

Chapter

7

Solid Part Six

This chapter will cover the following to World Class standards:

- **Sketch of Solid Problem**
- **Draw the Profile**
- **Add Multiple Fillets**
- **Finish the 2D Sketch**
- **Revolve a 2D Sketch**
- **Create Another Sketch Off a Projected Geometry**
- **Offset Lines on a Sketch**
- **Extrude another 2D Sketch**
- **Add Addition Fillets on a Solid**

Sketch of a Solid Part

In this project, we will make a 3D cup with a handle. We start this project by making two profiles. After making the cup profile, we will revolve the shape 360 degrees. Then, we will make a handle profile, extrude and fillet it. In this problem, we will notice that handle profile will extend into the body of the cup, because the revolve section of the cup needs to mate up with the entire depth of the extrusion. When we combine two solids, one that is curved and the other that is straight, we want them to overlap.

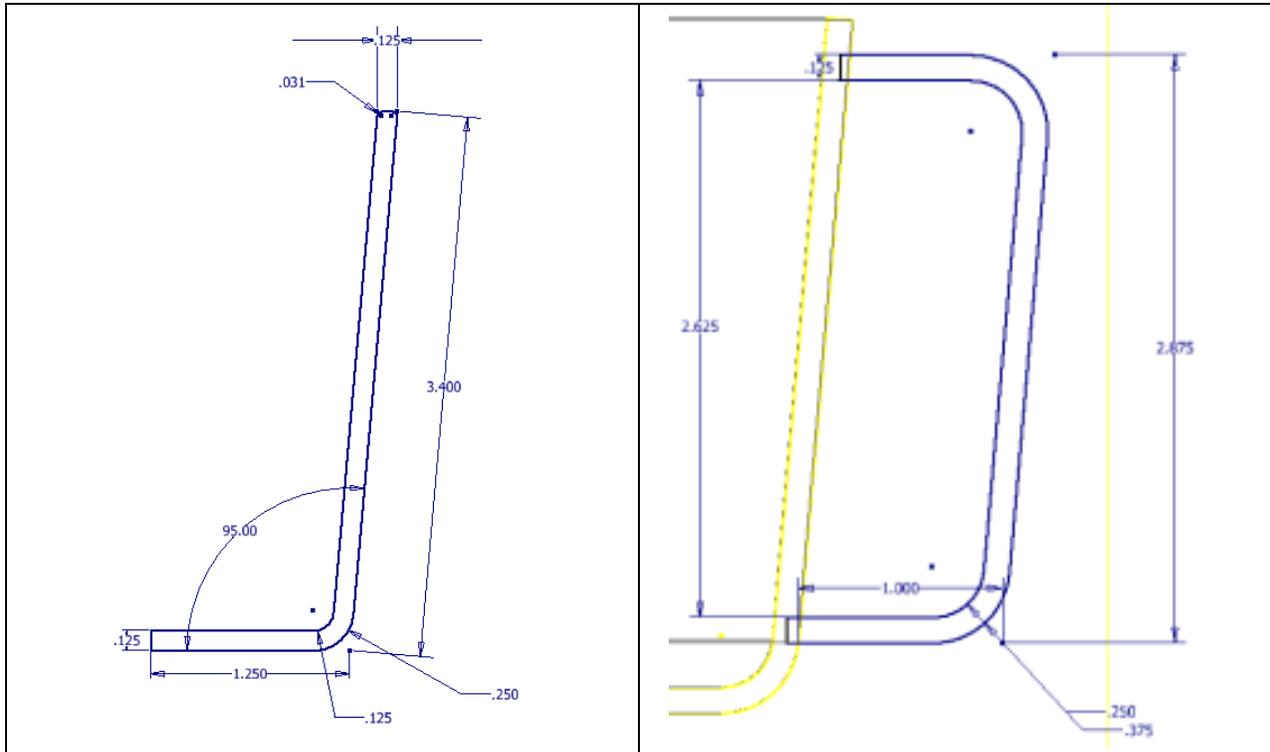


Figure 7.1 – Problem Six Sketch

In this problem, we will practice techniques that we learned in the previous solid parts and add some new experiences such as offset and viewing a solid with hidden lines.

Starting a 3D Part Drawing Sketch

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

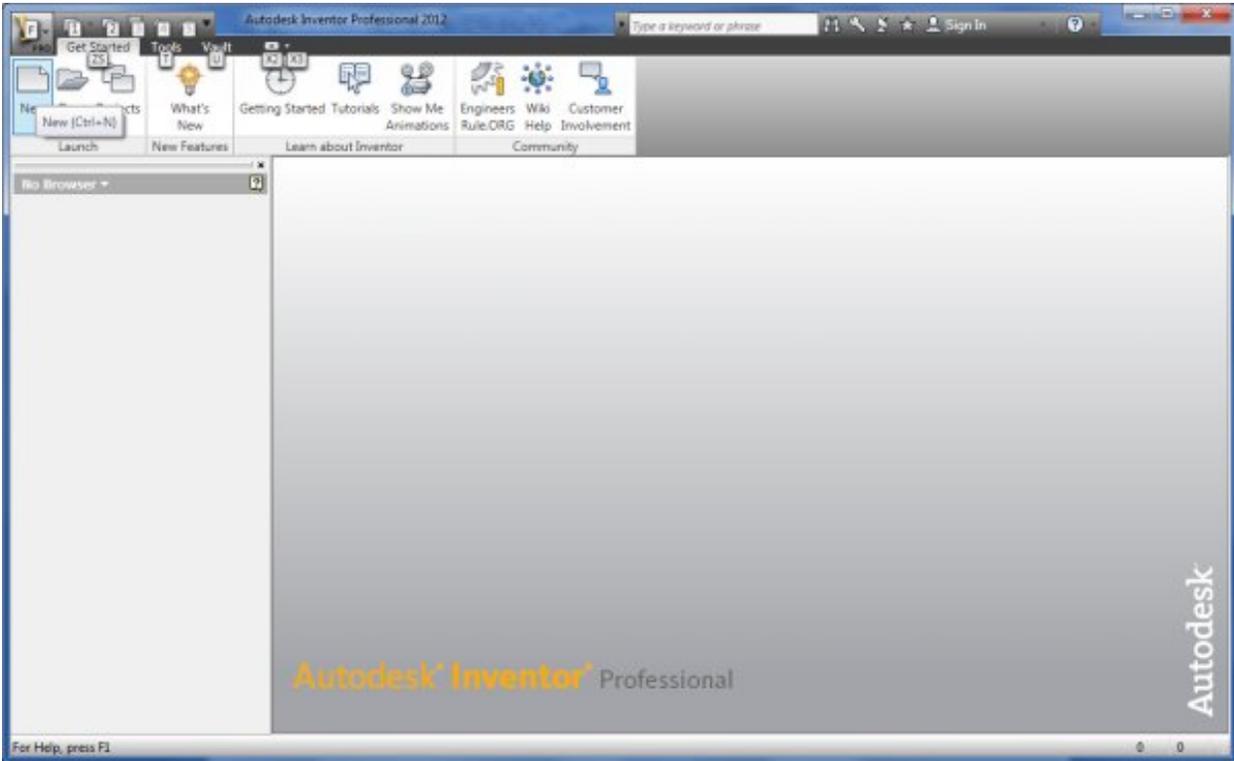


Figure 7.2 – AutoCAD Inventor Professional 2012

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.

