Chapter

4

Solid Part Three – A Bracket

This chapter will cover the following to World Class standards:

- **Sketch of Solid Problem Three**
- **Draw a Series of Lines**
- **Finish the 2D Sketch**
- **Extrude a 2D Sketch**
- **Change View to Wireframe**
- **Turn on the Plane's Visibility**
- **Project New Geometry**
- **Add another Sketch on a Model**
- **Extrude another 2D Sketch**
- **Add Multiple Fillets**
- **Add Multiple Holes**

Sketch of Solid Part Three

Again we start any project by making a sketch, so we can efficiently produce a drawing. In part 3, we see a sketch of a bracket. The length of the bracket legs are 2 inches long. The thickness of the material is 0.125. There are four 0.5 rounds or fillets and four 0.3125 diameter holes on the center points of each 0.5 fillet. There is a 0.125 thick rib that runs down the middle of the part.

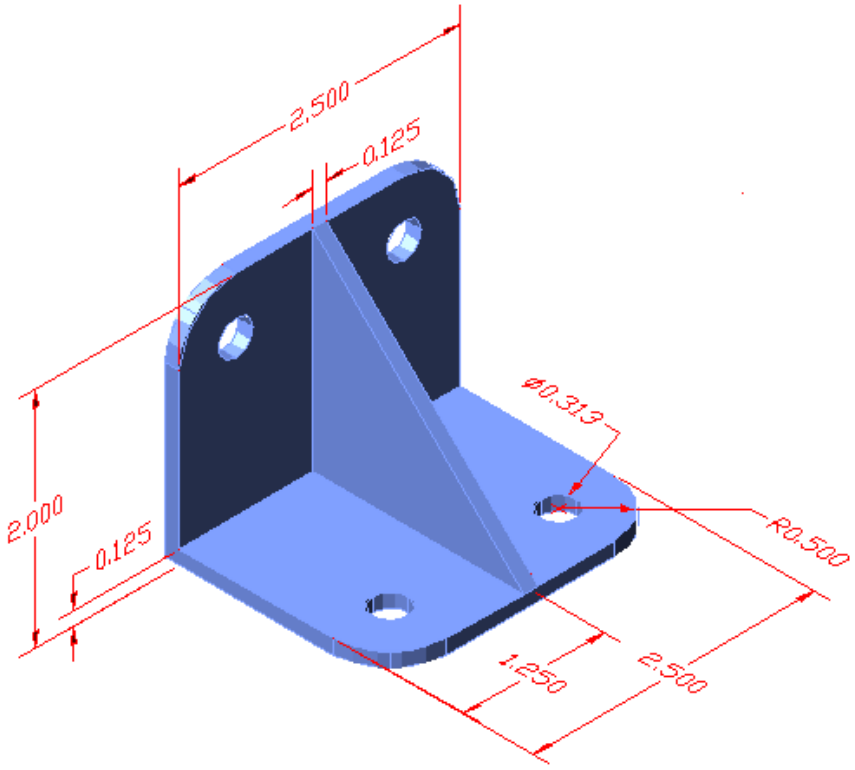


Figure 4.1 – Problem Three Sketch

In the third problem, we will practice techniques that we learned in the first part sketch and add some new experiences such as adding holes and fillets. We will continue to use multiple sketches and extrusion techniques to create the solid part.

Starting a 3D Part Drawing Sketch

When we open the AutoCAD Inventor application, we will select New from the menu. 

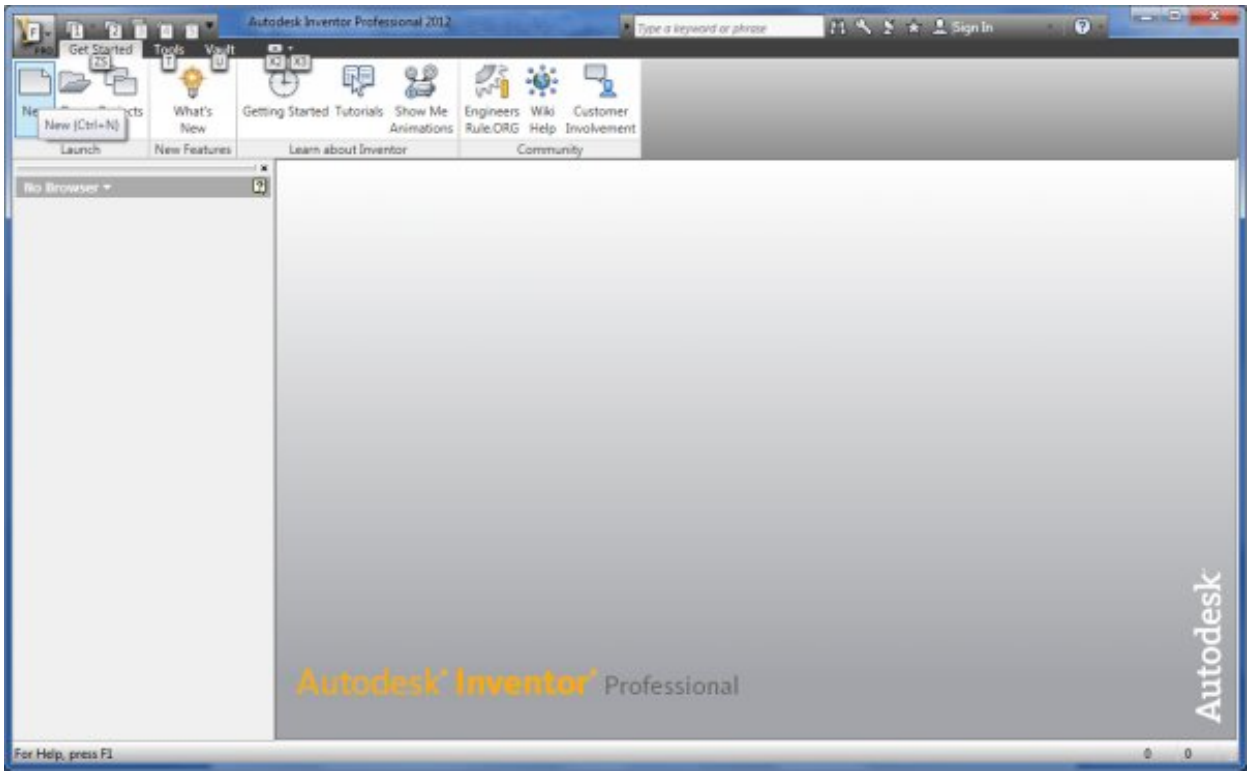


Figure 4.2 – AutoCAD Inventor Professional 2012

A New File window will appear and there are four tabs on this dialogue box. They are Default, English, Metric and Mold design. For this drawing, we will select the English tab and the Standard (in) ipt template. We will press the OK button to continue.

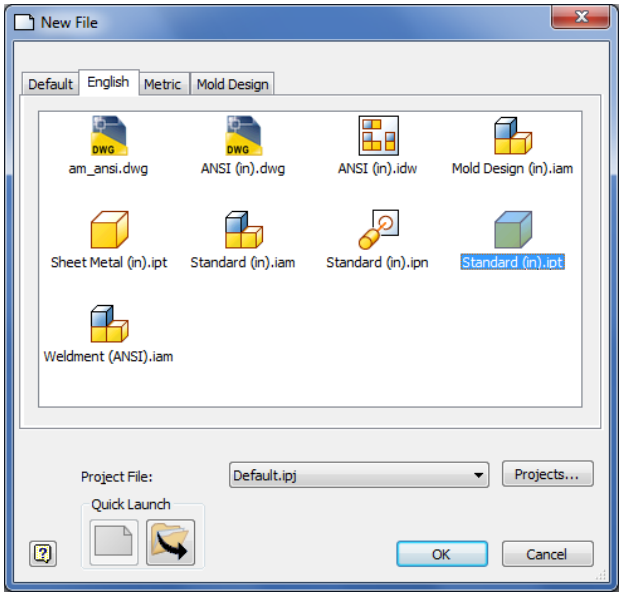


Figure 4.3 – Starting the drawing using the Standard IPT template

To turn off the grid if it is on the new drawing, we will go to the Tools tab on the Ribbon and choose Applications Options.

