

# Chapter

# 8

## Solid Part Seven

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**This chapter will cover the following to World Class standards:**

- **Sketch of a Casting**
- **Draw the Profile of a Casting**
- **Add Multiple Fillets**
- **Finish the 2D Sketch**
- **Revolve a 2D Sketch**
- **Create Another Sketch**
- **Cut a Hole Pattern**
- **Add Addition Fillets on a Solid**
- **Mirror the Casting**

## Sketch of a Casting

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When we make a casting that needs machining in several places, we will actually end up making two drawings, the first drawing will be the actual sand casting that has draft angles and added material to be removed by turning, boring and drilling the part, and the second file that will contain the machined piece. In our sketch, we need to portray the part we will place in an assembly, but then we will modify the outline with several features that are typical to items that are molded.

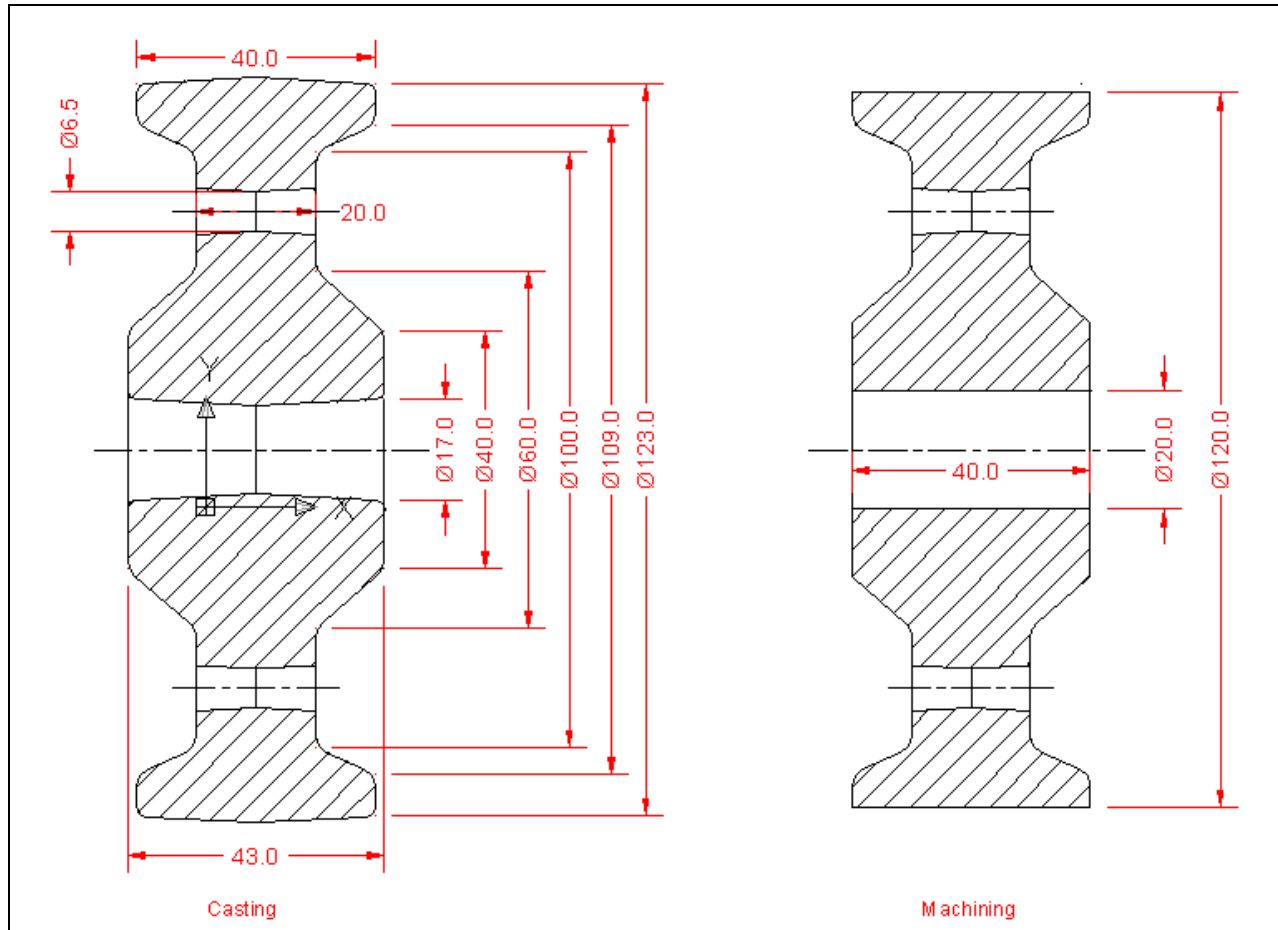
Foremost, a cast part needs to have draft angle which allows the matchplate that has the pattern of the part to be removed from the green sand without destroying the impression of the component. Depending on the process, the angle can vary, but a minimum of 3 degree draft angle is the required by many foundries to accomplish their work. Once we identify the direction that the piece will be placed on the matchplate, we will increase the angle of the surfaces towards the parting line.

