Chapter

5

Solid Part Four – A Bracket Made by Mirroring

This chapter will cover the following to World Class standards:

- **Sketch of a Solid Problem**
- **Draw a Series of Lines**
- **Finish the 2D Sketch**
- **Extrude a 2D Sketch**
- **Add Multiple Fillets**
- **Add Multiple Holes**
- **Add a Slotted Hole**
- **Add a Tapped Hole**
- **Mirror the Solid**

Sketch of Solid Part Four

Again we start any project by making a sketch, so we can efficiently produce a drawing. In part 4, we see a sketch of another bracket. The length of the bracket legs are 1 inch long. The thickness of the material is 0.125. There are four 0.125 rounds or fillets and four 0.201 diameter holes on the feet of the bracket. At the top of the bracket, we have a 0.5 by 1.0 slotted hole and two 6-32 UNC tapped holes.

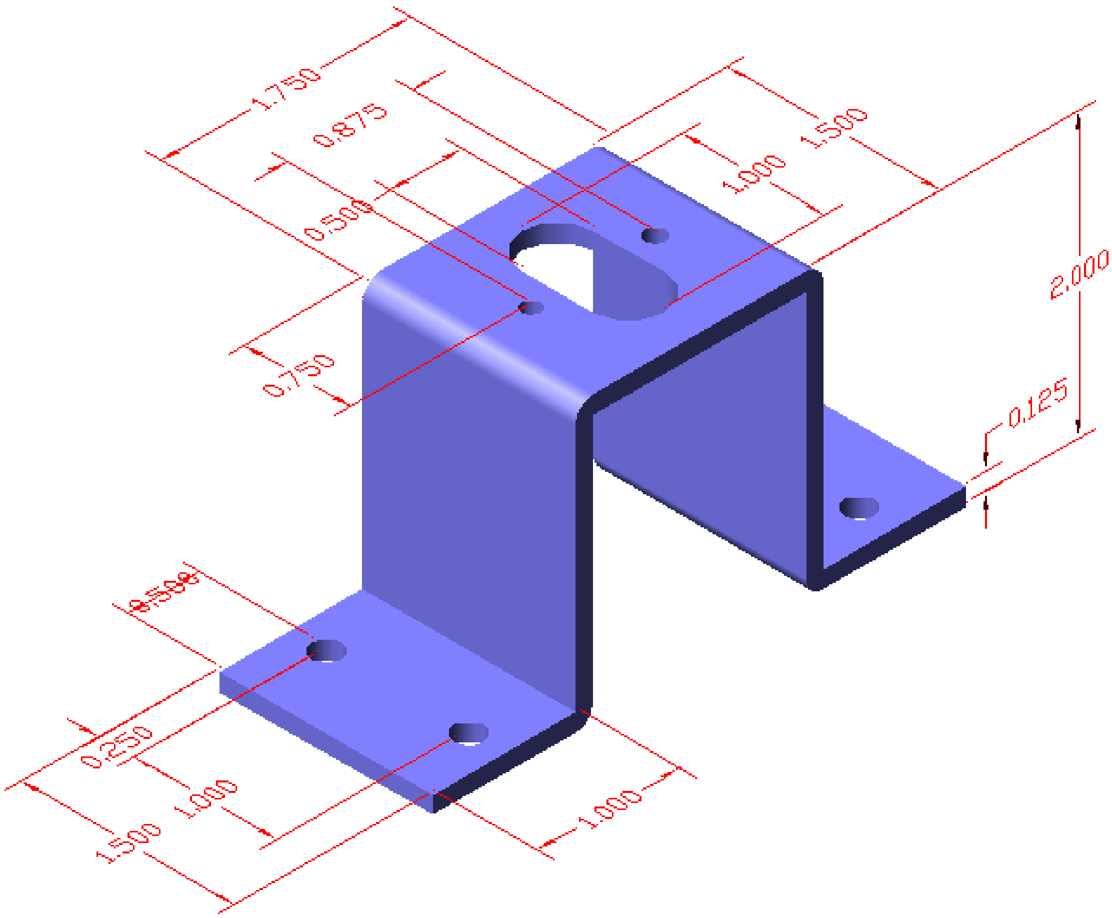


Figure 5.1 – Problem Four Sketch

In the fourth problem, we will practice techniques that we learned in the previous part sketches and add some new experiences such as slotted holes, tapped holes and mirrored solids. We will continue to add holes and fillets. We will still use multiple sketches and extrusion techniques to create the solid part.

In this project, we will only draw half of the bracket and then when we are done with graphically describing all of the features, we will mirror the solid to make the entire bracket.

Starting a 3D Part Drawing Sketch

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

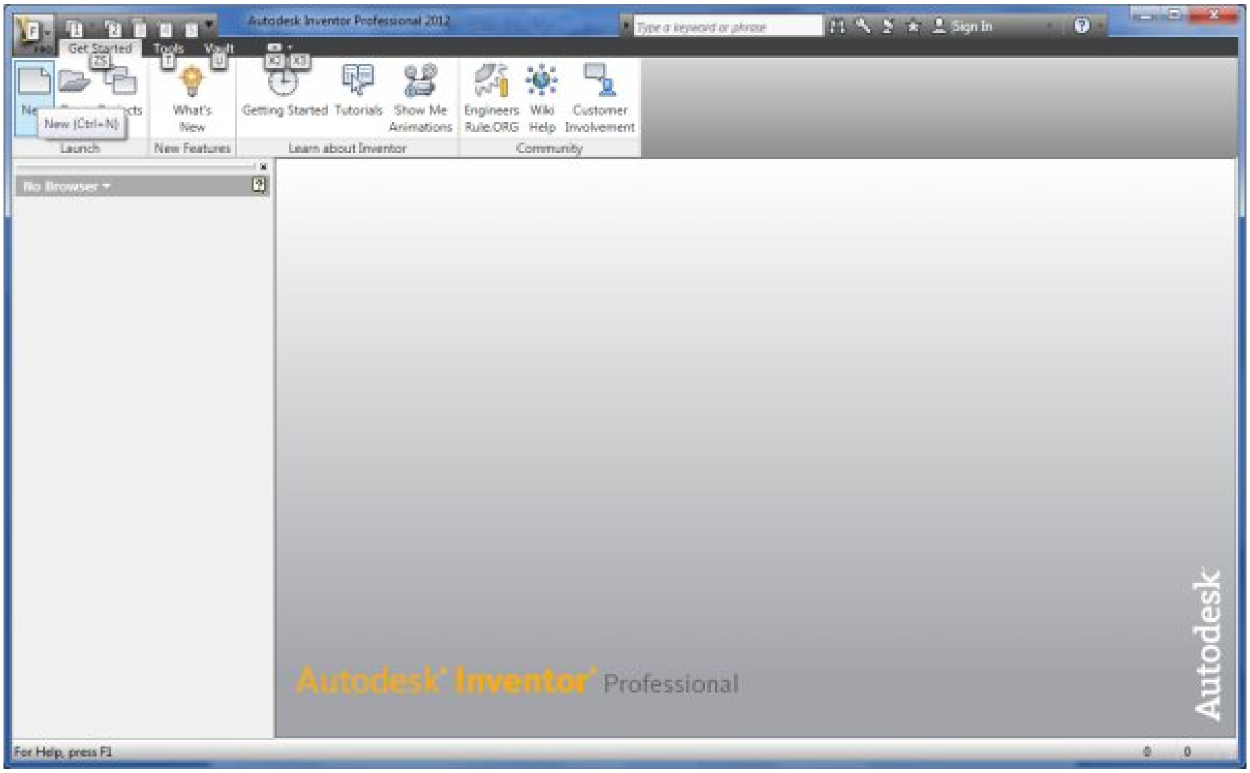


Figure 5.2 – AutoCAD Inventor Professional 2012

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.