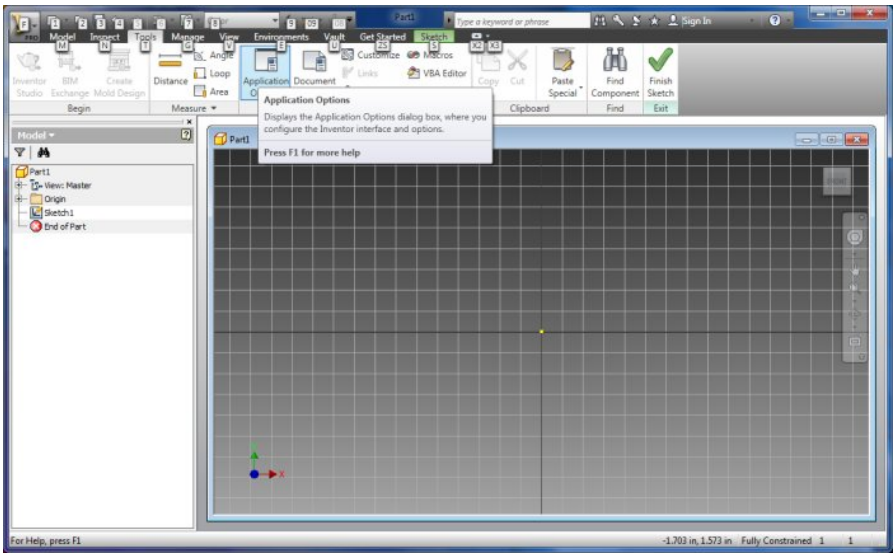


Figure 4.4 – Starting the drawing using the Standard IPT template

In the Applications Options dialogue box, we will turn off the Grid Lines.

For this chapter, we picked the Colors tab on the Applications Options and we select 1 background color and Presentation for the Color Scheme list. Having the grid and color on the drawing sketch background has no effect on the drawing, but is the designer's personal preference.

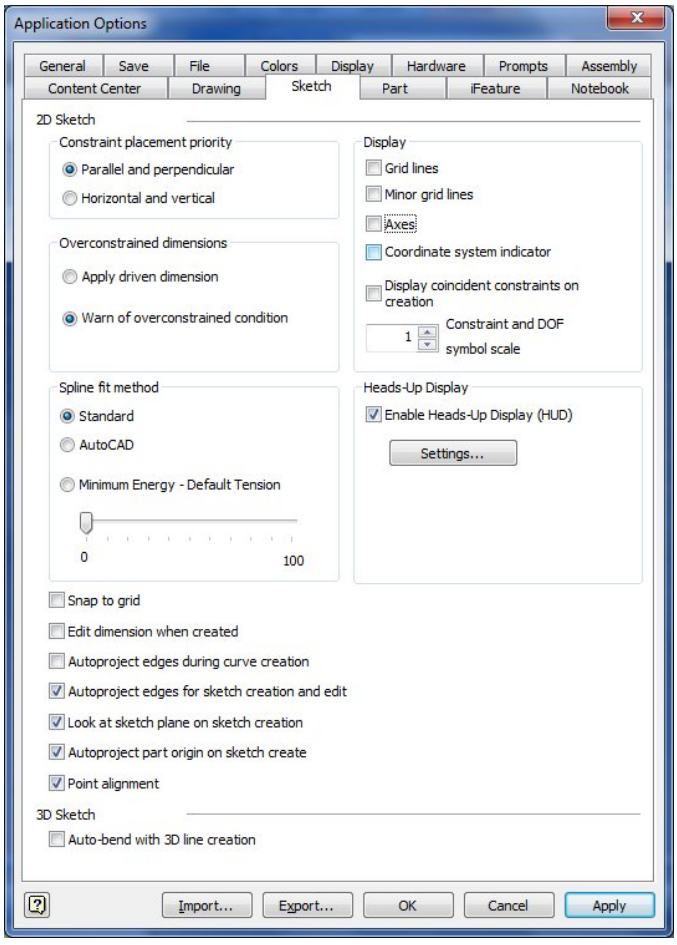
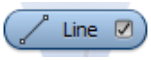


Figure 4.5 – Application Options Window

Drawing a Series of Lines

The entity we will learn to draw in Inventor is a Line. We right click on the drawing and we can see Line in the center top of the menu.



To draw a line, we right click on the drawing and we can see Line, Center Point Circle, Two Point Rectangle and many more choices. We pick Line and we will single click on the center portion of the graphical display and then we can pull the line in any direction.

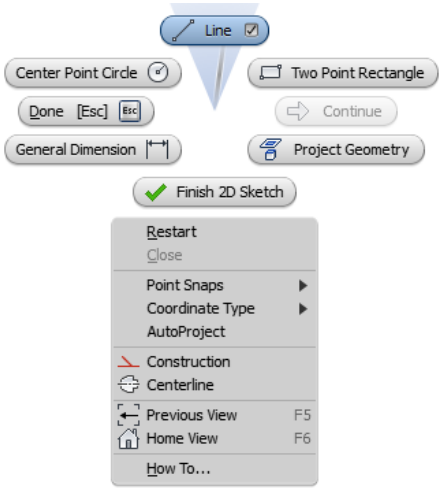


Figure 4.6 – Graphical Display Menu

To draw a series of lines to create a profile, we begin by picking a point on the lower left portion of the graphical display and then we pull the line to the right. We keep the cursor directly to the right and the application will report 0.00 degrees in the horizontal. We will input 2 in the measurement textbox and press the Enter key.

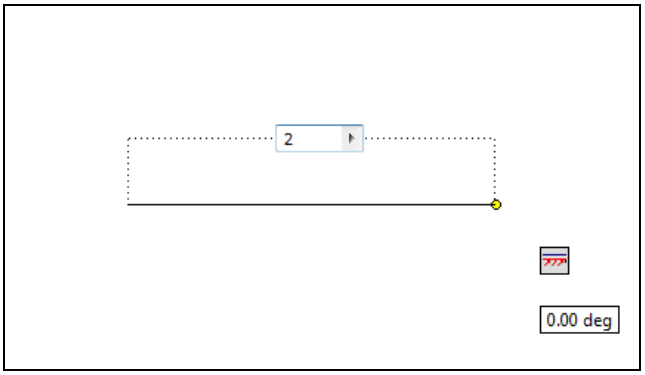


Figure 4.7 – First Line Segment

Next, we draw a line perpendicularly upward at 90 degrees. We will input 2 in the measurement textbox and press the Enter key.

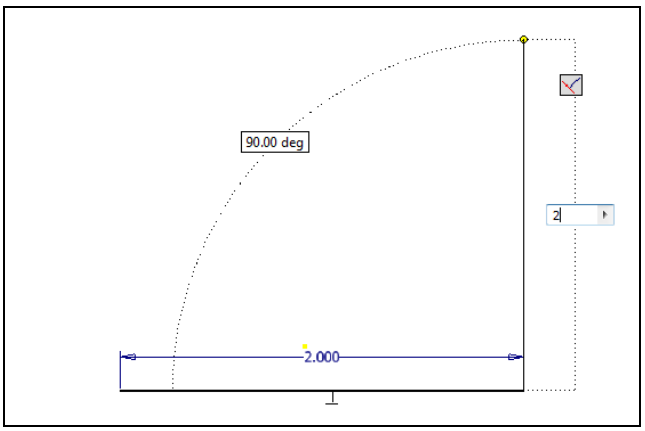


Figure 4.8 – Second Line Segment

Then we draw a line perpendicularly to the left at 90 degrees. We will input 0.125 in the measurement textbox and press the Enter key.