Another feature in a casting and will be found in this exercise are fillets or rounds on any edge of the part. We require curved intersections of lines so that the cast part will cool evenly which is essential to having a homogenous material and similar strength and characteristics as the piece returns to ambient temperature. Designers and engineers taking a strength of material course at a technical school will testify that they have observed parts obtaining different mechanical characteristics after cooling down a metal part at different rates. So we will want to have even cooling on our cast piece and removing sharp edges promotes the consistent release of heat.

Adding material to the casting where we will turn, bore or drill to shape the component into proper tolerance is just as important as placing draft angles and fillets into our design. There are several factors that will determine the amount of material we add to the cast part. One aspect is the tolerance control of measurements of the cast process since we can lose material from the inaccuracy of making the impression. Typically, the tolerance control of measurements from the parting line to a feature in depth from that plane is smaller than for distances that are perpendicular to it. Therefore, in this exercise, the tolerance on the six hole pattern is not as good as the distance from the parting line to the bottom of the impression. If we do not add enough material in the casting, the machinist's drill bit will only drill into air and the finished part can have a part that does not meet the dimensional controls of the mechanical drawing.

Therefore, we will look at the sketch of the casting profile. We added 1.5 mm of material anywhere we wanted to bore or turn. In the areas where we do the machining, we reduced the fillets and rounds to 1.5 mm. On other areas of the casting, we set the radius for fillets to 3 mm. We will draw the intersection of lines at the theoretical dimension shown in the sketch of the casting, but when we apply the radius to the intersection, we cannot measure an exact dimension for where the two lines come together. We see two holes in the section view of the sketch, but there are actually six holes, equally spaced on a 360 degree array.

In the appendix following this exercise, we will create a new file from the casting file and make a machining drawing as we see on the left side of the sketch.