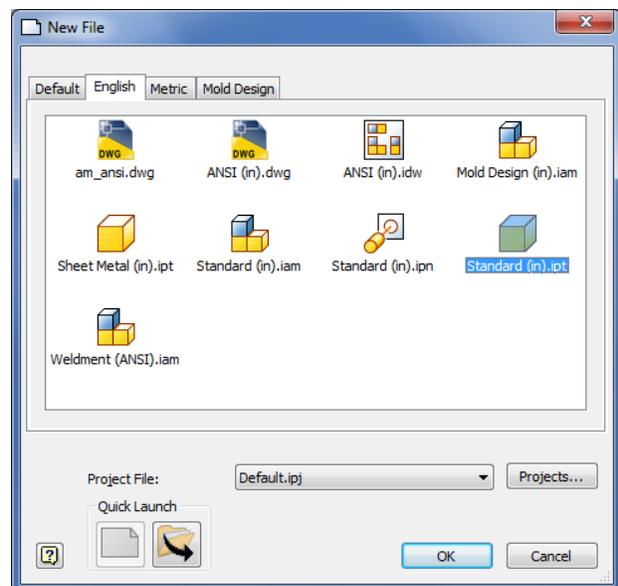


Figure 7.3 – Starting the drawing using the Standard IPT template

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.

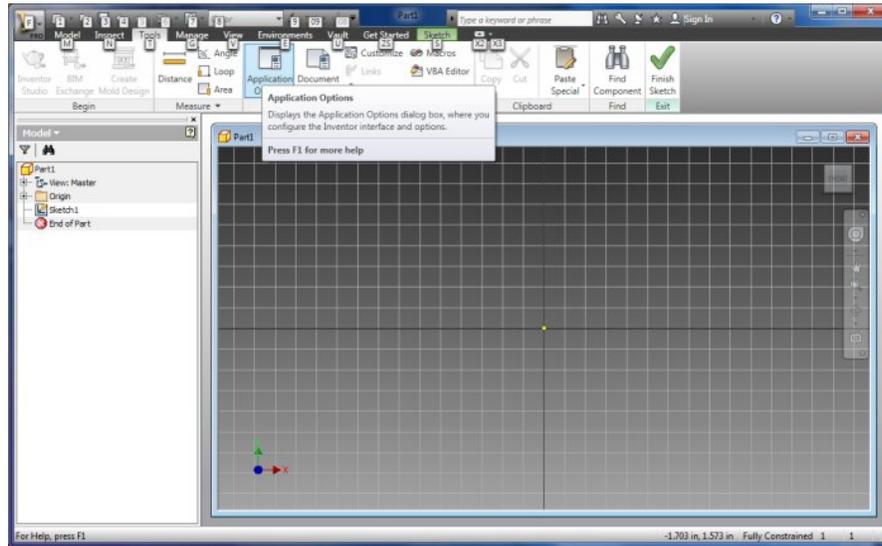


Figure 7.4 – Starting the drawing using the Standard IPT template

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.

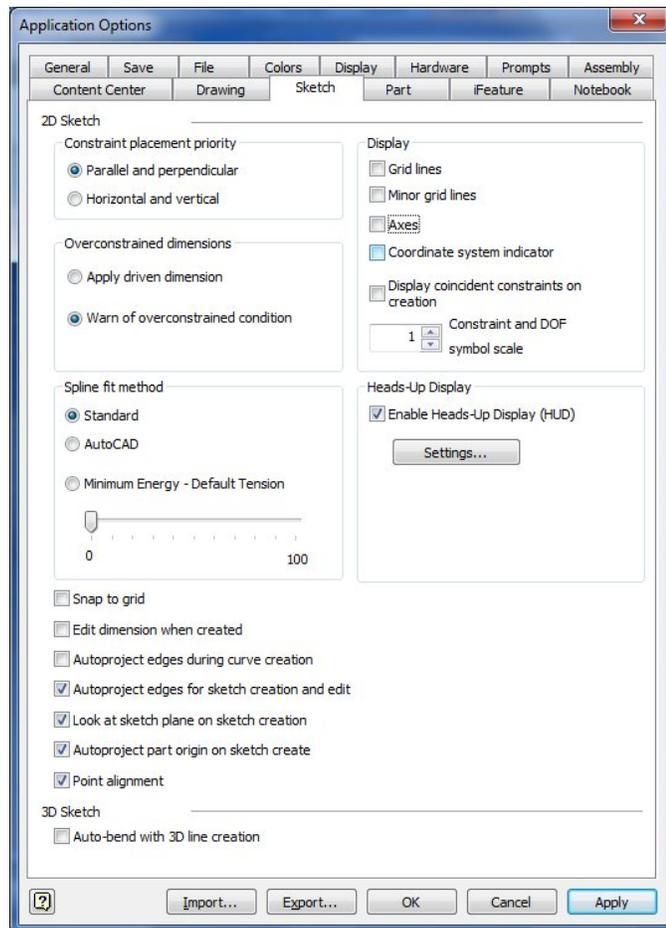


Figure 7.5 – Application Options Window

Drawing the Profile of the Casting

We will begin our sketch by drawing a series of lines. We right click on the drawing and we can see Create Line above and to the left of Two Point Rectangle.



We pick Create Line and we select the first point in the center of the display at the X equals 0 and Y equals 0 point. By using the origin 0,0, we can reference the middle of the part later in the development process. We draw a line horizontally to the right and input 1.25 in the measurement textbox.



Figure 7.6 – Graphical Display Menu

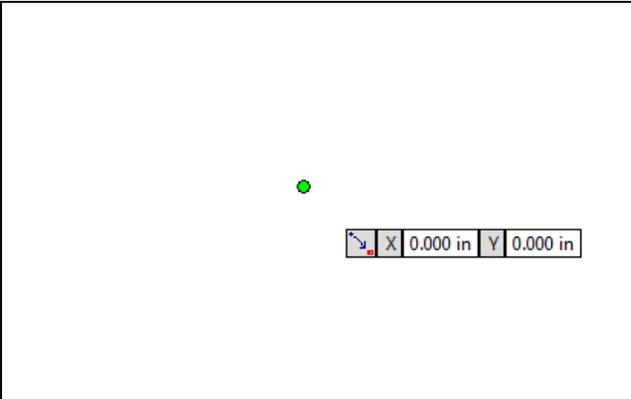


Figure 7.7 – Pick a Center Point

We draw the next line at angle of 95 degrees and with a distance of 3.4 inches.

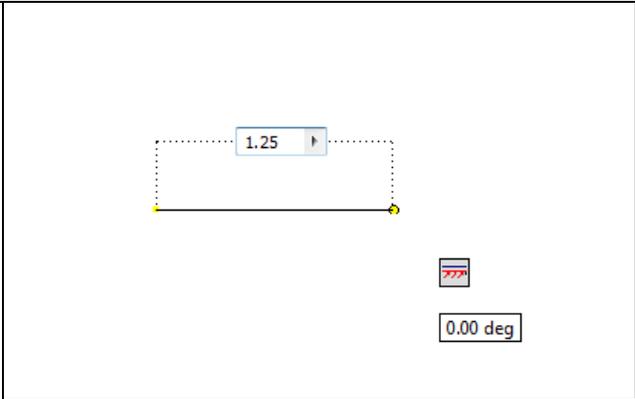


Figure 7.8 – Enter the Dimension

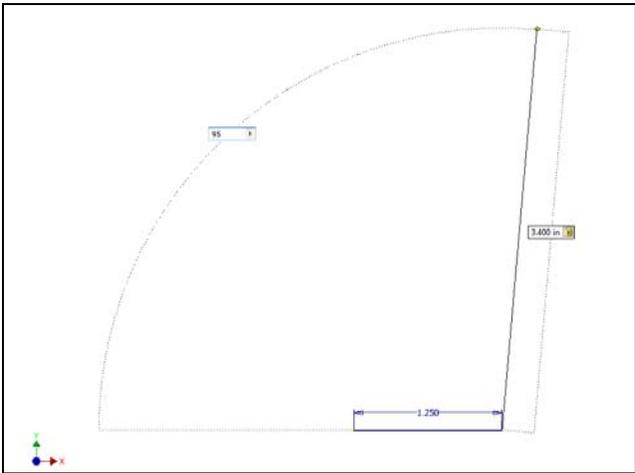


Figure 7.9 – Second Line of the Profile

Then we draw the third line at angle of 90 degrees off of the previous line and with a distance of 0.125 inches.