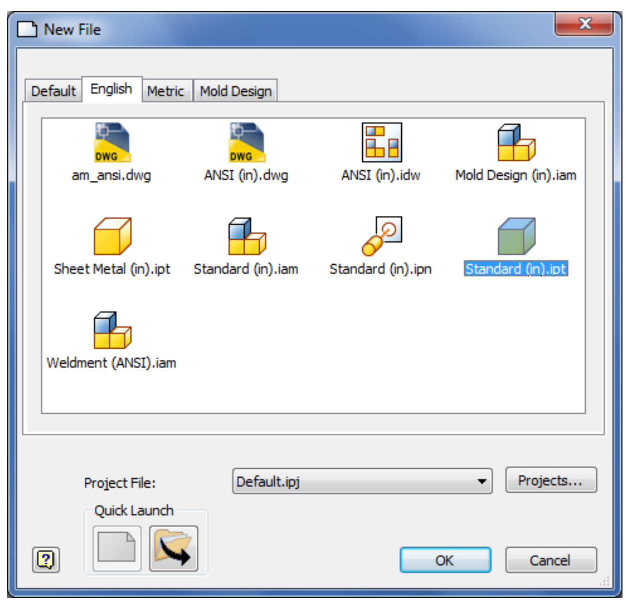


Figure 5.3 – Starting the drawing using the Standard IPT template

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.

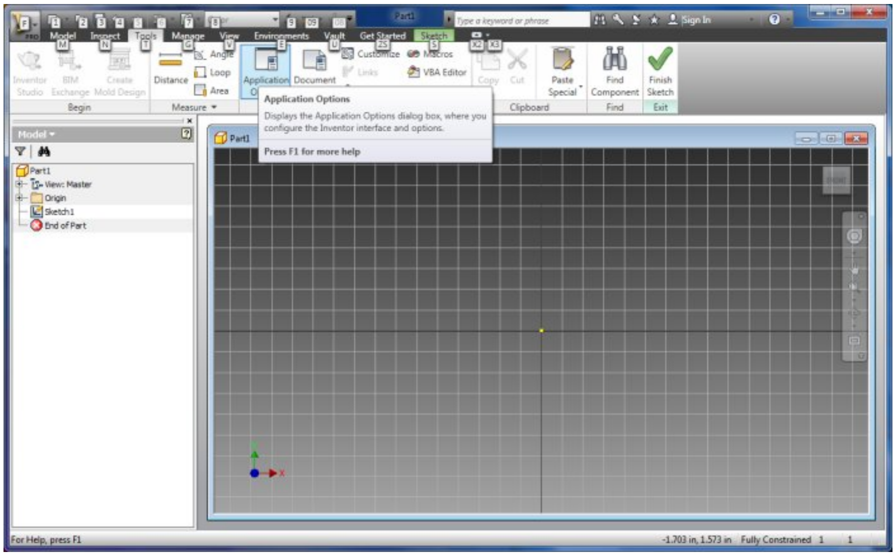


Figure 5.4 – Starting the drawing using the Standard IPT template

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.

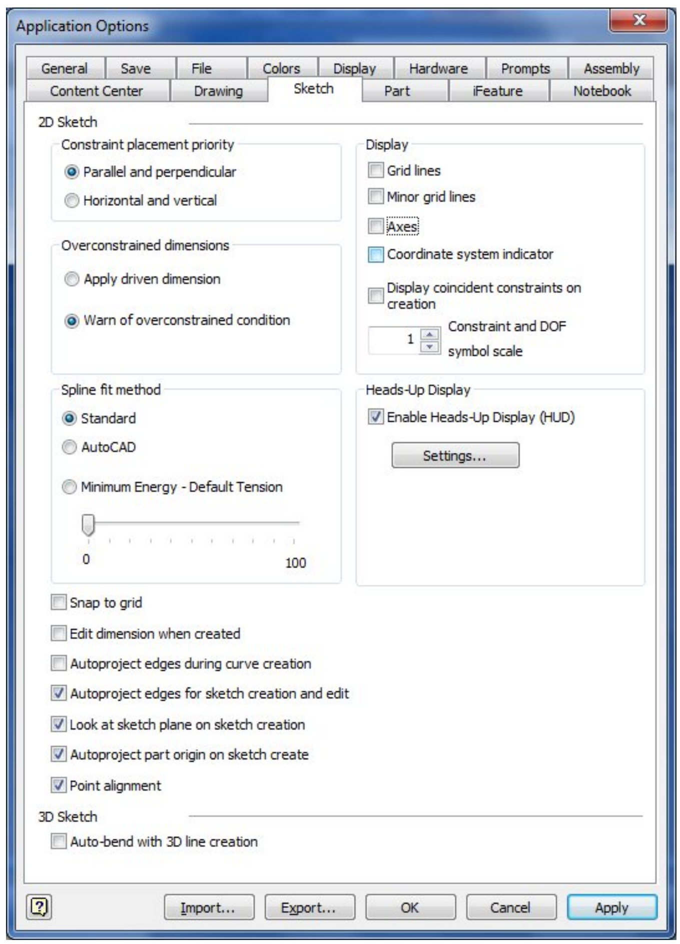


Figure 5.5 – Application Options Window

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.

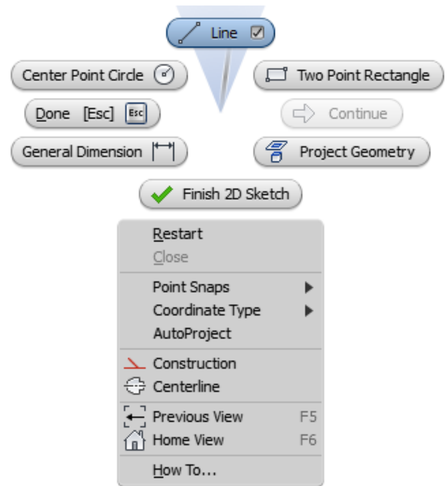


Figure 5.6 – Graphical Display Menu

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 1.125 in the measurement textbox and press the Enter key.

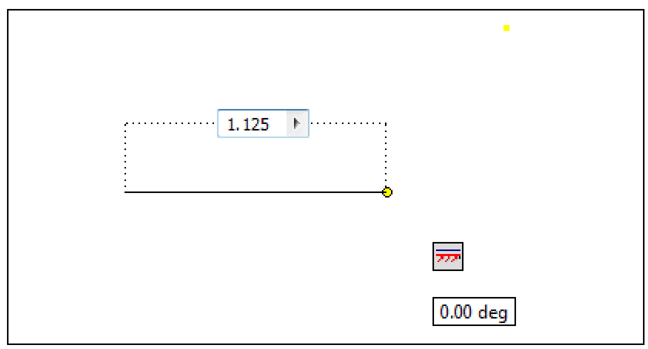


Figure 5.7 – First Line Segment

Next, we draw a line perpendicularly upward at 90 degrees. We will input 1.875 in the measurement textbox and press the Enter key.