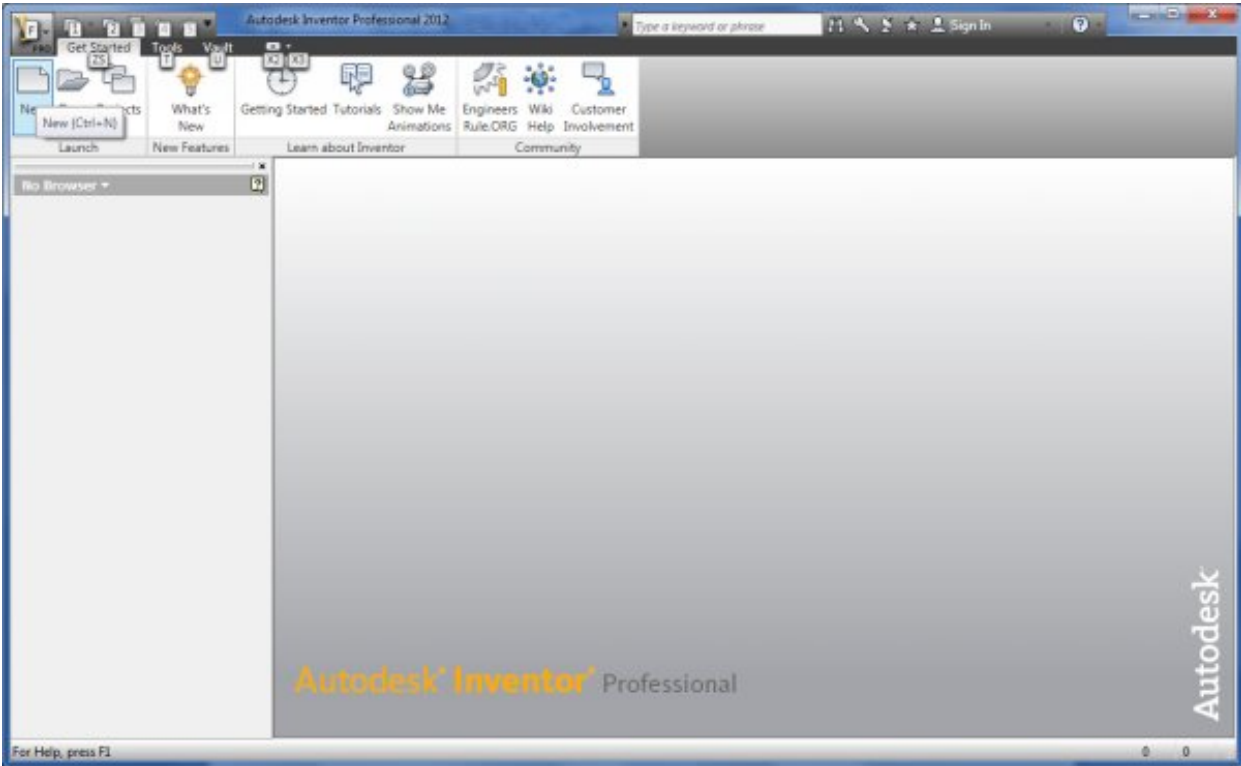


**Figure 8.1 – Problem Seven Sketch**

In this problem, we will practice techniques that we learned in the previous solid parts and add some new experiences such as drawing random lines and then constraining them with exact dimensions.