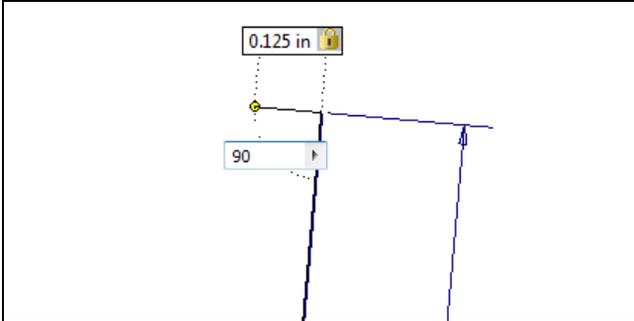


Figure 7.10 – Third Line of the Profile

Then we draw the fourth line of the profile at angle of 90 degrees off of the previous line and with a distance of 3.0 inches. Later, we will extend this entity when we fillet this line with another. We can notice the two slash line icon (//) that appears when our new line is parallel to the one next to it.

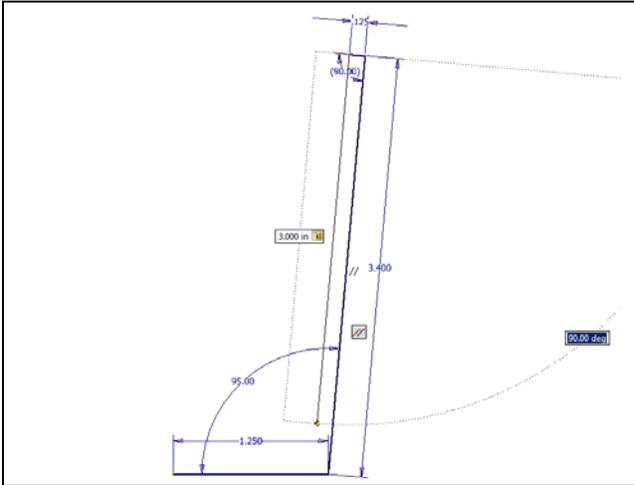


Figure 7.11 – Fourth Line of the Profile

We will Enter to end the continuous line.

Next, we begin a new line from the 0,0 coordinated and draw the fifth line at angle of 90 degrees off of the previous line and with a distance of 0.125 inches.

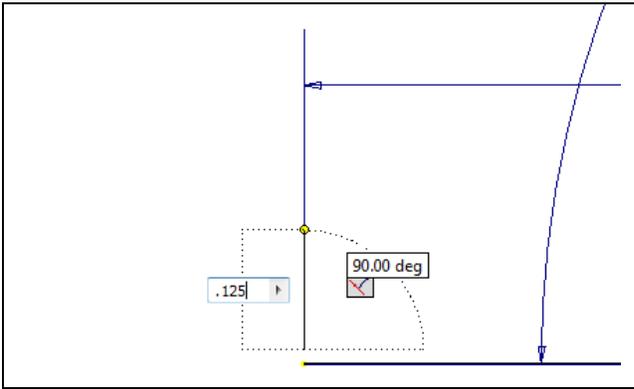


Figure 7.12 – Fifth Line of the Profile

We draw the next line at angle of 90 degrees to the right and with a distance of 1.0 inches.

We will Enter to end the continuous line.

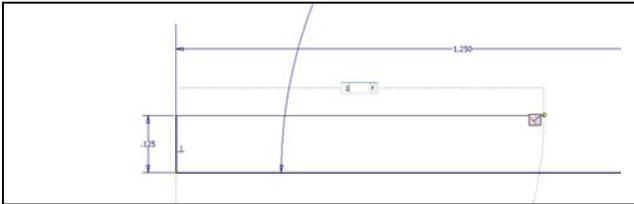


Figure 7.13 – Sixth Line of the Profile

Add Multiple Fillets

The next feature we will add to our sketch is the 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 for the inside arc. We will select the 1.0 and 3.0 lines as shown in the figure and the two lines will extend and a 0.25 radius arc will appear at the intersection of the two lines..

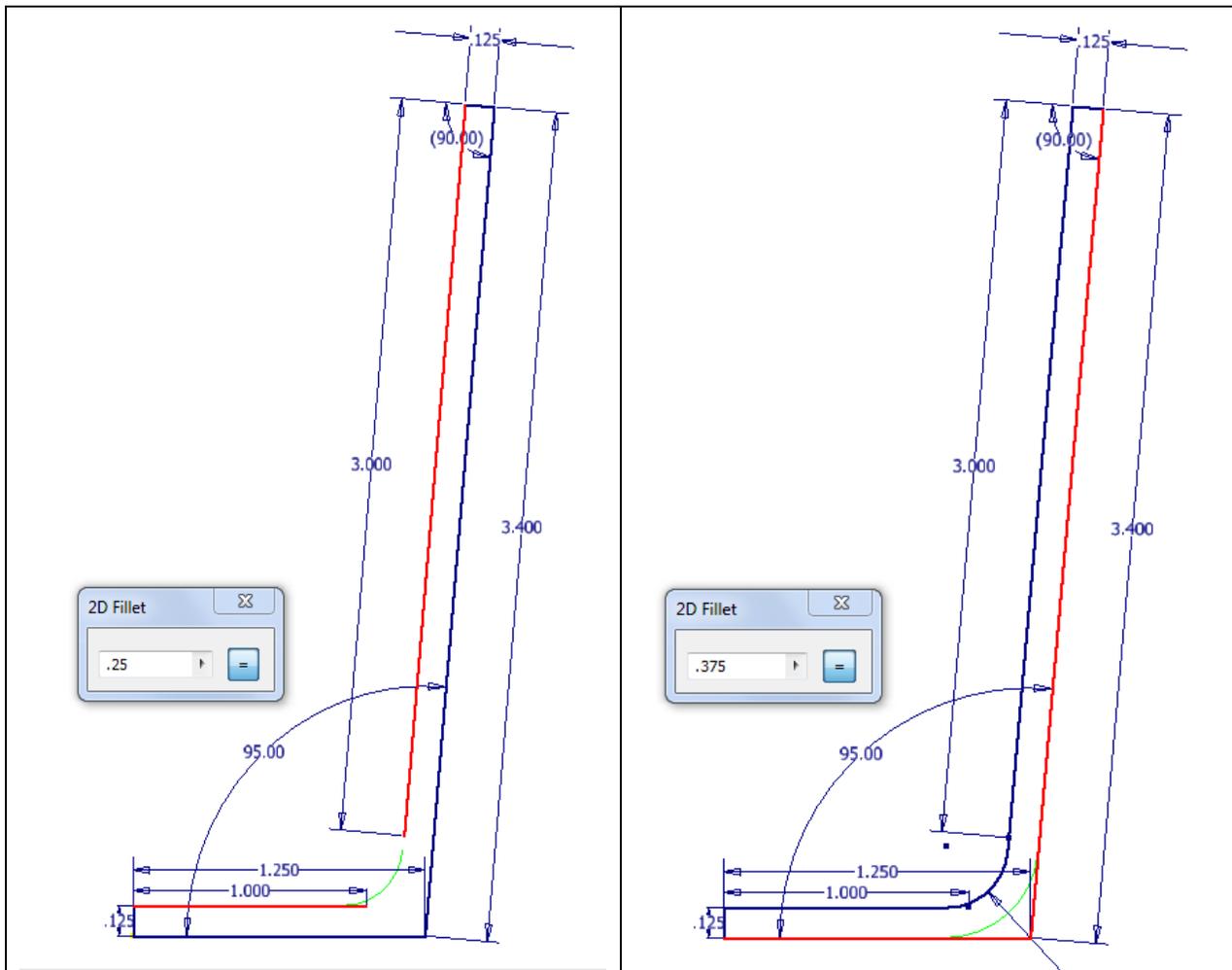


Figure 7.14 – Inside Arc

Figure 7.15 – Outside Arc

We set the fillet radius to 0.375 for the outside arc. We will select the 1.25 and 3.4 lines as shown in the figure and a 0.375 arc will appear where there was a sharp pointed junction.