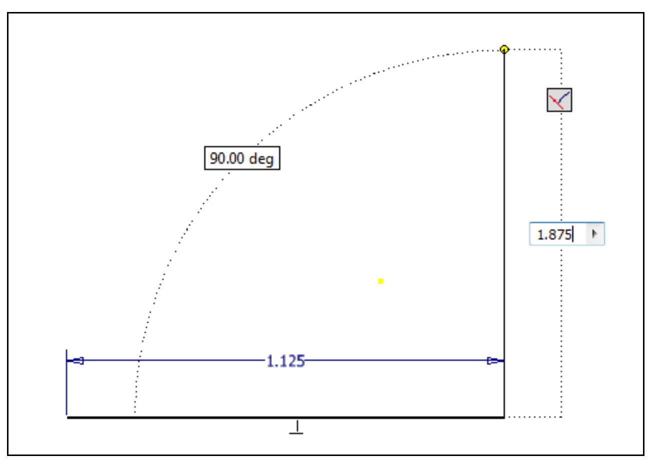


Figure 5.8 – Second Line Segment

Then we draw a line perpendicularly to the right at 90 degrees. We will input 0.75 in the measurement textbox and press the Enter key.

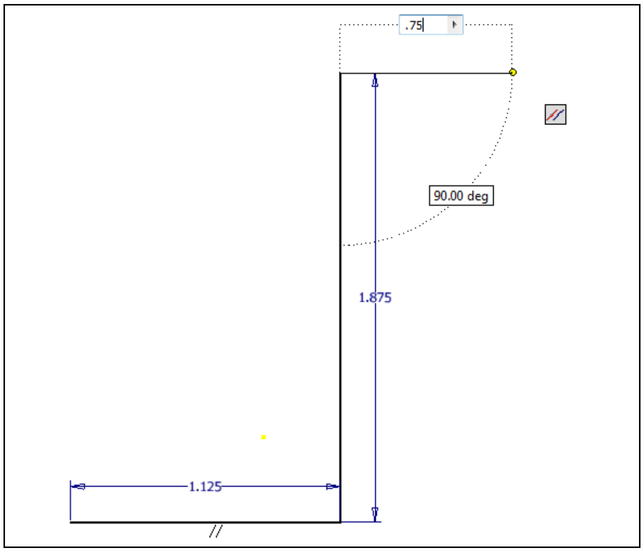


Figure 5.9 – Third Line Segment

We then draw a line perpendicularly upward at 90 degrees. We will input 0.125 in the measurement textbox and press the Enter key.

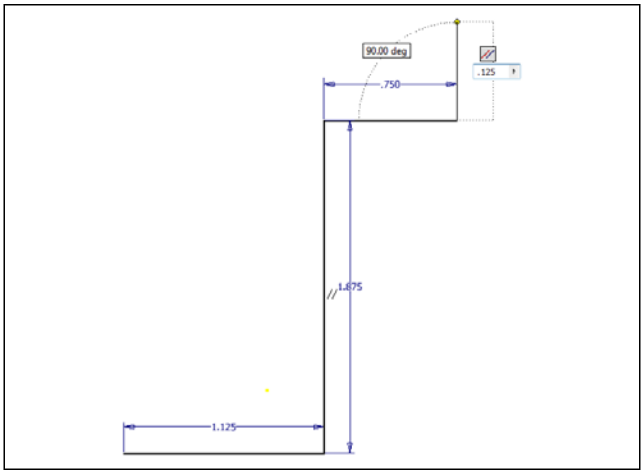


Figure 5.10 – Fourth Line Segment

We then draw a line perpendicularly to the left at 90 degrees. We will input 0.75 in the measurement textbox and press the Enter key.

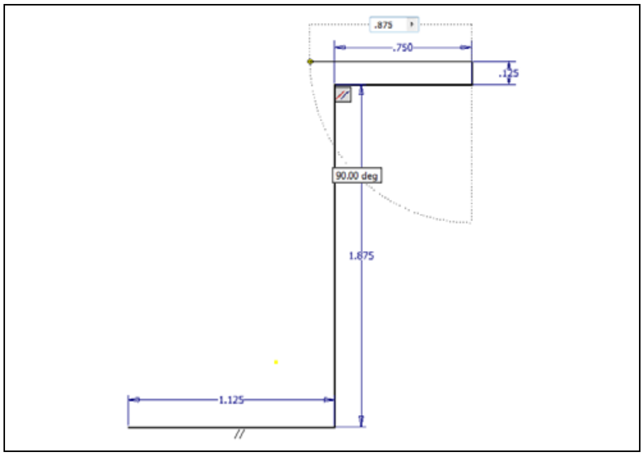


Figure 5.11 – Fifth Line Segment

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

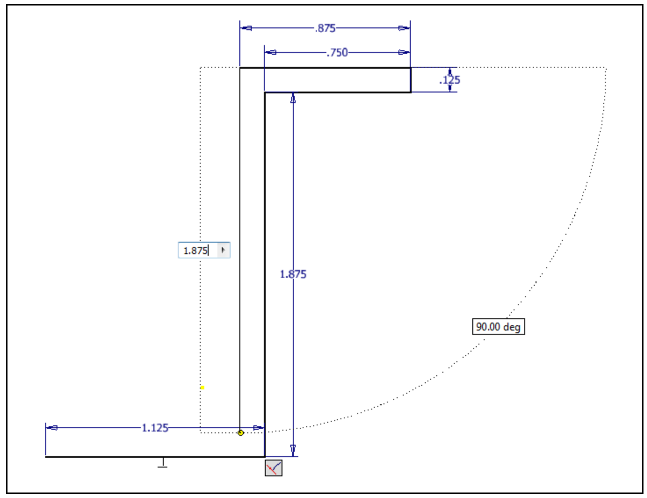


Figure 5.12 – Close the Profile

We then draw a line perpendicularly to the left at 90 degrees. We will input 1.0 in the measurement textbox and press the Enter key.

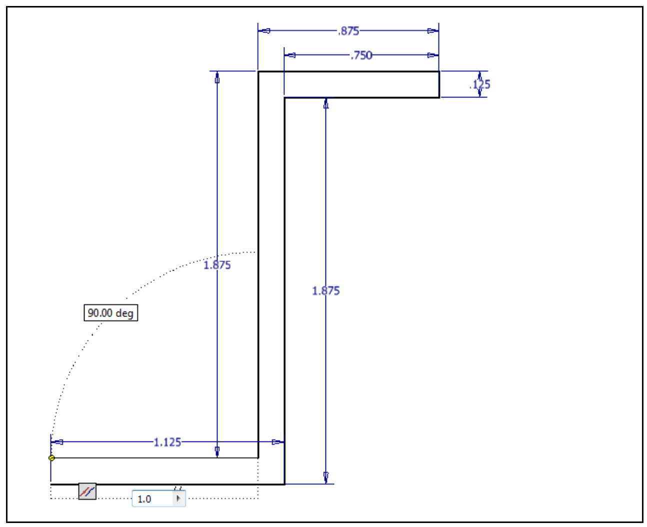


Figure 5.13 – Diagonal Line

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 5.14.