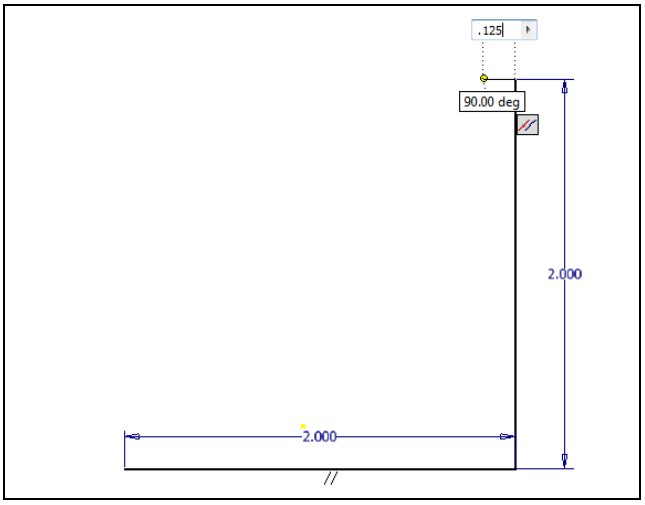


Figure 4.9 – Third Line Segment

We then draw a line perpendicularly downward at 90 degrees. We will input 1.875 in the measurement textbox and press the Enter key.

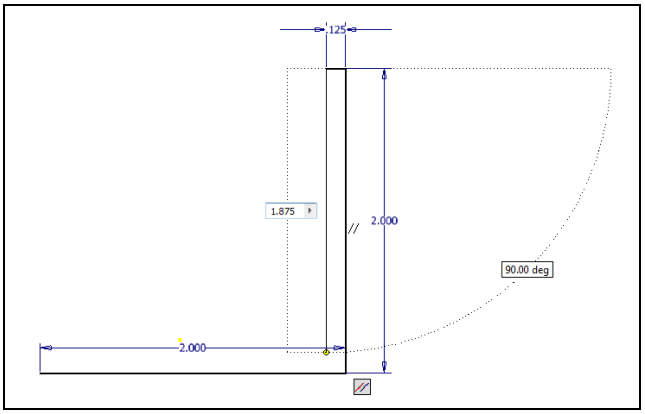


Figure 4.10 – Fourth Line Segment

We then draw a line perpendicularly to the left at 90 degrees. We will input 1.875 in the measurement textbox and press the Enter key.

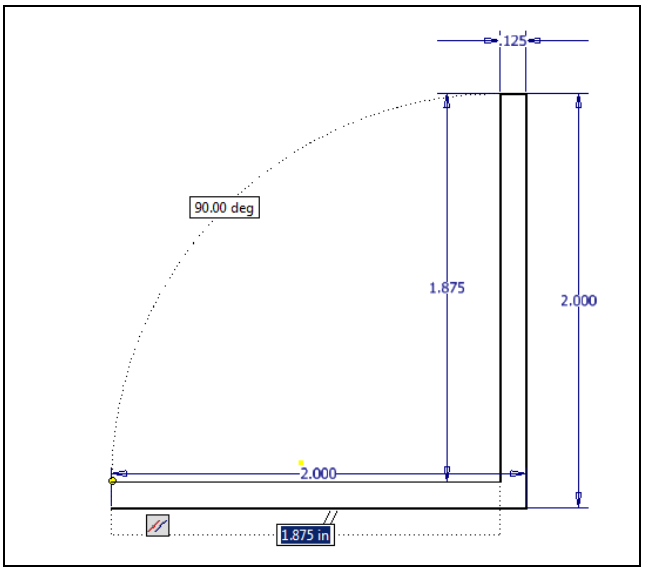


Figure 4.11 – Fifth Line Segment

For the last segment, we will want to close the profile, so we right click on the graphical display and we select Close from the menu. The bracket will appear as shown in Figure 4.13.

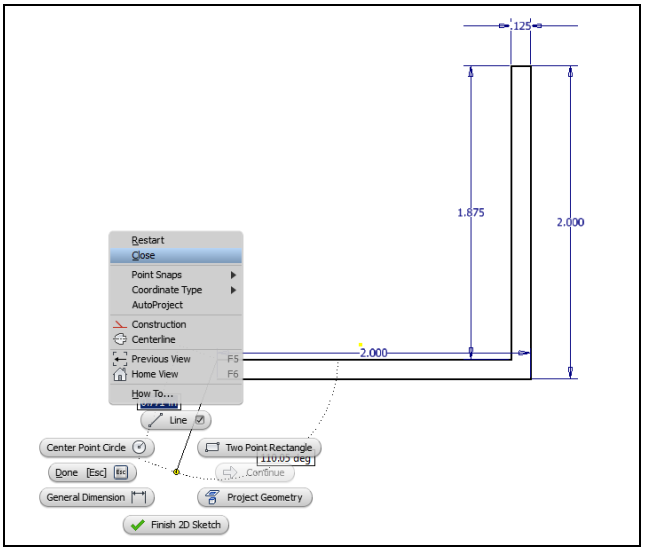


Figure 4.12 – Close the Profile

We will draw another line segment from the inside leg of the profile as shown in figure 4.19.

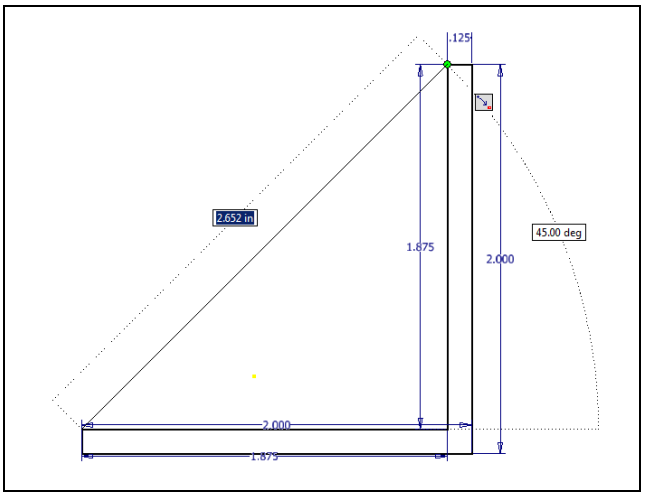


Figure 4.13 – Diagonal Line

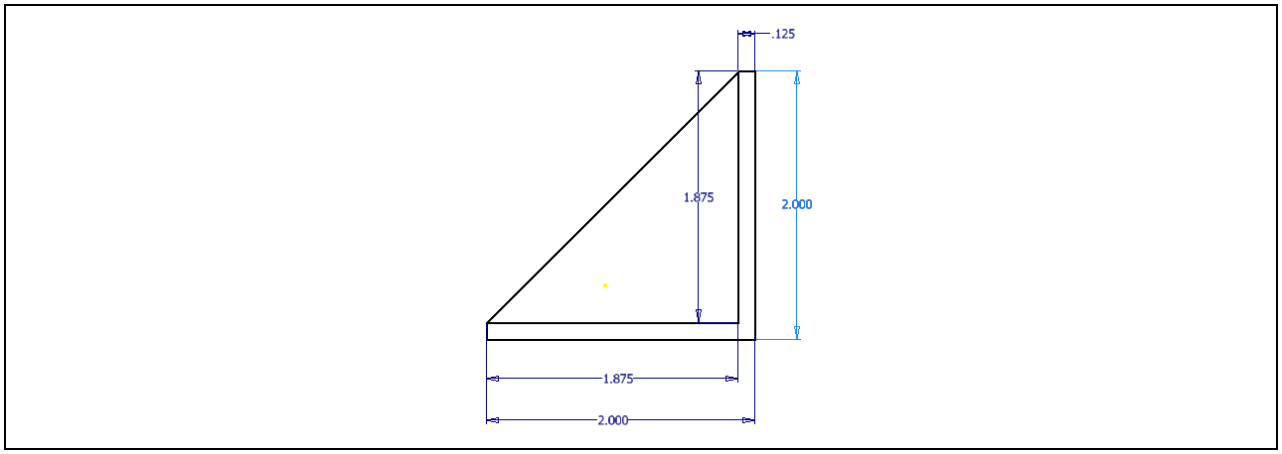


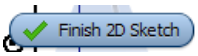
Figure 4.14 – Profile with Dimension Showing