Save the drawing and we will now make a model off the sketch. To improve our sketching abilities, we should practice these drills until we know each function thoroughly.

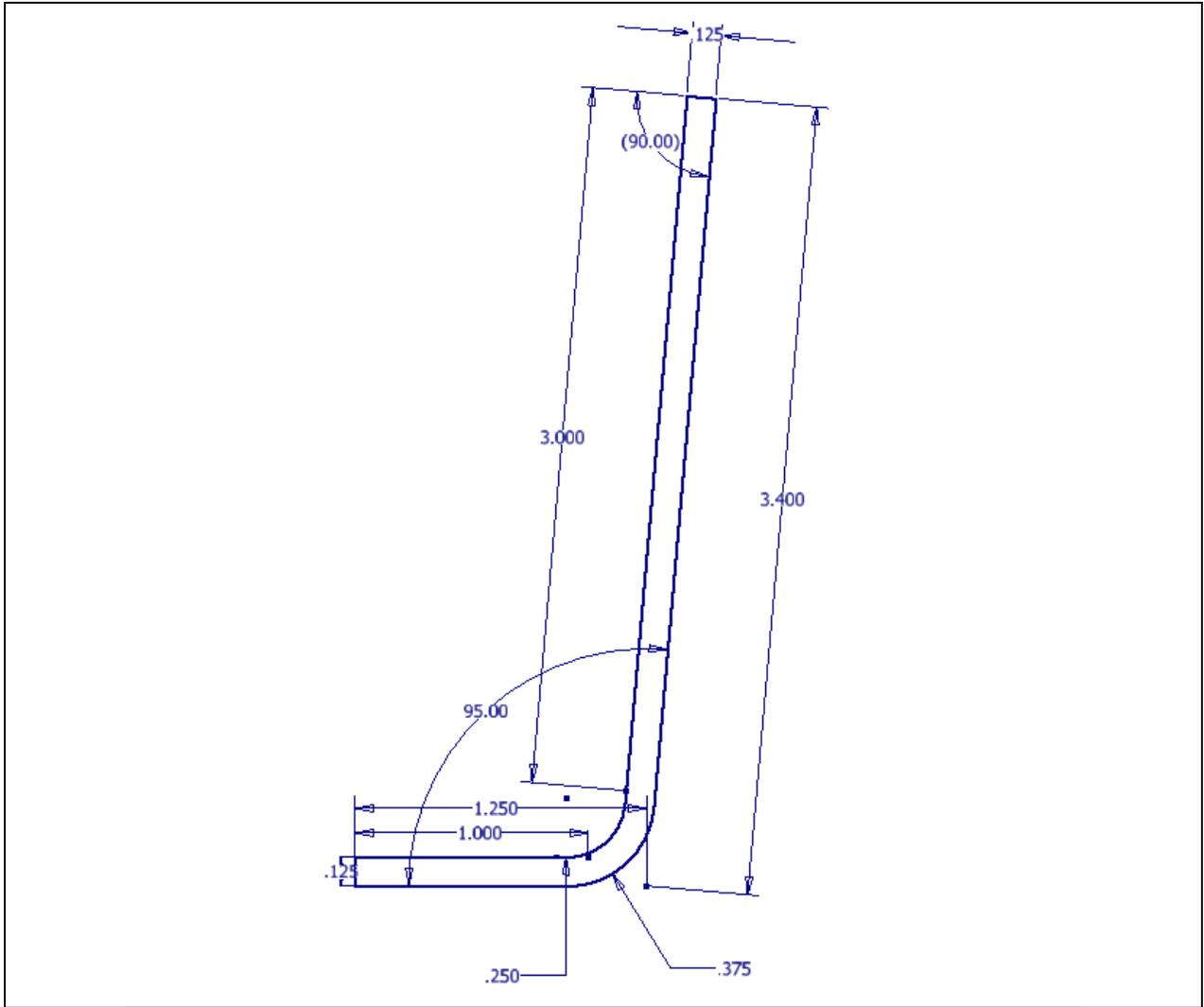


Figure 7.16 – The Finished Sketch

*** World Class CAD Challenge 61-12 * - Close this drawing file. Create a New file and draw the profile with 6 lines and two arcs. 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.

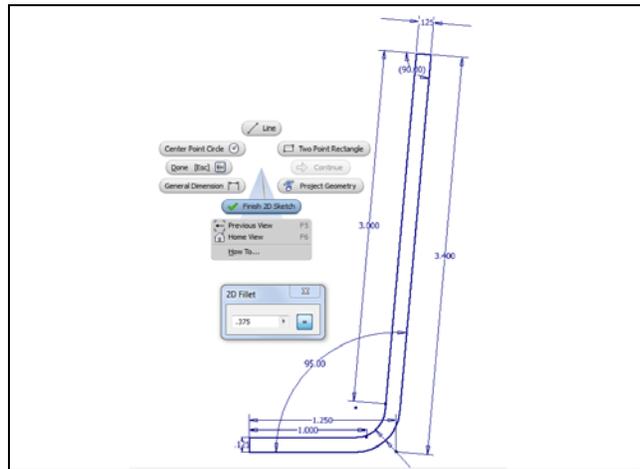


Figure 7.17 – Finished Part Two Sketch

Revolving a 3D Sketch

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



On the Revolve window, we choose solid for the Output and we select the Profile icon and we can choose the shape we just made.

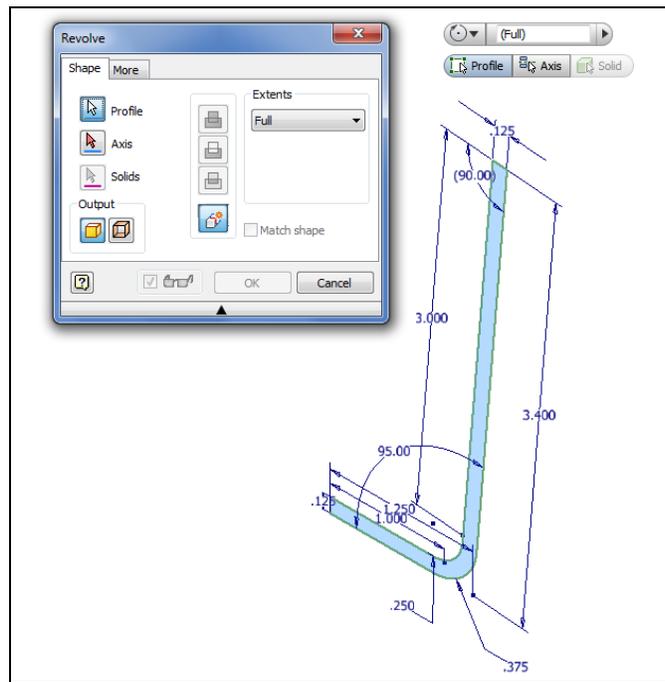


Figure 7.18 – Revolve Window

We will revolve it to the full extents or 360 degrees.