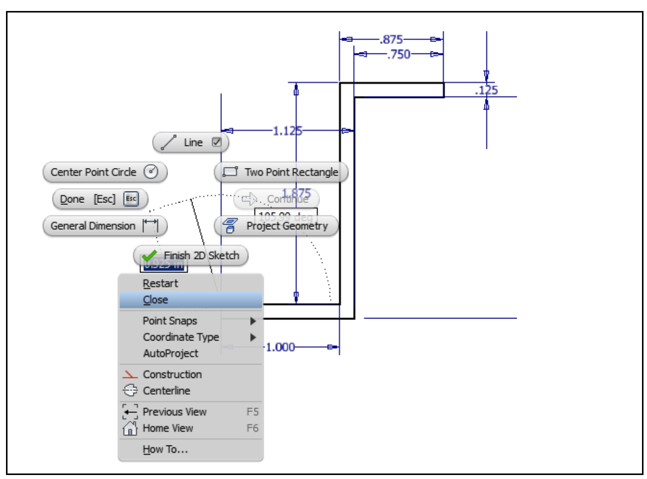


Figure 5.14 – Diagonal Line

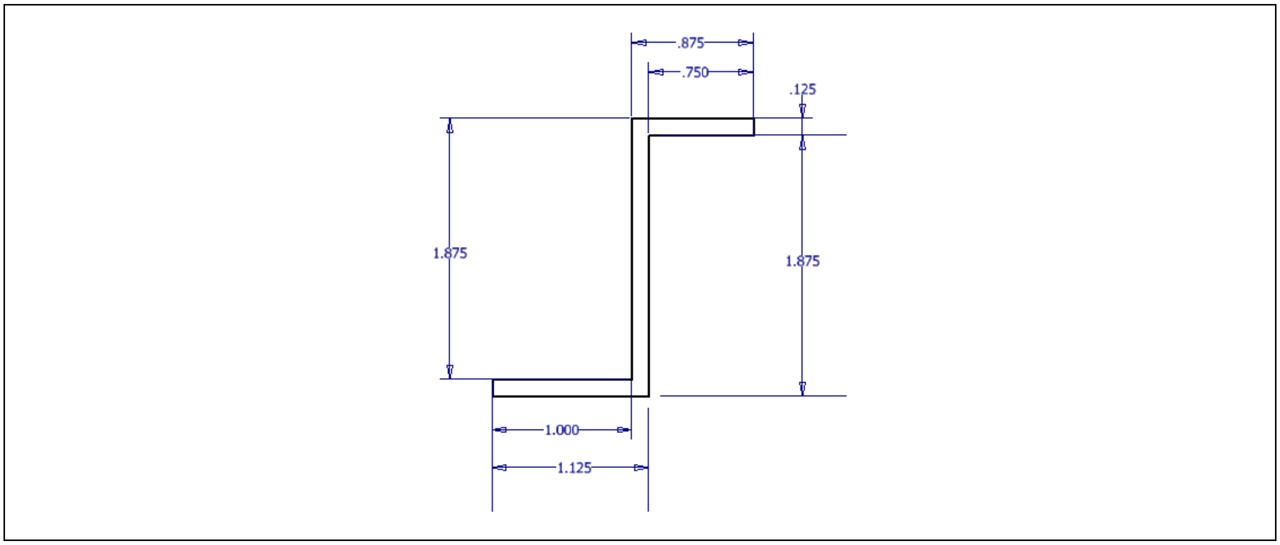


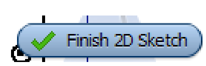
Figure 5.15 – Profile with Dimension Showing

*** World Class CAD Challenge 61-09 * - Close this drawing file. Create a New file and draw the profile of eight 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.



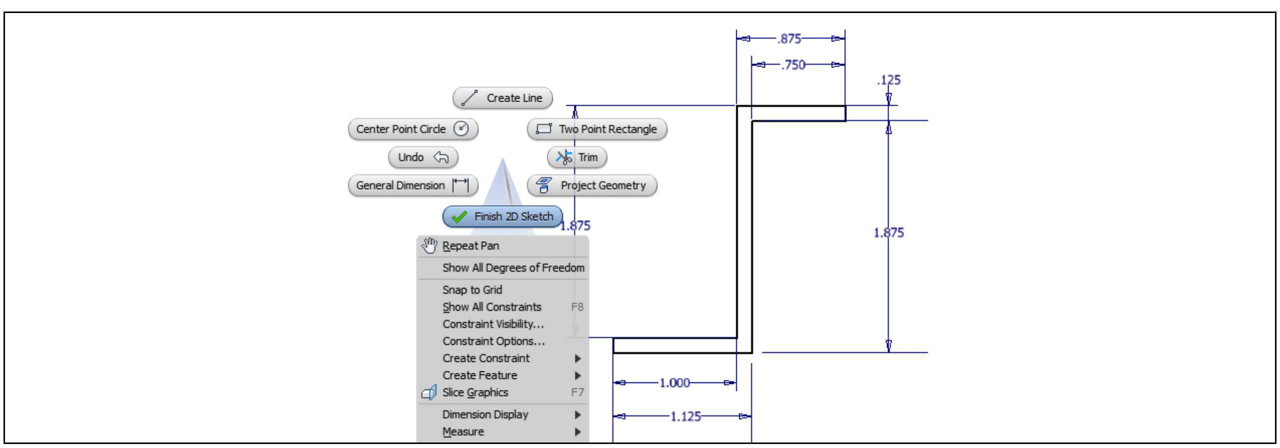


Figure 5.16– Finish 2D Sketch

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 1.5.

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

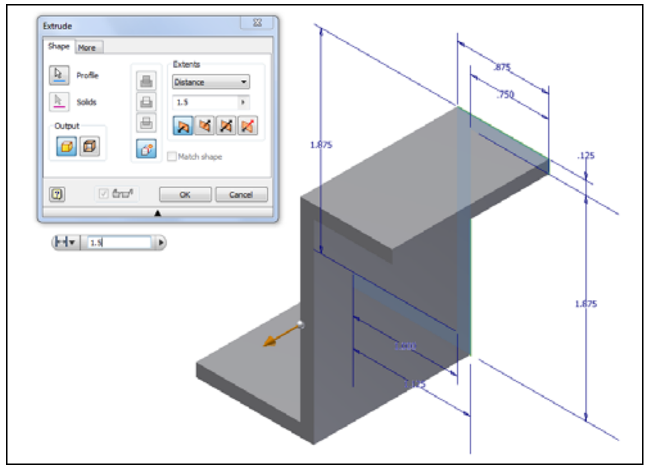


Figure 5.17 – The Extrude Window

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 0.125. When we select the straight edged corner, the pointed edge will change to a 0.125 inch rounded corner.

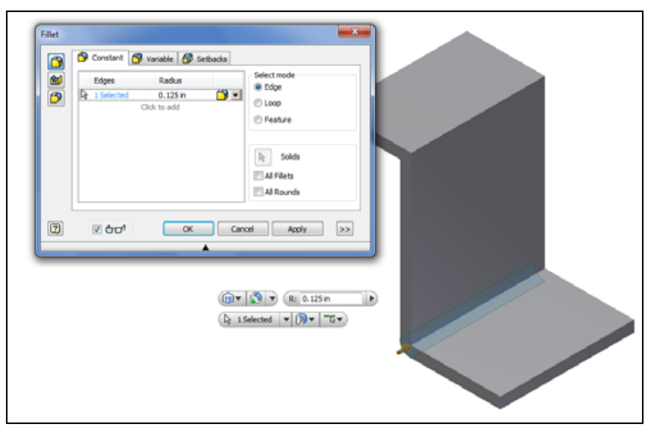


Figure 5.18 – First Fillet

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.