*** World Class CAD Challenge 61-06 * - Close this drawing file. Create a New file and draw the profile of seven lines. Complete the task in less than 5 minutes. Continue this drill four times, each time completing the drawing under 5 minutes to maintain your World Class ranking.**

*** World Class CAD Challenge * - Report your best times to World Class CAD at www.worldclasscad.com to obtain your world class ranking.**

Finish 2D Sketch of Solid Part One

Before we extrude the sketch, we need to right click on the graphical display and on the menu; we choose the Finish 2D Sketch button.



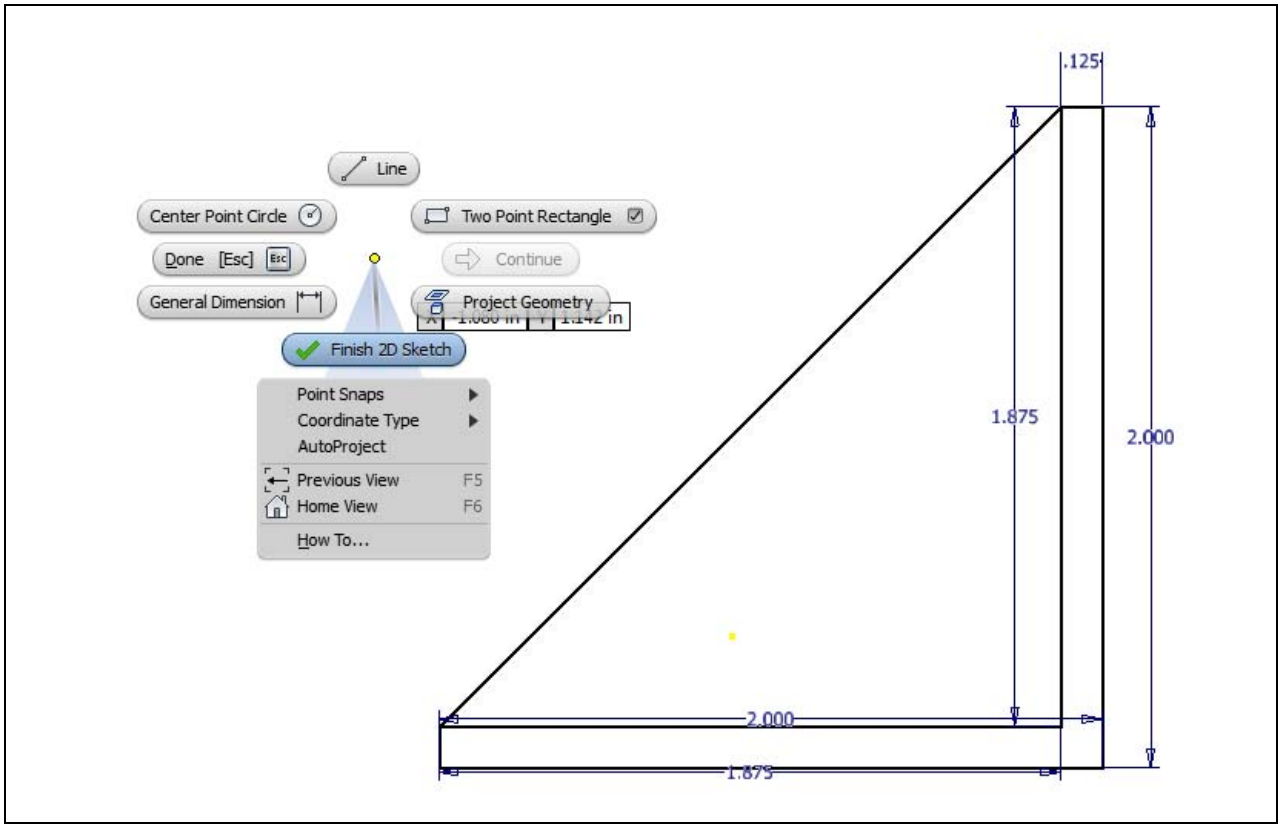
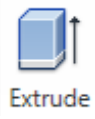


Figure 4.15 – Finish 2D Sketch

Extruding a 2D Sketch

Now that we have a finished sketch, we need to extrude the part. We can go ahead and pick the Extrude button on the Model tab of the Invertor ribbon. The Extrude window will appear on the display.



On the Extrude window, we can either output a solid or surface. The differences between the two are that the first is like a hard piece of aluminum and the second choice is similar to a box. We will pick the Solid output on the left. Next, our part will be made from finished aluminum, so we will change the Extents distance from 1.0 to 0.125.

Underneath the distance textbox, we will pick the symmetric button (second from the right), so that the sketch plane will stay in the middle of the part.

We select the triangular area and when it turns red, we click again to extrude the solid.

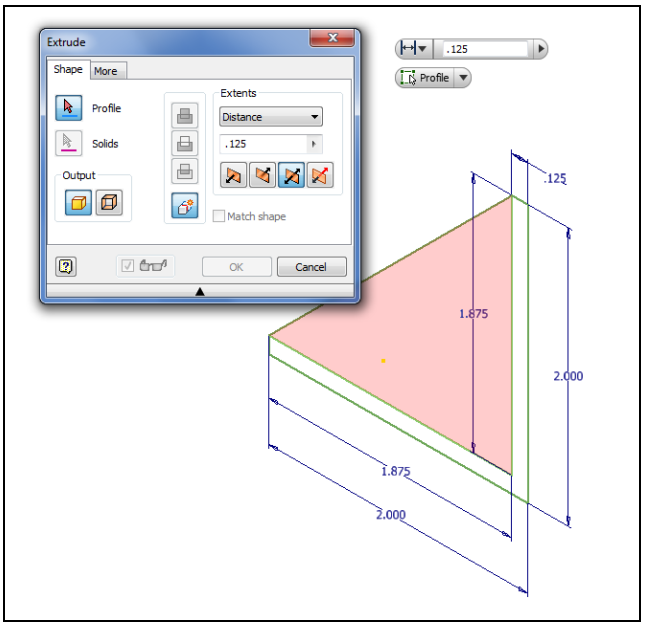


Figure 4.16 – The Extrude Window

We then select the bracket legs and when it turns red, we click again to extrude the solid.