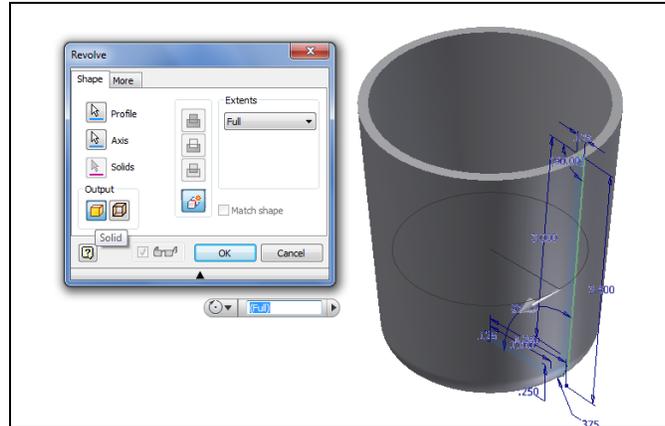


Figure 7.19 – Revolve the Profile 360 Degrees

Create Another Sketch Off Projected Geometry

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.

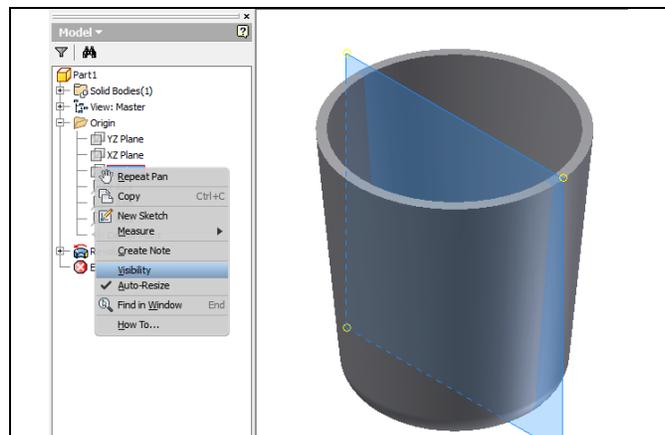


Figure 7.20 – Turn On XY Plane Visibility

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

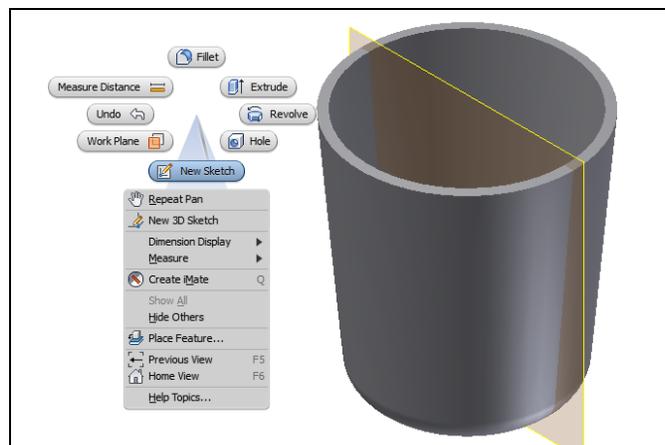


Figure 7.21 – Start a New Sketch

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.

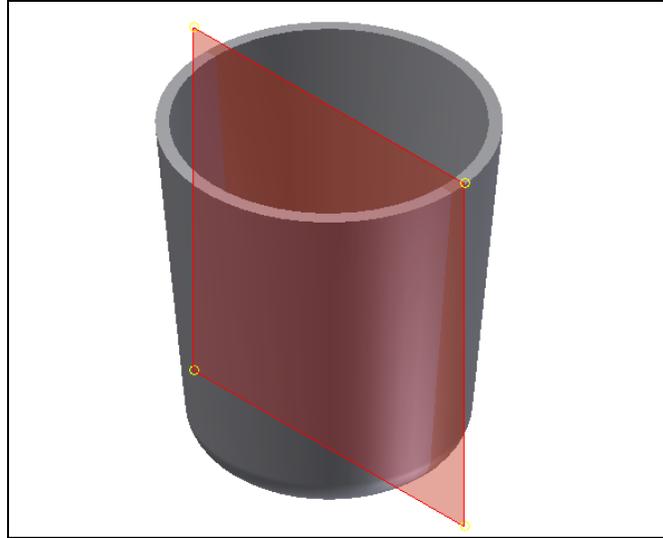


Figure 7.22 – Select the X-Y Plane

We will work off the original profile so we go to View on the Inventor ribbon and we choose Visual Style and Wireframe.

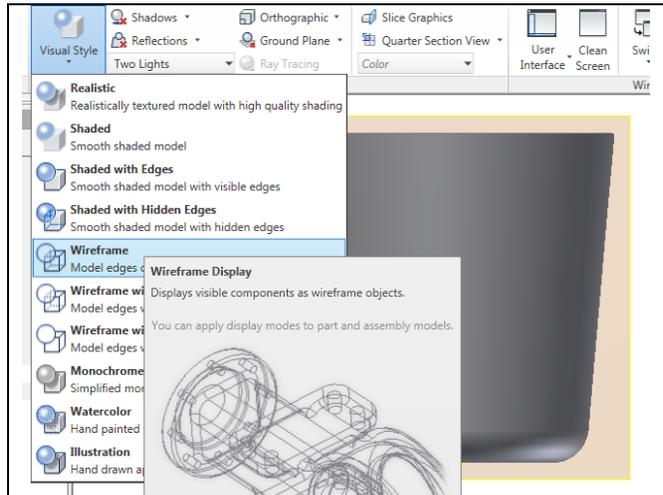


Figure 7.23 – Change the View to Wireframe

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

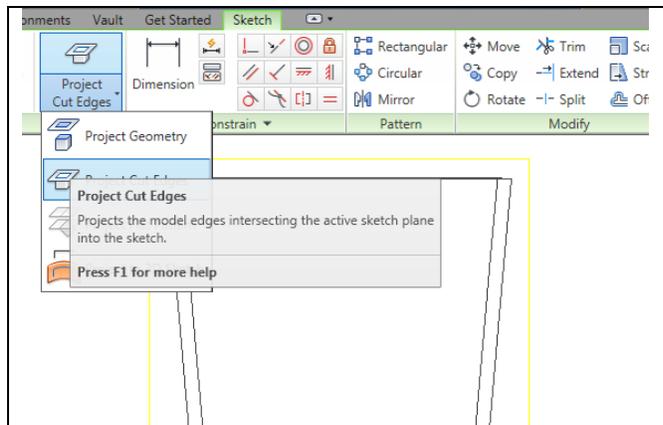


Figure 7.24 – Project Cut Edges

Then we draw a line starting at the green dot where the 0.375 radius arc and line intersect.