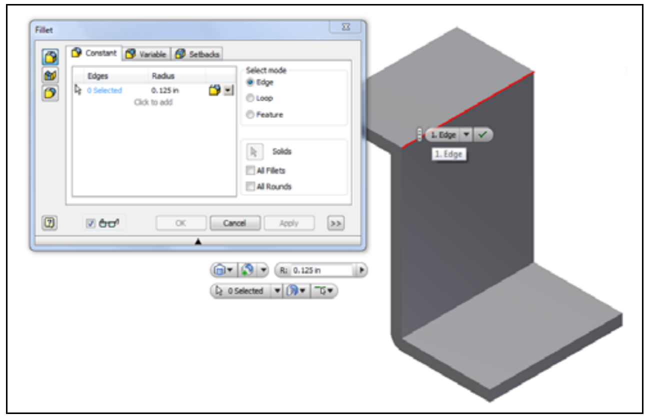


Figure 5.19 – Second Fillet

After closing the Fillet window, we are ready to add the clearance holes in the base of the bracket.

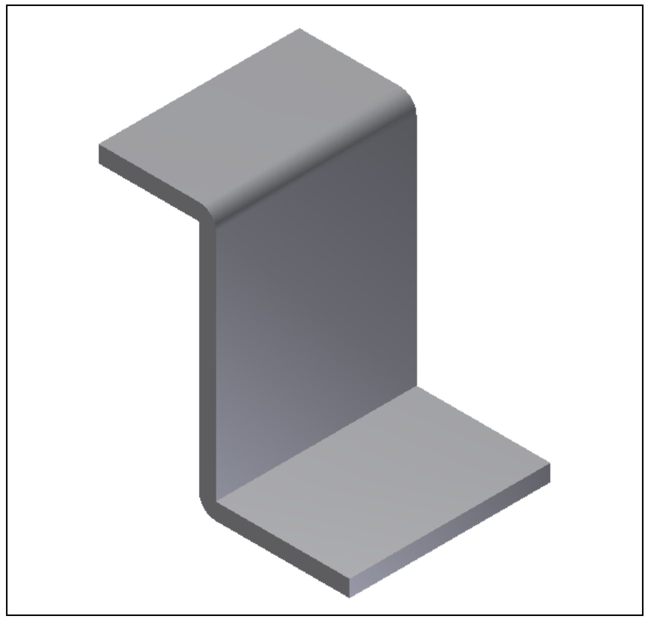
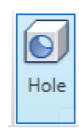


Figure 5.20 – Finished Fillet

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.

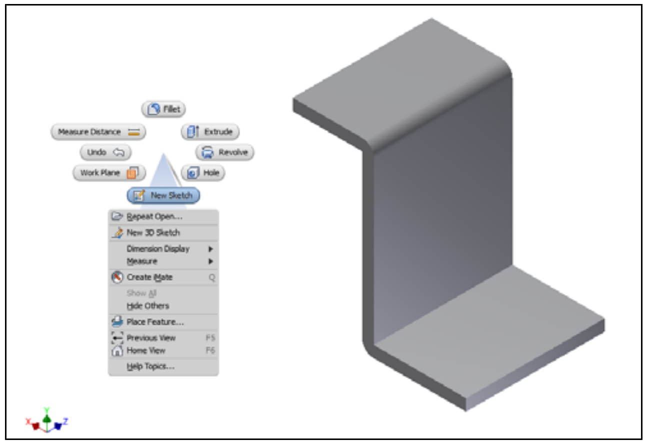


Figure 5.21 – New Sketch

We then choose the plane for the new sketch, so we pick the surface as shown in Figure 5.22.

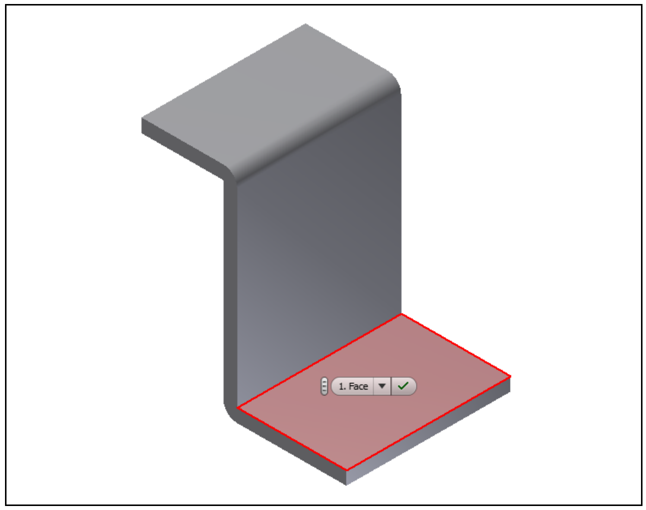


Figure 5.22 – Select the Plane

Now, we will select Point on the Sketch tab of the Inventor ribbon.

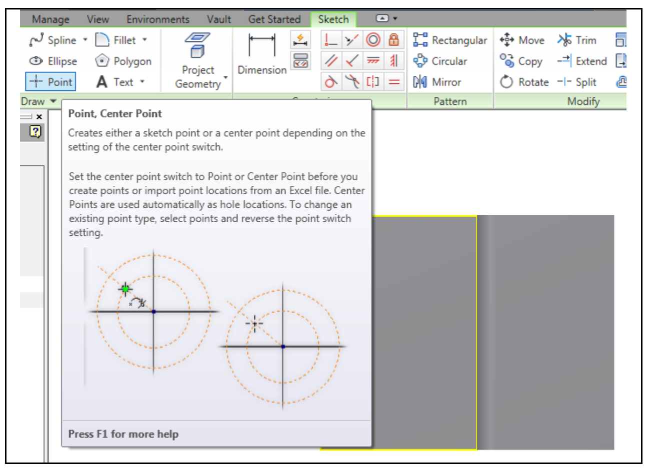


Figure 5.23 – Select Center Point

We place the cursor over the top yellow line of the plane to find the midpoint and we place a point towards the top of the plane and another directly below it. We will add dimensions to locate the centers precisely.

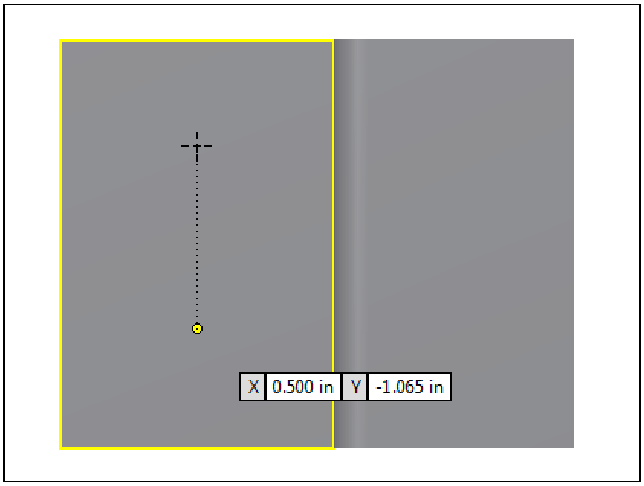


Figure 5.24 – Select the Two Centers

We choose Dimension on the Inventor ribbon and we pick the top left corner of the plane and the center of the top center point. The dimension is 0.500. We again choose Dimension on the Inventor ribbon and we pick the top left corner of the plane on the center of the top center point. We will edit the dimension and input 0.25.