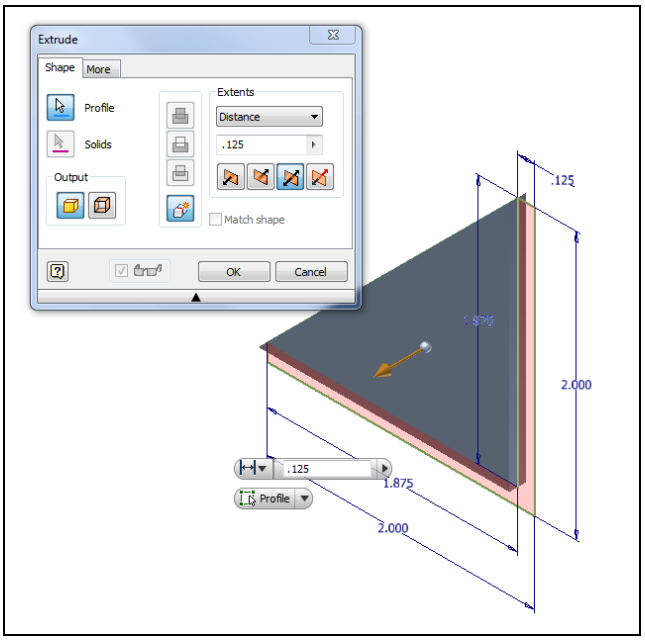


Figure 4.17 – More Extrusions

We are going to extrude the bracket legs next, so we go to View on the Inventor ribbon and we choose Visual Style and Wireframe.

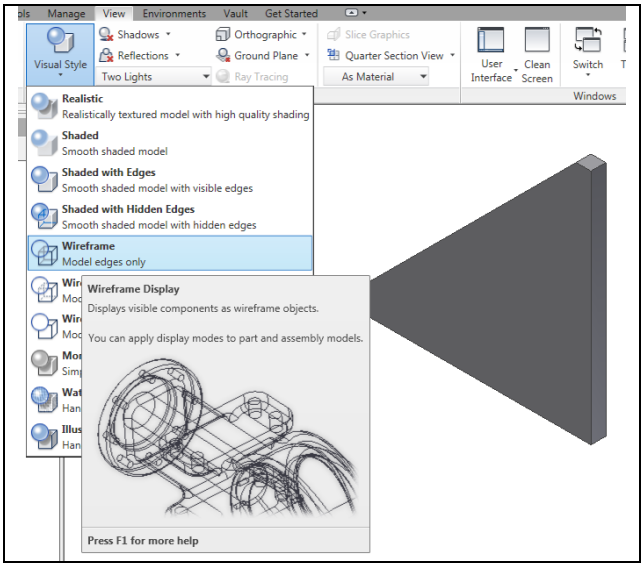


Figure 4.18 – Change to Wireframe View

We expand the Origin folder in the left pane and we right click on the X-Y plane and turn on the visibility so we can pick the plane to do the new sketch for the width of the bracket.

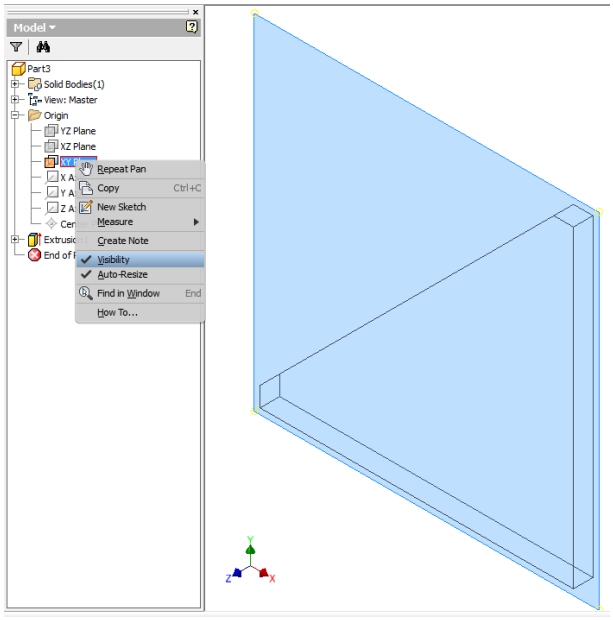


Figure 4.19 – Turn On X-Y Plane Visibility

Create New Geometry and Extrude a New 2D Sketch

We right click on the graphic display and we choose the New Sketch button.

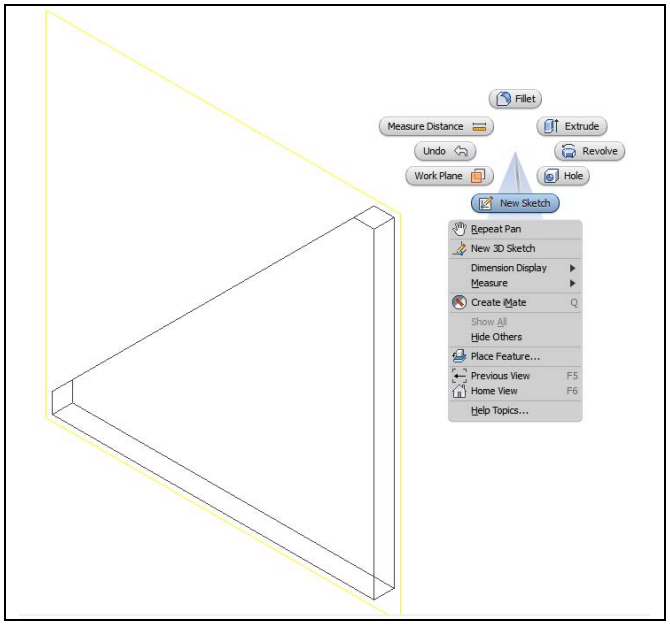


Figure 4.20 – Create a New Sketch

We will be prompted to select what plane to make our sketch on and we will pick the visible X-Y plane that will turn red.

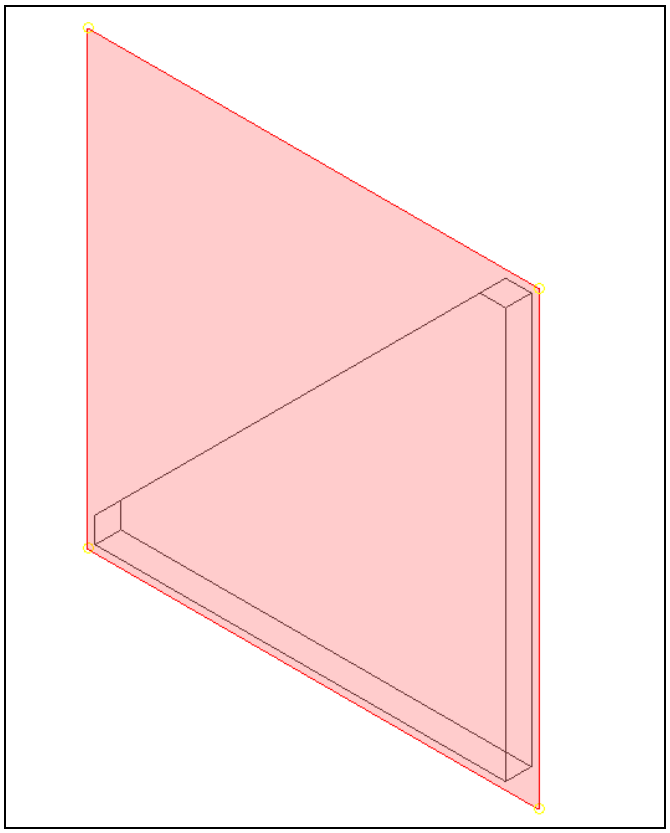


Figure 4.21 – Pick the Plane

We will select Project Geometry from the Inventor ribbon and Project geometry from the menu.