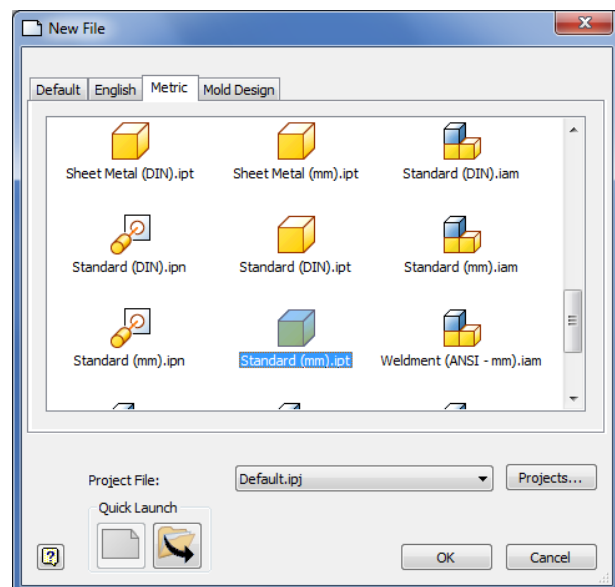
## Starting a 3D Part Drawing Sketch

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



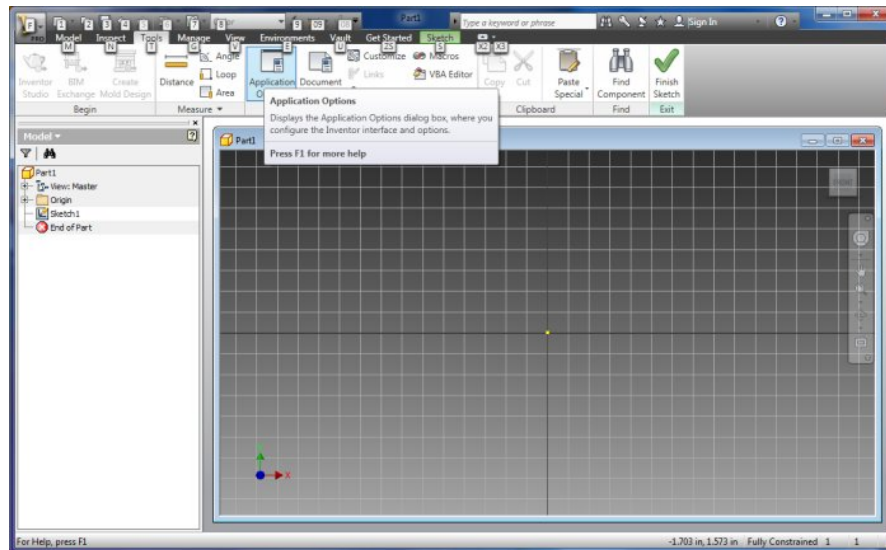
**Figure 8.2 – AutoCAD Inventor Professional 2012**

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 Metric tab and the Standard (mm) ipt template. We will press the OK button to continue.



**Figure 8.3 – Starting the drawing using the Standard Metric IPT template**

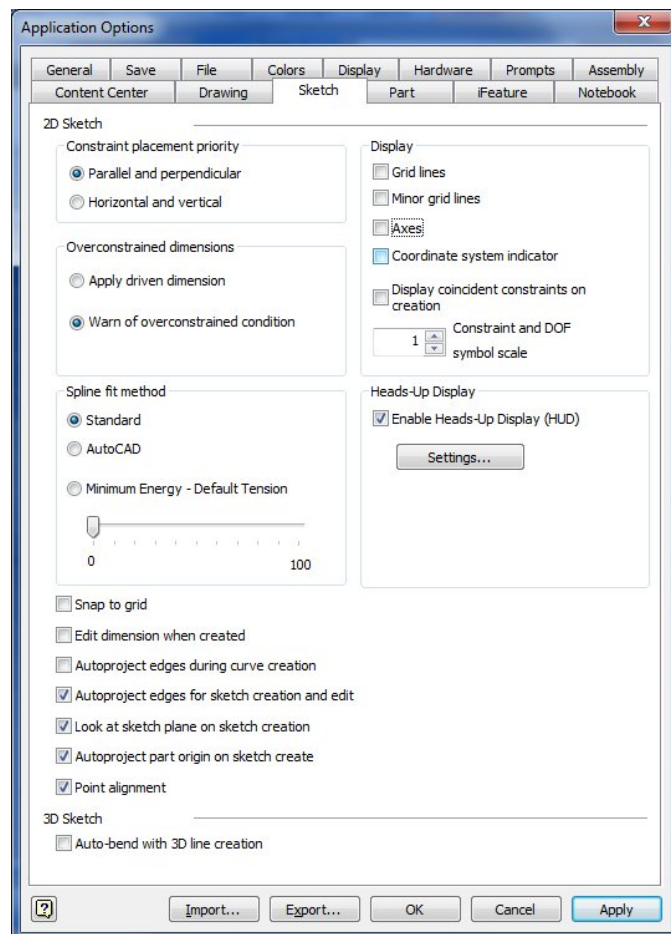
To turn off the grid if it is on the new drawing, we will go to the Tools tab on the Ribbon and choose Applications Options.



**Figure 8.4 – 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 8.5 – Application Options Window**

## Drawing the Profile of the Casting

We start our sketch with by drawing nine 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 20 mm the measurement textbox. We end the line by pressing the Enter key. This line is the centerline of the part.