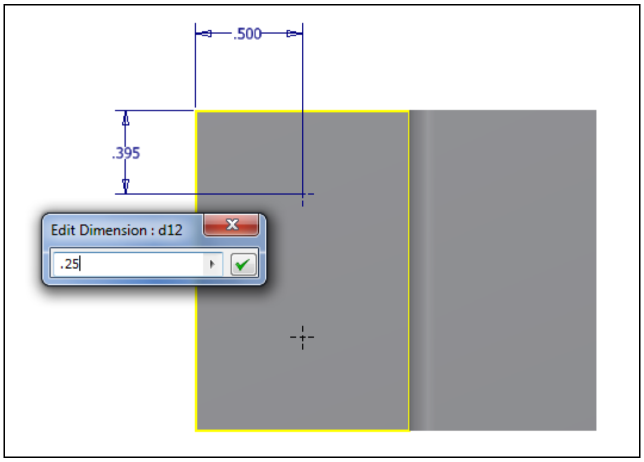


Figure 5.25 – Dimension the Top Center

For the third time, we select Dimension on the Inventor ribbon and we pick the center of the top center point and the center of the bottom center point. We will edit the dimension and input 1.0.

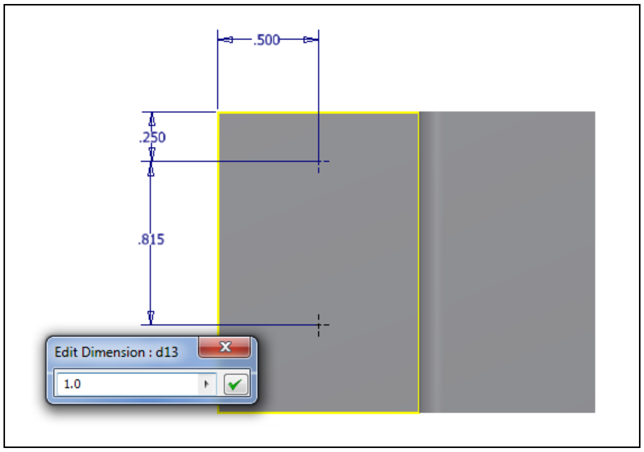


Figure 5.26 – Dimension the Bottom Center

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.

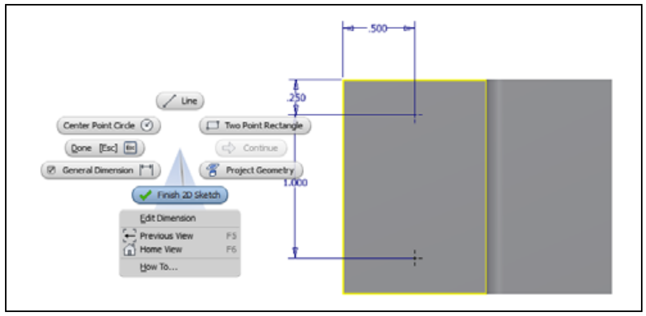


Figure 5.27 – Finish the 2D Sketch

We then select the Hole button on the Inventor ribbon and the Hole window will appear on the graphical display. We are making a 0.201 through hole, so we change the diameter textbox from 0.25 to 0.201. The two hole placements appear automatically and we press the OK button to retain the feature.

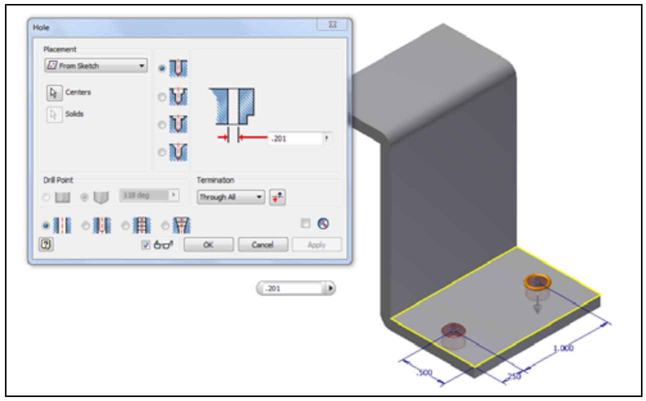


Figure 5.28 – Add Holes to the Centers

Drawing a Slotted Hole

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

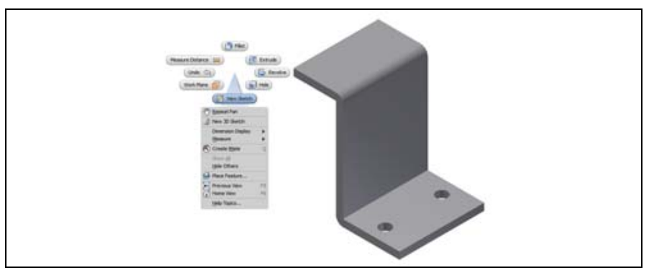


Figure 5.29 – Another New Sketch

We then choose the plane for the new sketch, so we pick the surface as shown in Figure 5.30.

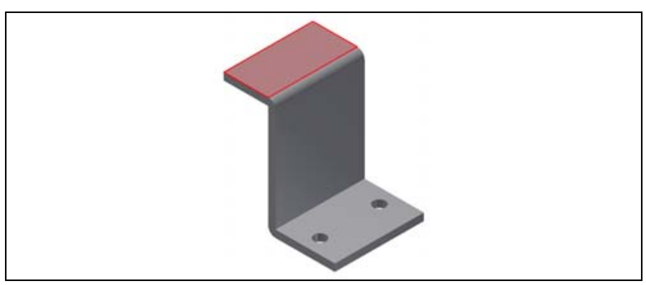


Figure 5.30 – Select the Plane

We want to draw a rectangle on the plane, so we right click on the drawing and we can see Create Line, Center Point Circle, Two Point Rectangle and many more choices. We pick Two Point Rectangle and we will single click on the yellow right edge of the plane and another point on the middle of the plane.

This any sized rectangle on the plane has two dimensions. The horizontal measurement is highlighted and we can type 0.25. We press the tab on the keyboard to switch to the vertical dimension and we input 1.0.

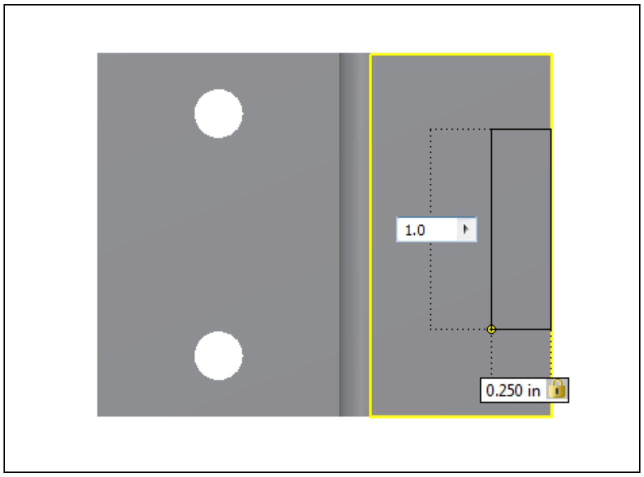


Figure 5.31 – Add a Rectangle

We choose Dimension on the Inventor ribbon and we pick the top right corner of the plane and the top right corner of the rectangle. The dimension is 0.25.