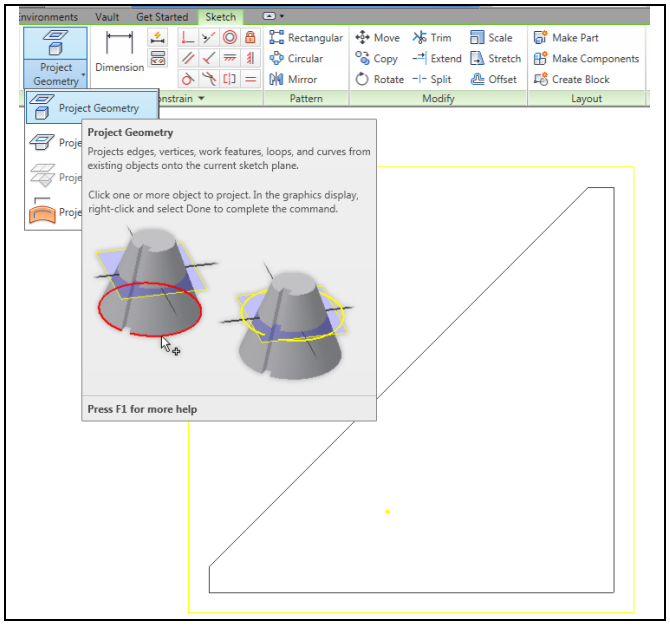


Figure 4.22 – Project the Geometry

We will select the four lines on the projected geometry.

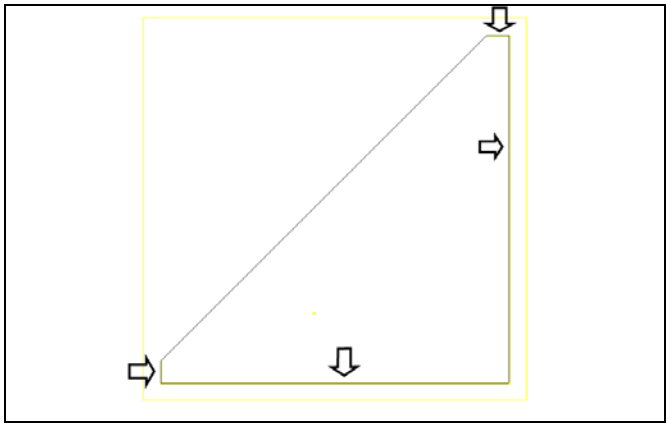


Figure 4.23 – Select the Projected Geometry

We then draw two new 1.875 lines as shown in figure 4.24.

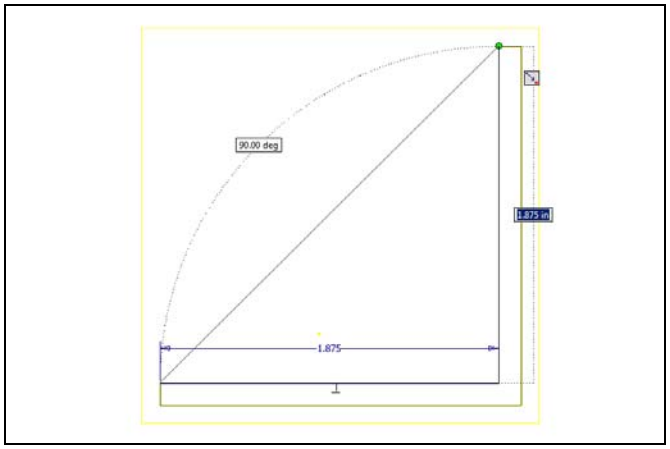


Figure 4.24 – Draw Two New Lines

Before we extrude the sketch, we need to right click on the graphical display and on the menu, we choose the Finish 2D Sketch button.

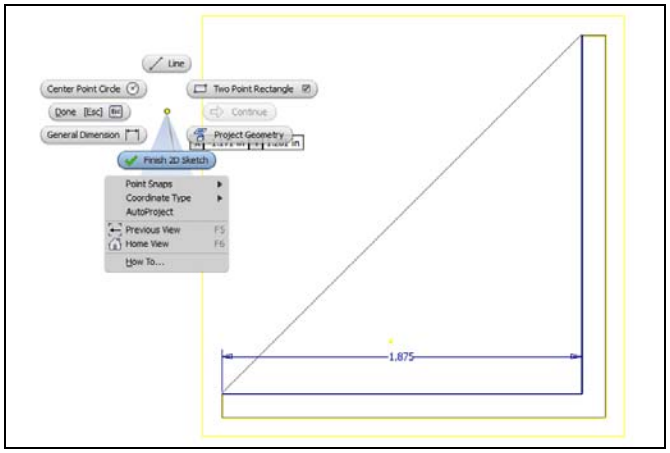


Figure 4.25 – Extrude the Second 2D Sketch

We now chose the Extrude button on the Inventor ribbon. We will maintain the Solid output on the left. Next, our part will be made from finished aluminum, so we will change the Extents distance from 2.5 to 0.125.

Underneath the distance textbox, we will pick the symmetric button (second from the right), so that the sketch plane will stay in the middle of the part.

We select the bracket legs area and when it turns blue, we click again to extrude the solid.

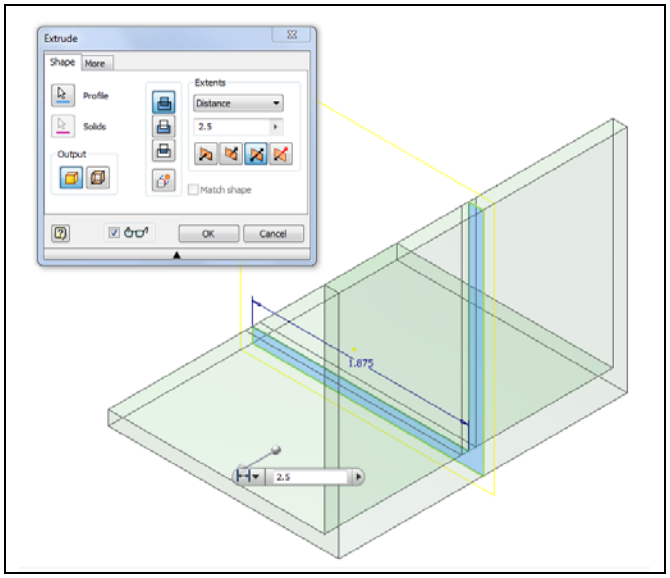


Figure 4.26 – Extrude the 2D Sketch

We change the view back, so we go to View on the Inventor ribbon and we choose Visual Style and Shaded with Edges.

Save the drawing and we will now have to add four fillets and holes to the bracket.

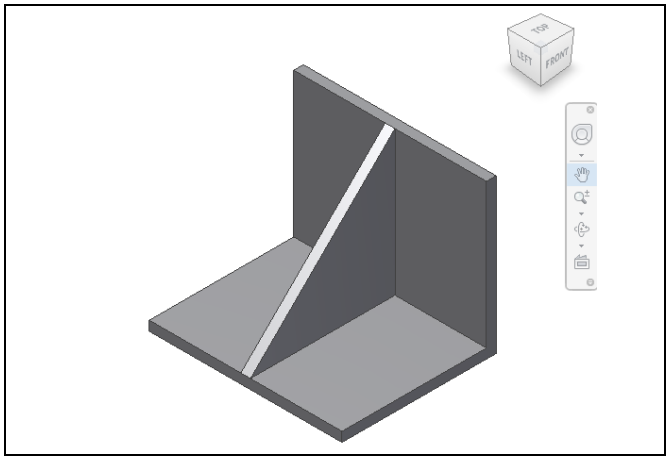


Figure 4.27 – Finished Solid

*** World Class CAD Challenge 61-07 * - Close this drawing file. Create a New file and draw the sketch, extrude it and then add geometry to extrude the bracket width. Extrude the part. Complete the task in less than 7 minutes. Continue this drill four times, each time completing the drawing under 7 minutes to maintain your World Class ranking.**

*** World Class CAD Challenge * - Report your best times to World Class CAD at www.worldclasscad.com to obtain your world class ranking.**

Drawing Multiple Fillets

The next feature we will add to our bracket is the four fillets. We choose the Fillet button on the Inventor ribbon and the Fillet window will appear on the graphical display.