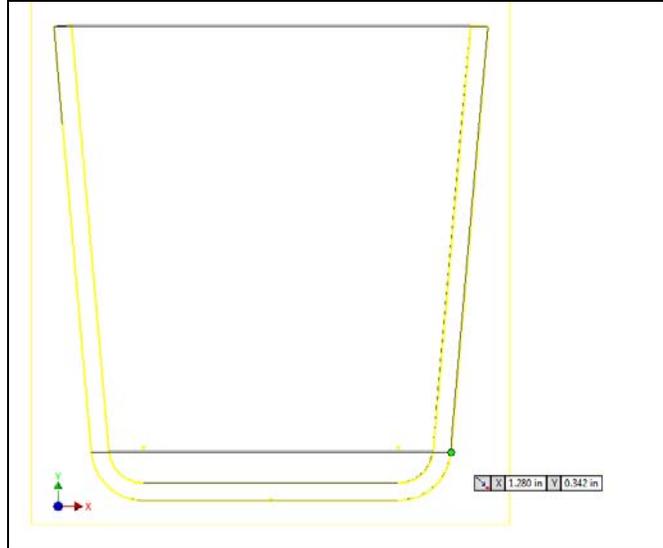


Figure 7.25 – Draw a Line Starting at a Point

We draw a line horizontally to the right and input 1.0 in the measurement textbox.

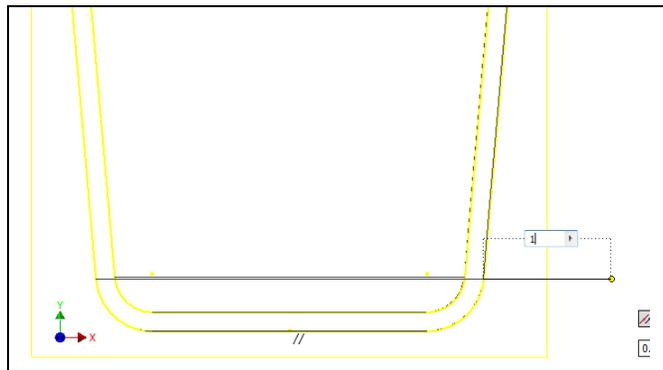


Figure 7.26 – 1.0 Measurement

Then we draw another line of the handle profile at angle of 95 degrees off of the previous line and with a distance of 2.0 inches. Later, we will extend this entity when we change the dimension.

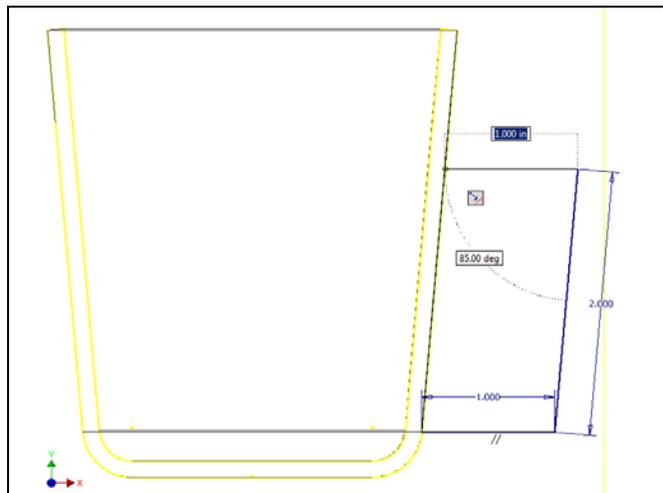


Figure 7.27 – Draw a Parallel Line Upward

Then we draw another line of the handle profile at angle of 95 degrees off of the previous line and with a distance of 2.0 inches. Later, we will extend this entity when we change the dimension.

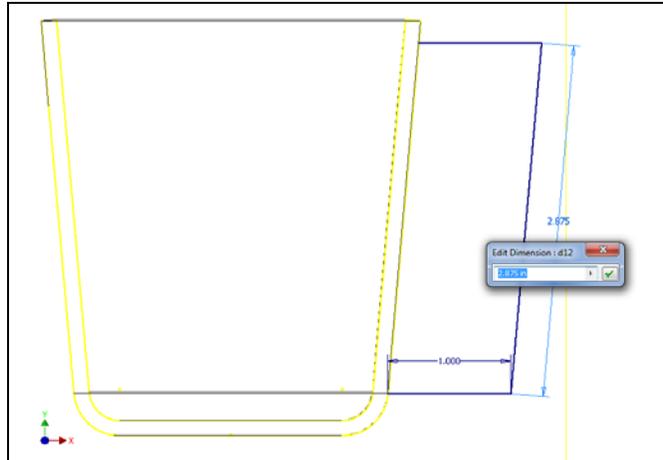


Figure 7.28 – Change Dimension

Offset Lines on a Sketch

We select the Offset icon on the Inventor ribbons.

We then select the three lines we have on the sketch for the handle and we offset them to the inside of the handle we will add dimensions to make the thickness accurate.

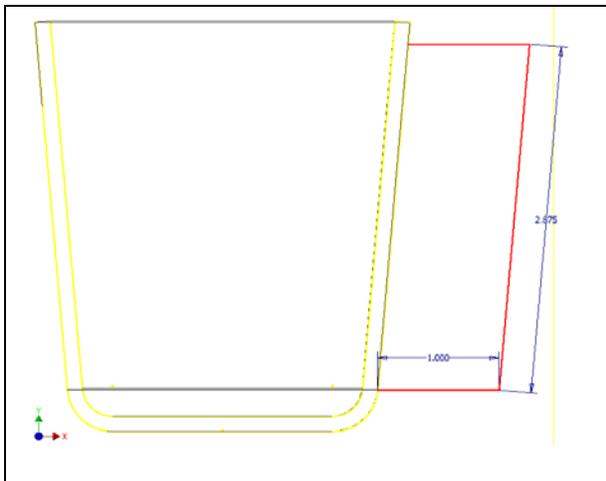


Figure 7.29 – Select Lines to Offset

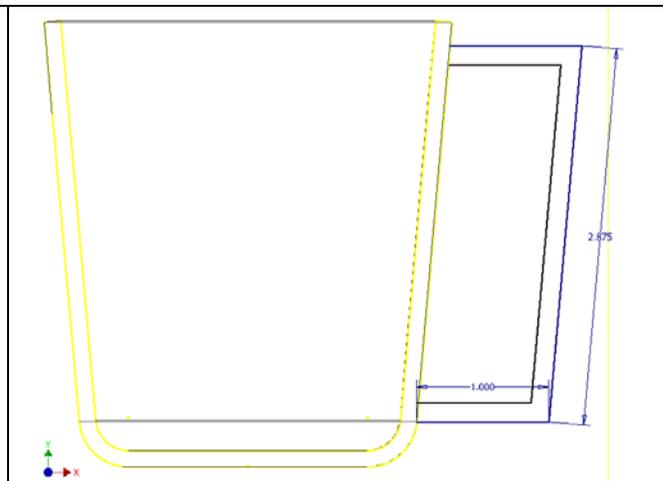


Figure 7.30 – Offset Lines

We select the Offset icon on the Inventor ribbons. We highlight the diagonal dimension and we delete it from the drawing. We add two dimensions. We change the outside vertical measurement to 2.875 and the inside distance to 2.625.

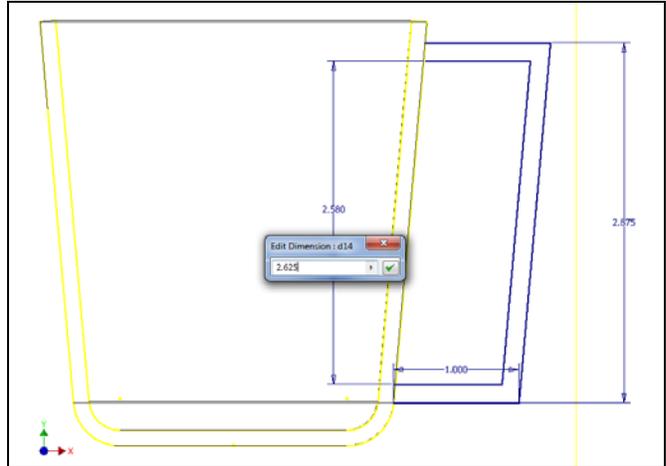


Figure 7.31 – Add Vertical Dimensions

We set the fillet radius to 0.25 for the inside arc. We will select the two inside lines as shown in the figure and the two lines will have a 0.25 radius arc appear at the intersection of the two lines. Then we will select the two outside lines as shown in the figure and the two lines will have a 0.375 radius arc appear at the intersection of the two lines.