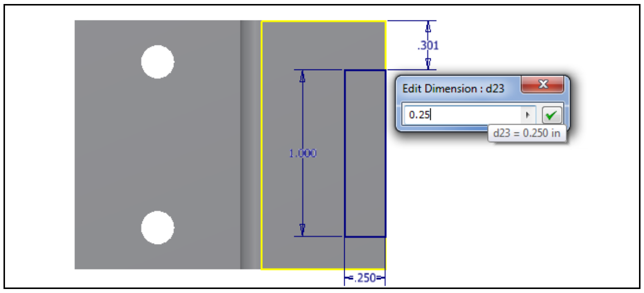


Figure 5.32 – Add a Dimension

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.

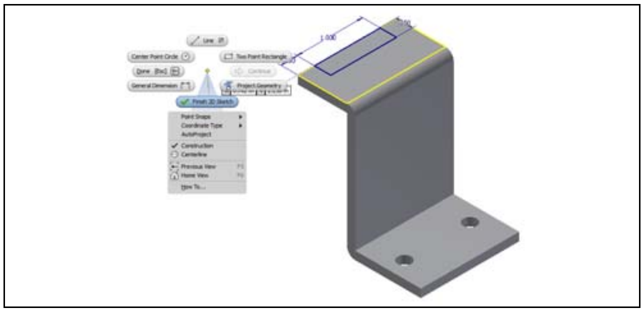


Figure 5.33 – Finish the 2D Sketch

Now that we have a finished sketch, we need to extrude the slot. 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. Then we will click on the slot and it will become highlighted in red.

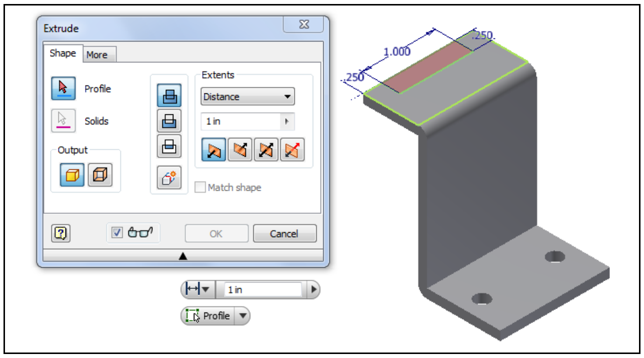


Figure 5.34 – Select the Extrusion Area

Click on the direction icon so that the new solid goes down into the bracket. Click on the red area again and a rectangular slot will appear in the top of the bracket.

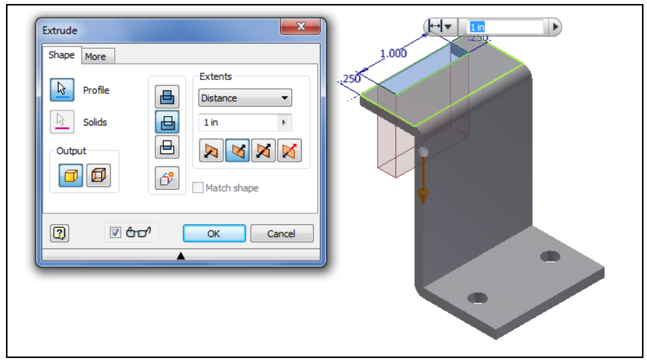


Figure 5.35 – Extrude the Slot

The next feature we will add to our slot is two 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 0.25. When we select the straight edged corner inside the rectangular slot, the pointed edge will change to a 0.25 inch rounded corner.

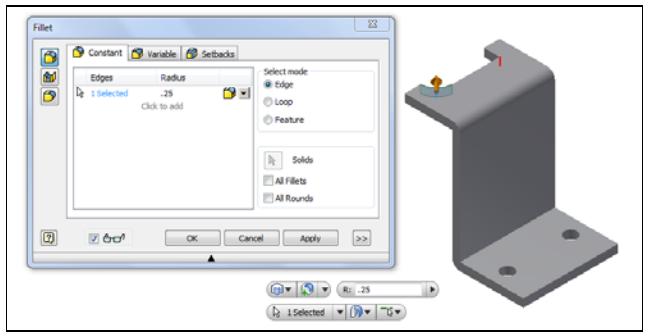


Figure 5.36 – Add Two Fillets

Drawing a Tapped Hole

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

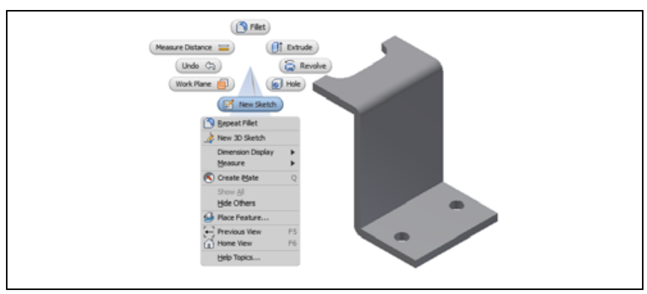


Figure 5.37 – Start a New 2D Sketch

We then choose the plane for the new sketch, so we pick the surface as shown in Figure 5.38.

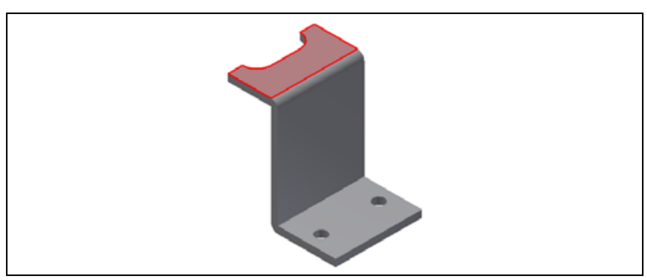


Figure 5.38 – Select the Plane

Now, we will select Point on the Sketch tab of the Inventor ribbon. Then we place the cursor over the left yellow line of the plane to find the midpoint and we place a point towards the middle of the plane. We will add dimensions to locate the centers precisely.

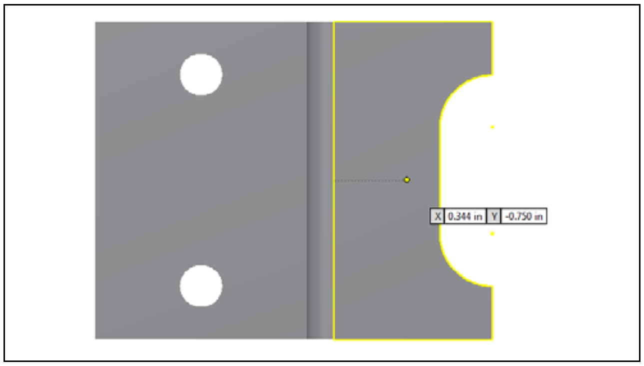


Figure 5.39– Add a Center Point

We choose Dimension on the Inventor ribbon and we pick the top right corner of the plane and the center of the center point. The dimension is 0.4375, so we will edit the dimension and input 0.4375.