We set the fillet radius to 0.5. When we select the straight edged corner, the pointed edge will change to a 0.5 inch rounded corner.

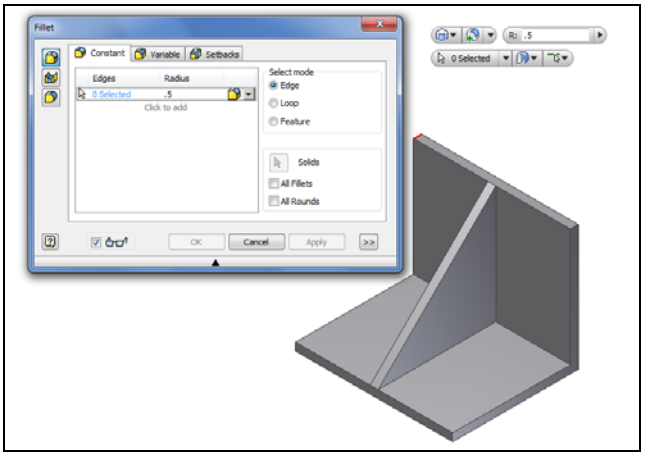


Figure 4.28 – First Fillet

We should select the other three straight edges and the pointed edges will change to a 0.5 inch rounded corners.

To make the placements permanent, we press the OK button.

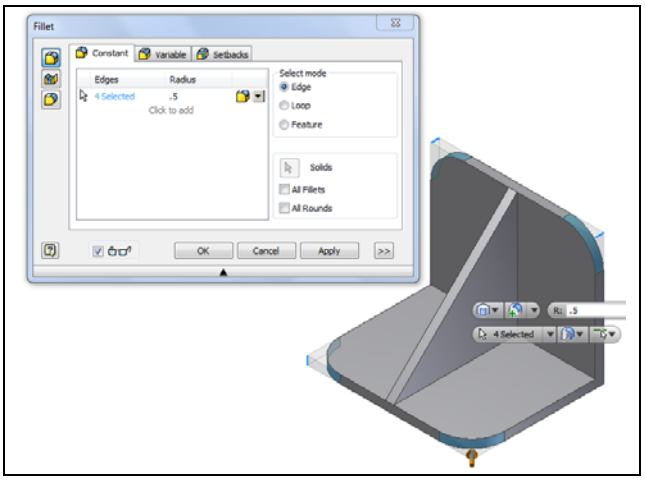
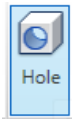


Figure 4.29 – Second through Fourth Fillet

Drawing Multiple Holes

The next entity we will learn to draw in Inventor is a Hole. We choose the Hole button on the Inventor ribbon and the Hole window will appear on the graphical display.



We begin the process of adding a hole by making a new sketch.

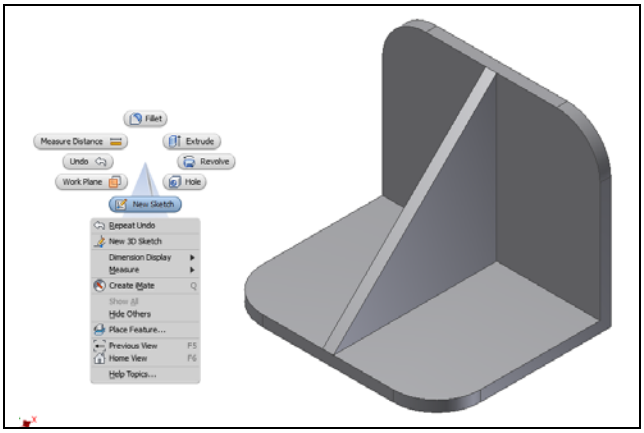


Figure 4.30 – New Sketch

We then choose the plane for the new sketch, so we pick the surface as shown in figure 4.31.

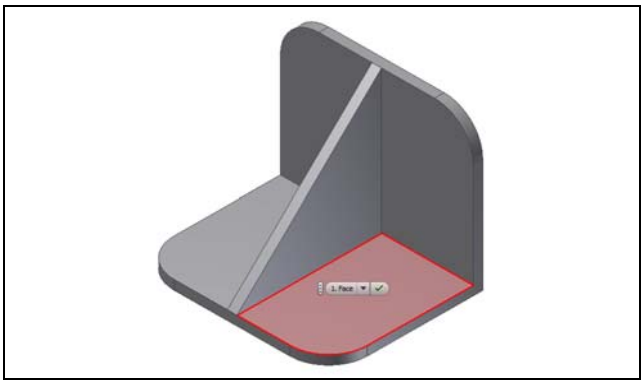


Figure 4.31 – Select the Plane

We then finish the new sketch since the center of the hole is the same as the center of the fillet radius.

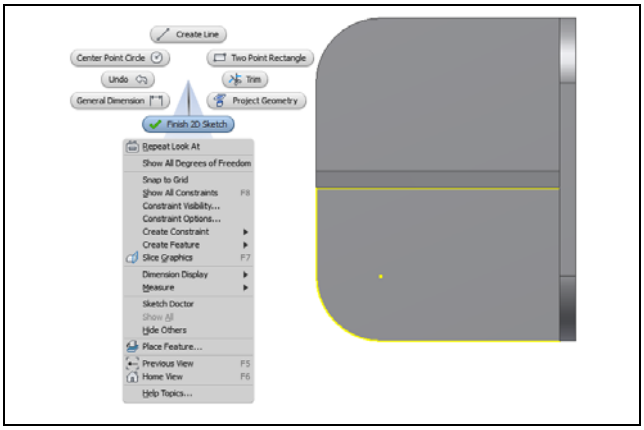


Figure 4.32 – Finish the 2D Sketch

We then select the Hole button on the Inventor ribbon and the Hole window will appear on the graphical display. We are making a 0.3125 through hole, so we change the diameter textbox from 0.25 to 0.3125. We then pick the Center icon and we choose the center point of the 0.5 radius. We see the red dot appear. A hole will appear and we press the OK button to retain the feature.

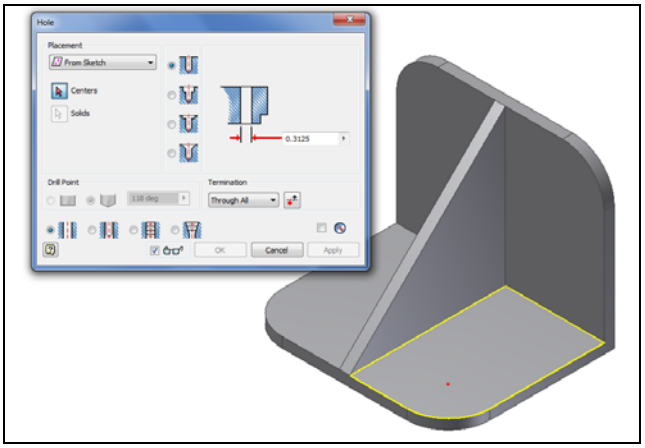


Figure 4.33 – Select the Hole Center

To repeat the process on the other side, we:

1. Create a new sketch
2. Select the plane
3. Finish the 2D sketch
4. Select Hole on the Inventor ribbon
5. Set diameter to 0.3125
6. Select the center (center of 0.5 radius)
7. Press OK to retain the hole

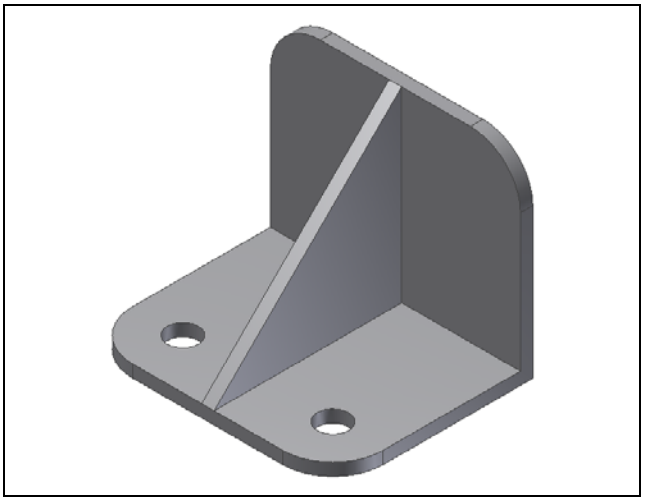


Figure 4.34 – Repeat the Process

Again, we repeat the process on the top side.

