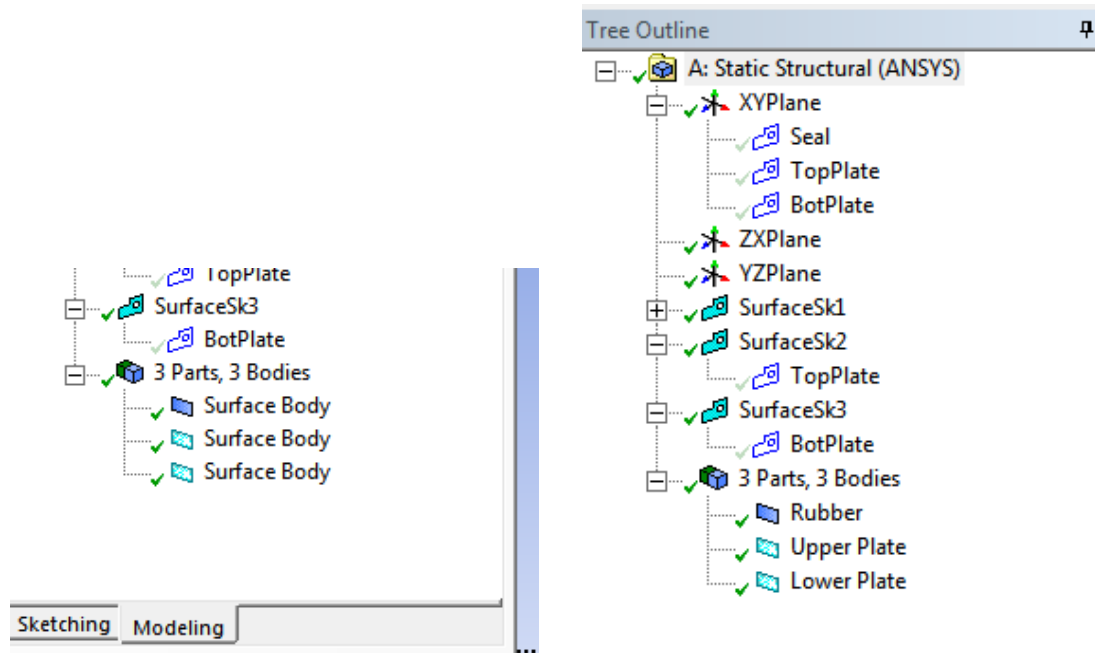


Rename the sketches to Seal, TopPlate, and BotPlate. Right click on the sketch in the tree outline and rename.

- In the Tree outline, select the topPlate surface. In Details View select Operation and change “Add Material” to “Add Frozen”. This will make the top plate separate from the seal. This step is required so that different material properties can be applied later to the seal and the plate. Select Generate after setting the operation to Add Frozen. Repeat for the bottom plate.



The 3 sections will have different colors after generating to indicate that the parts are separate. In the listing you will now see 3 Parts, 3 Bodies. Rename them to Rubber, Upper Plate, Lower Plate (right mouse click on name).



11. Proceed back to the tutorial for setting up and solving the simulation.