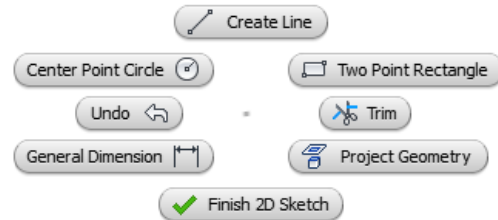


Figure 8.6 – Graphical Display Menu

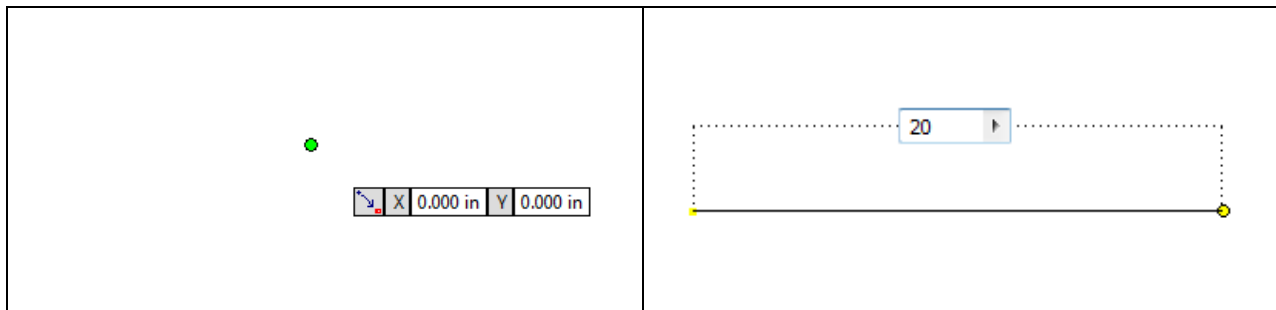


Figure 8.7 – Pick a Center Point

We select the Line tool and before we left click our starting point, we place the cursor over the left endpoint of the first line. We can see a dotted line rise as we pick a point exactly perpendicular to the first line.

Figure 8.8 – Enter the Dimension

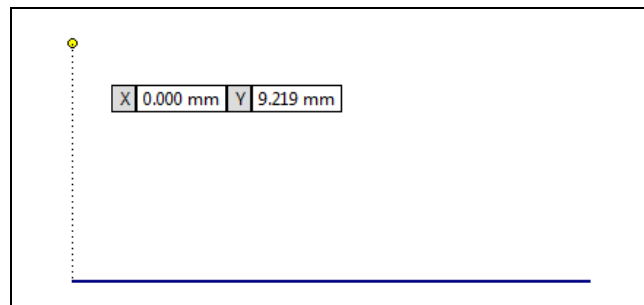
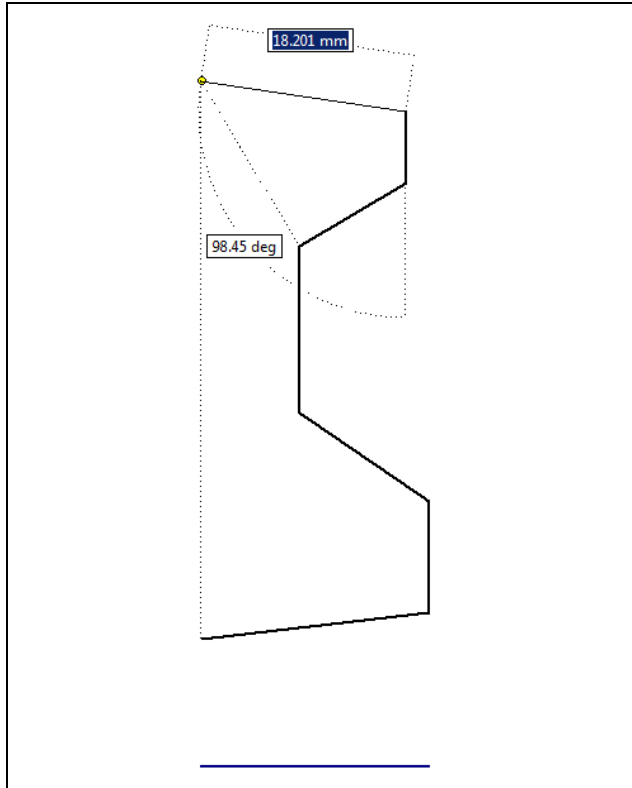
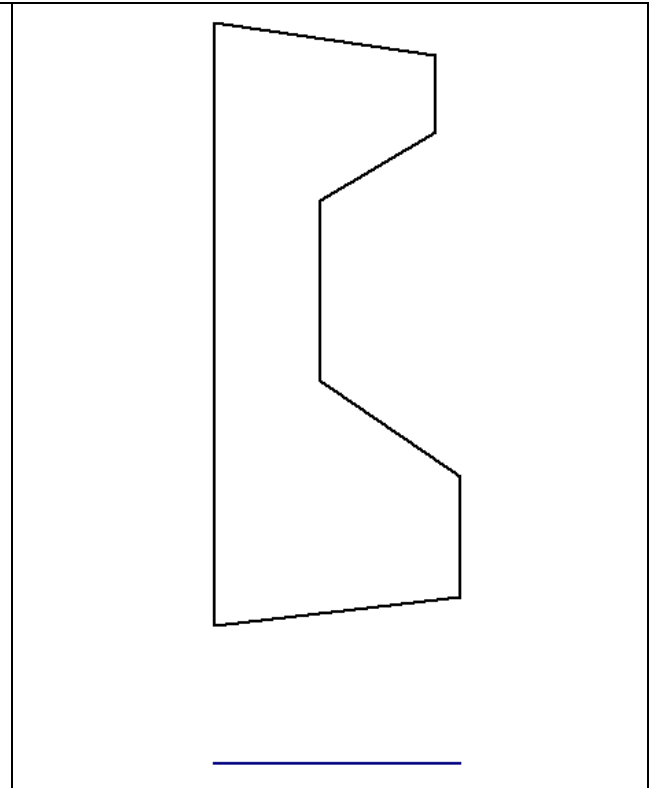