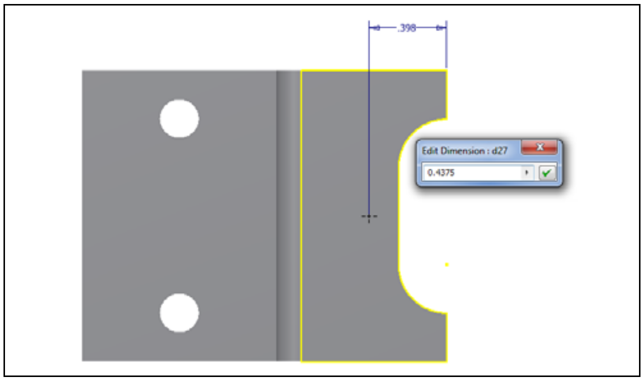


Figure 5.40 – Dimension the Center Point

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.

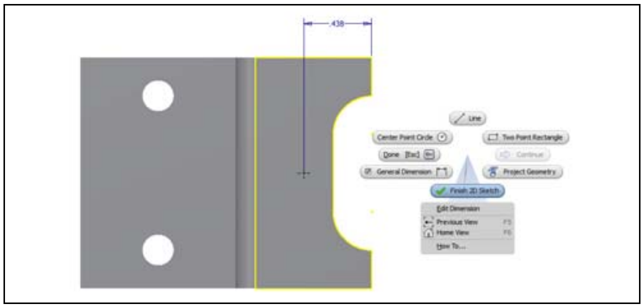


Figure 5.41 – Finish the 2D Sketch

We then select the Hole button on the Inventor ribbon and the Hole window will appear on the graphical display. We are making a 6-32 tapped hole, so we choose the tapped hole icon. We pick ANSI Unified Screw Threads, 0.138 (#6) for the size, 2B for the class and 6-32 UNC for the designation. The hole that automatically appeared will appear tapped and we press the OK button to retain the feature.

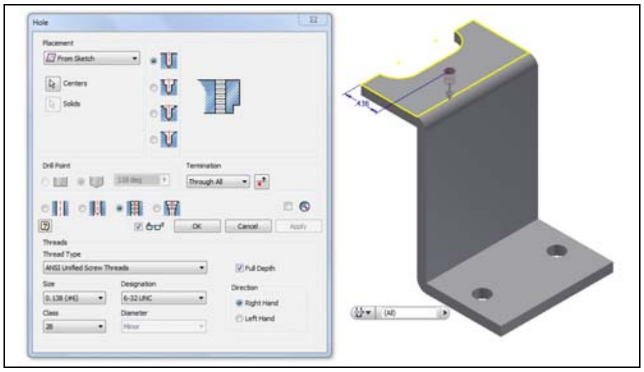


Figure 5.42 – Add a Tapped Hole

Mirror a Solid

The next function we will learn to draw in Inventor is mirror. We choose the Mirror button on the Inventor ribbon and the Mirror window will appear on the graphical display.

We select the Solid icon and then the Mirror Plane icon. We pick the plane as shown in figure 5.43.

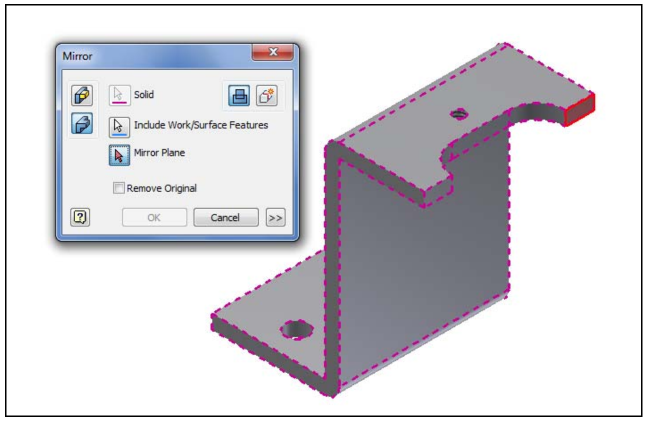


Figure 5.43 – Select the Mirror Plane

The other half of the solid will appear as a wireframe. We press the OK button to make the complete bracket.

We can change the view to see the hidden lines, so we go to View on the Inventor ribbon and we choose Visual Style and pick Shaded with Hidden Edges. The drawing appears as shown in figure

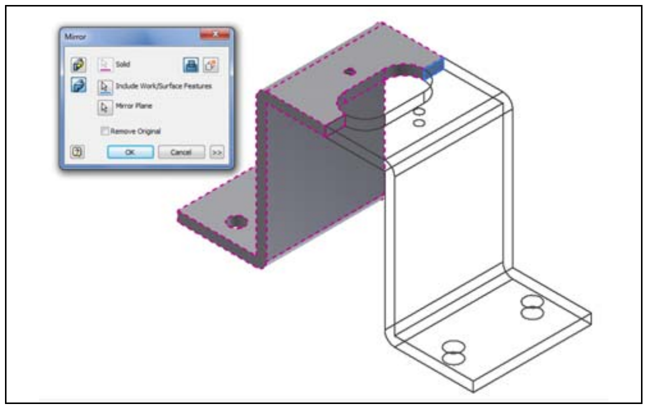


Figure 5.44 – The Mirrored Solid

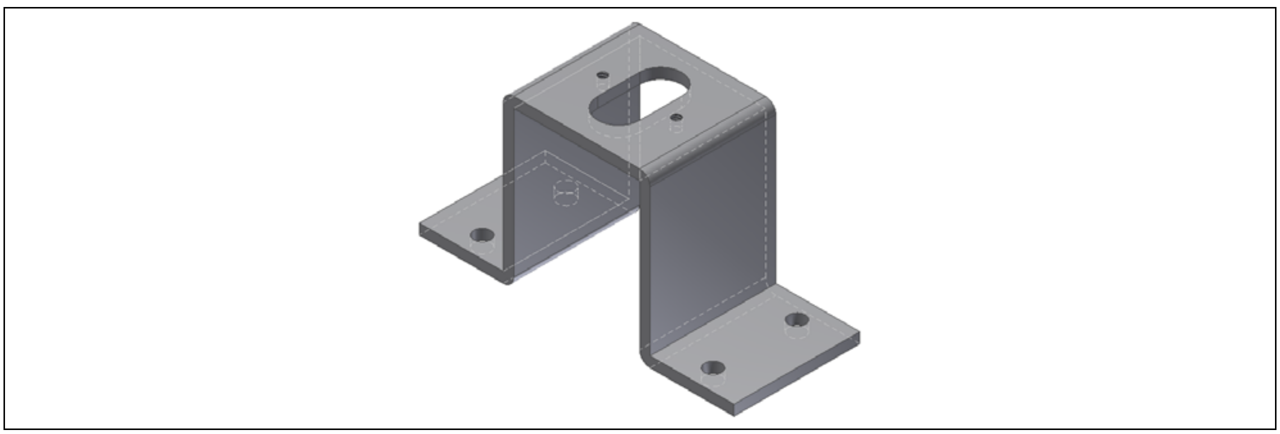


Figure 5.45 – The Finished Solid

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

*** World Class CAD Challenge 61-10 * - Close this drawing file. Create a New file and draw the profile of eight lines. Add two holes on the bottom plane. Add a slotted hole and a tapped hole on the top plane. Mirror the solid. 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.**