1. Create a new sketch
2. Select the plane
3. Finish the 2D sketch
4. Select Hole on the Inventor ribbon
5. Set diameter to 0.3125
6. Select the center (center of 0.5 radius)
7. Press OK to retain the hole

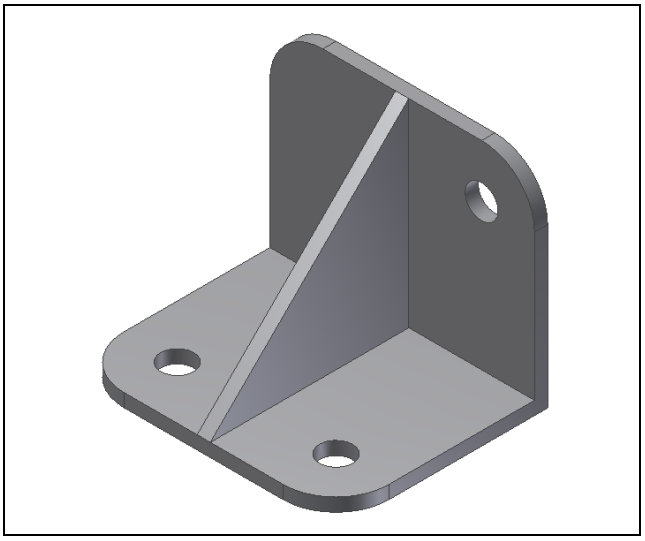


Figure 4.35 – Repeat the Process

We could have made one-half of the bracket and mirrored the features across the centerline, but we will save that for another lesson. We should make four holes and then the solid is completely finished and shown in figure 4.37.

Save the solid bracket and we will make a new one in the next chapter.

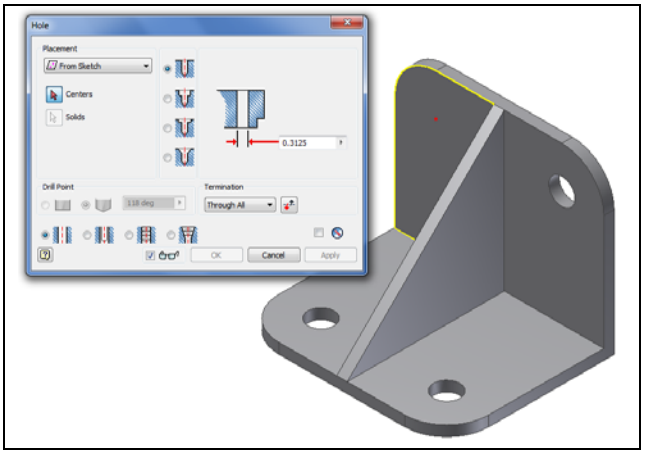


Figure 4.36 – Repeat the Process

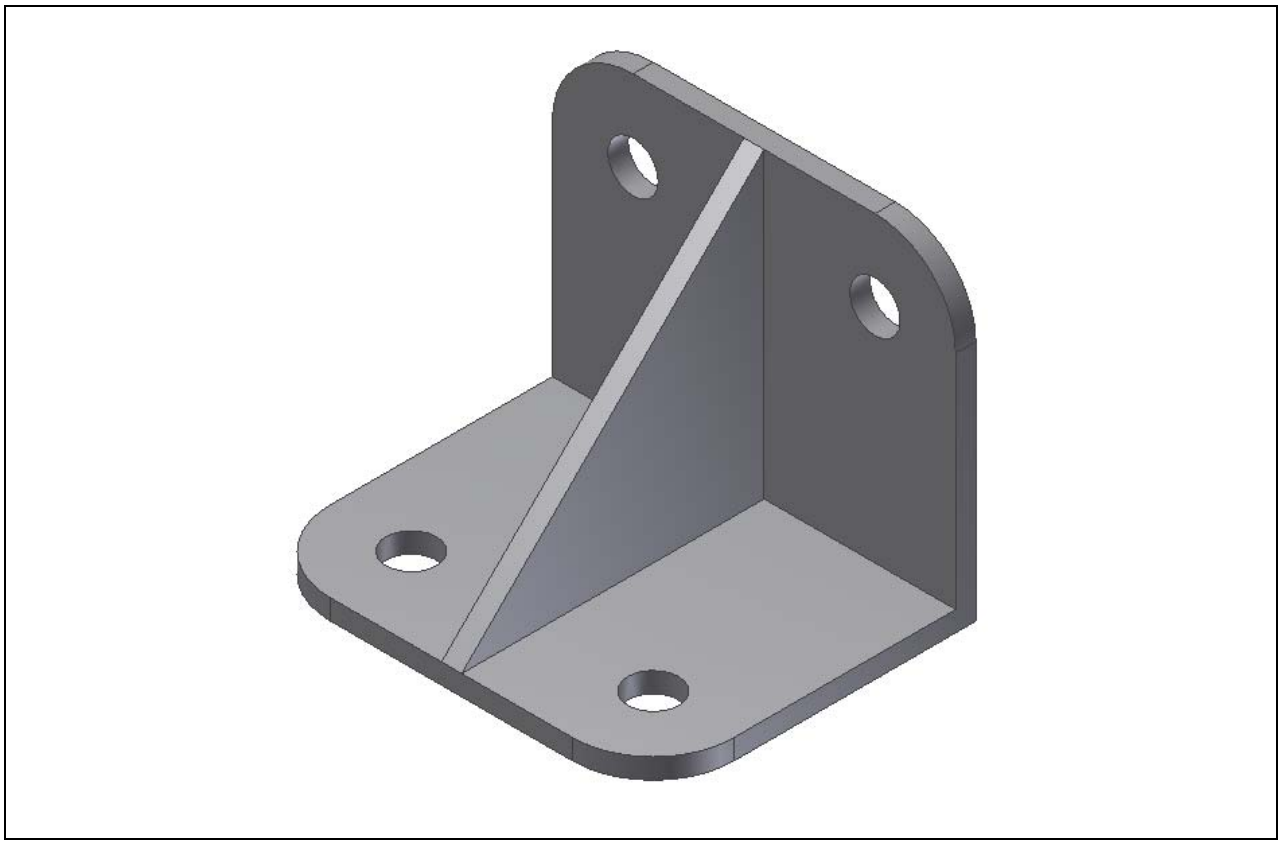


Figure 4.37 – The Finished Solid

*** World Class CAD Challenge 61-08 * - Close this drawing file. Create a New file and draw the sketch, extrude it and then add geometry to extrude the bracket width. Extrude the part. Add four fillets and four holes. Complete the task in less than 10 minutes. Continue this drill four times, each time completing the drawing under 10 minutes to maintain your World Class ranking.**

*** World Class CAD Challenge * - Report your best times to World Class CAD at www.worldclasscad.com to obtain your world class ranking.**