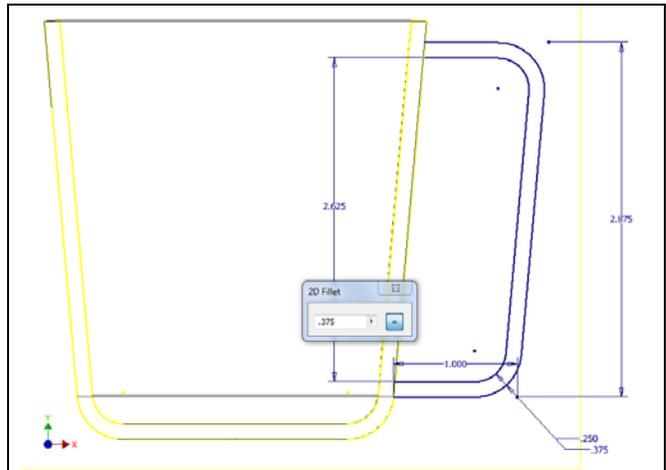


Figure 7.32 – Add Four Fillets

A straight extrusion against a revolved solid will result not combine as we want so we will extend the handle profile into the profile of the cup.

We start on the left end of the top line and we draw a line horizontally to the left, then 0.125 downward and then we connect at the green dot with the offset line.

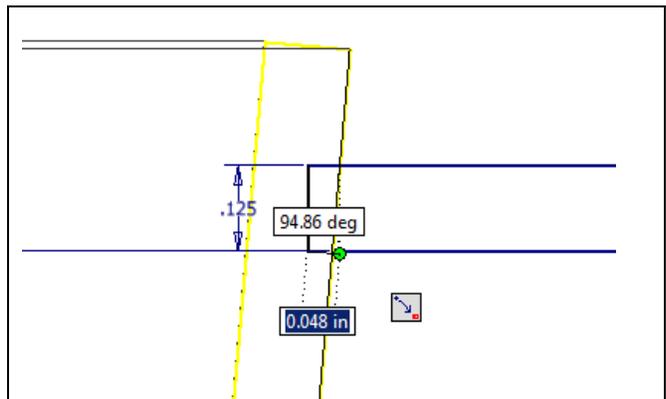


Figure 7.33 – Add Three Lines

We repeat the process on the bottom of the cup where the handle interfaces with the cup profile.

We start on the left end of the top line and we draw a line horizontally to the left, then 0.125 downward and then we connect at the green dot with the offset line.

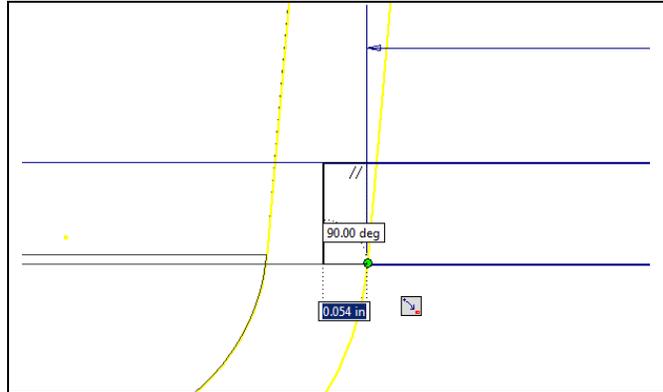


Figure 7.34 – Add Three More Lines

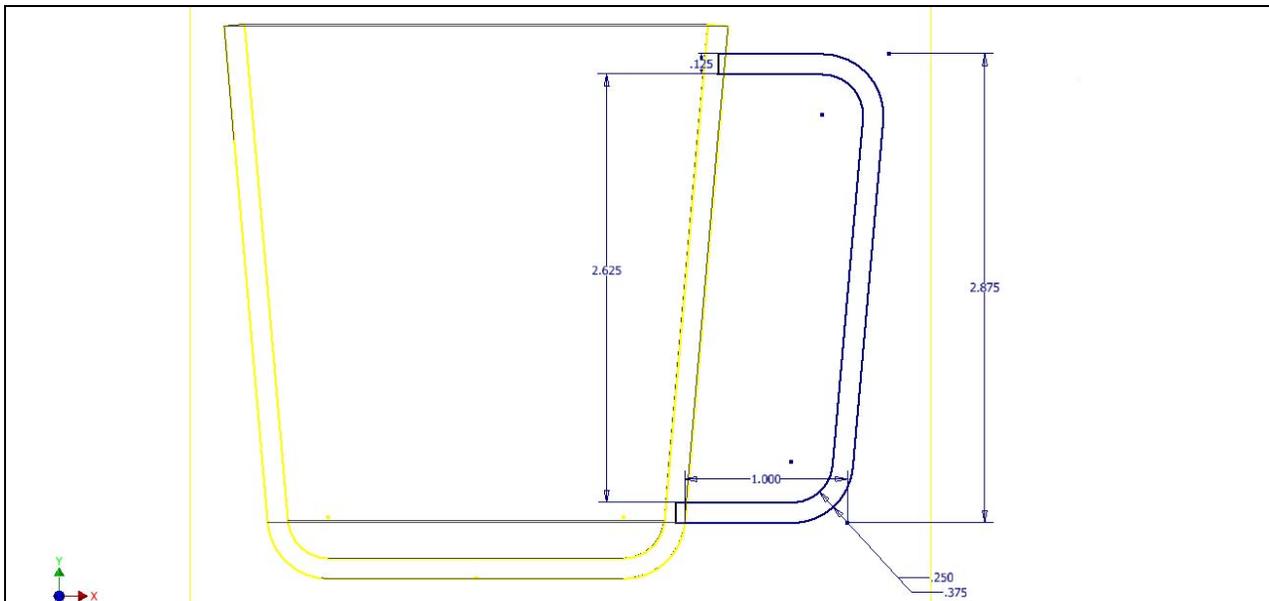


Figure 7.35 – Finished Handle Profile

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

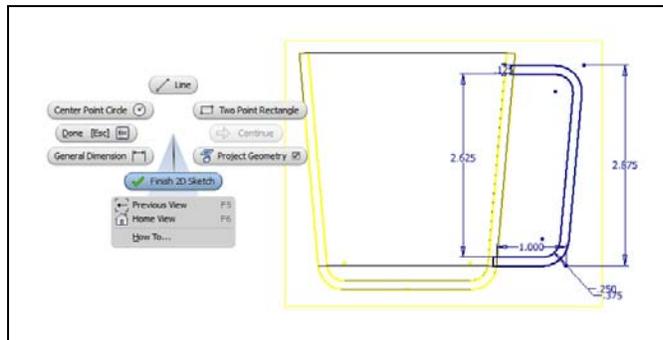
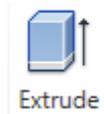


Figure 7.36 – Finish the 2d Sketch

Extrude the Handle Profile

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 Inventor 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. 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 handle area and when it turns red, we click again to extrude the solid.

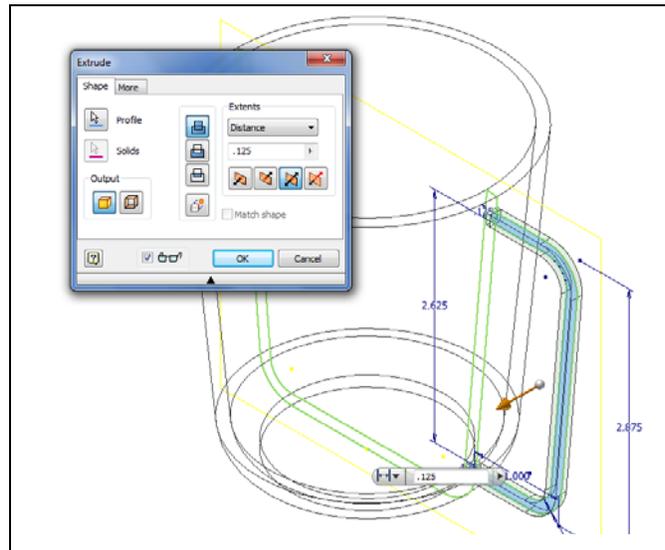


Figure 7.37 – The Extrusion Window

Add 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 0.0625. When we select the handle edged corner, the pointed edge will change to a 0.0625 inch rounded corner.

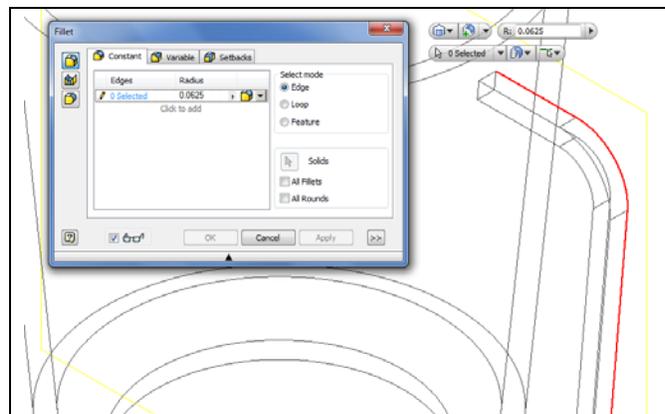


Figure 7.38 – Fillet Window

We should select the other three straight edges and the pointed edges will change to a 0.0625 inch rounded corners.

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

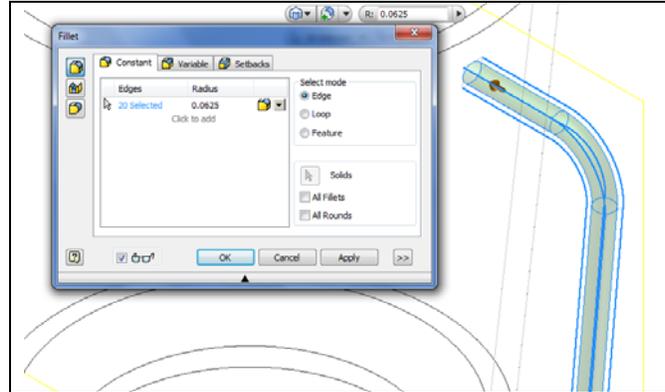


Figure 7.39 – Fillet All Four Edges

We will select fillet again and change the radius to 0.125. When we select the handle interface at the top of the cup, the straight edge will change to a 0.125 inch rounded corner as shown.

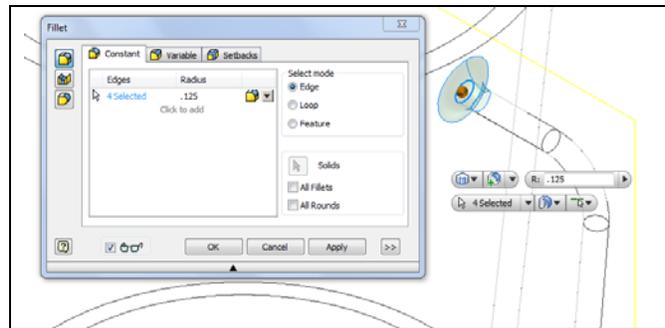


Figure 7.40 – Fillet Top Interface

When then we select the handle interface at the bottom of the cup, the straight edge will change to a 0.125 inch rounded corner as shown.

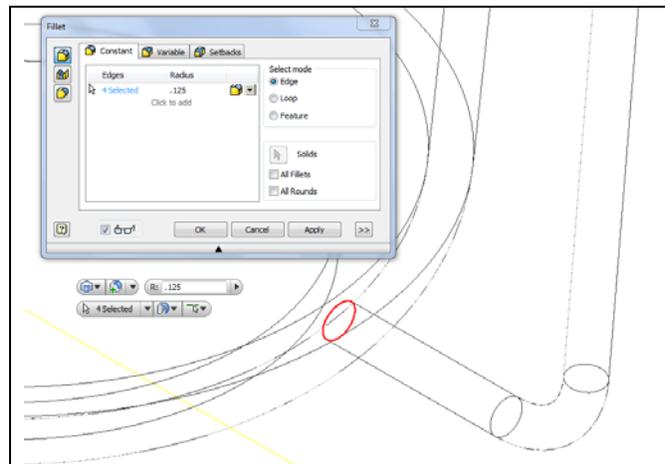


Figure 7.41 – Fillet Bottom Interface

Again, we will select fillet and change the radius to 0.03125. When we select the inside top lip of the cup, the straight edge will change to a 0.03125 inch rounded corner as shown.

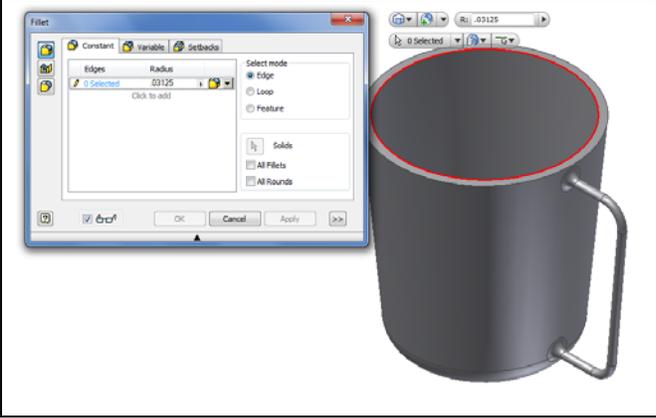


Figure 7.42 – Fillet Top Inside Lip

When we select the outside top lip of the cup, the straight edge will change to a 0.03125 inch rounded corner as shown.

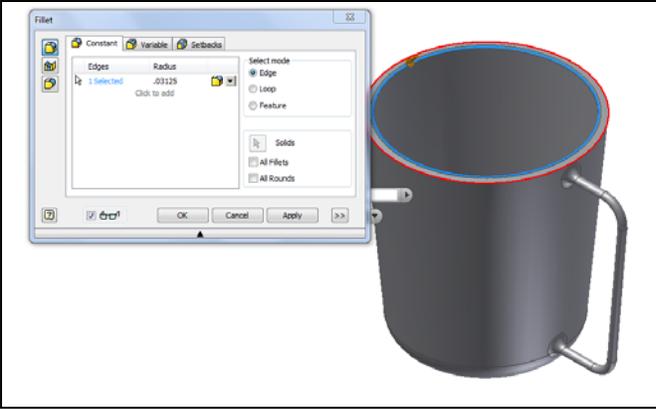


Figure 7.43 – Fillet Top Outside Lip

Save the drawing and we will now have a solid part to be used on a project.



Figure 7.44 – Finished Solid

*** World Class CAD Challenge 61-13 * - Close this drawing file. Create a New file and draw the six lines and 2 arcs and revolve the profile into a solid. Make a handle sketch and extrude the profile and add the fillets. Complete the task in less than 15 minutes. Continue this drill four times, each time completing the drawing under 15 minutes to maintain your World Class ranking.**

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