Figure 8.9 – Second Line of the Profile

Now that we have a starting point, we will draw eight lines in a pattern that has a similar shape as that shown figures 8.10 and 8.11. With practice, we can help ourselves by maintaining a sense of proportion, because drawing a shape too small can result in overlapping lines when we add dimensional constraints to the sketch. We are only drawing one quarter of the casting, since we are going to revolve the profile around the centerline of the part (the first line we drew) and we will mirror the part across the left line, which is referred to as the parting line of the part.

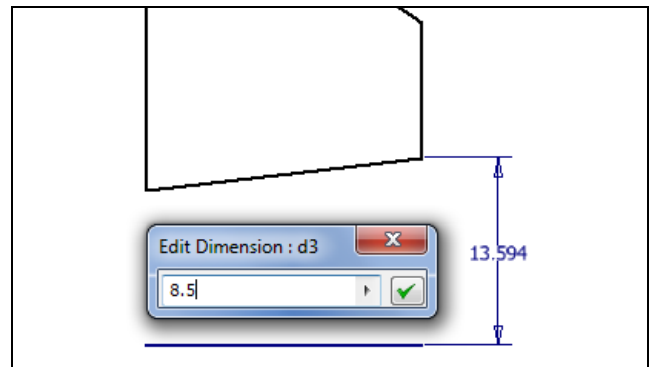


**Figure 8.10 – Creating a Random Profile**



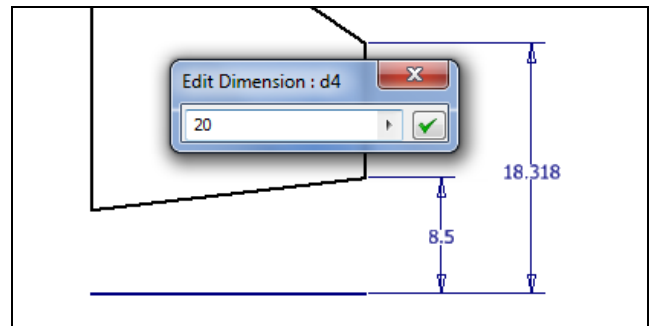
**Figure 8.11 – Creating a Random Profile**

We add a dimension from the centerline to the profile as shown. We do not care what measurement appears between the arrows, since we will double click on the number and the Edit Dimension window will appear. We will type 8.5 mm and press the green check icon. The profile will change to the new constraint.



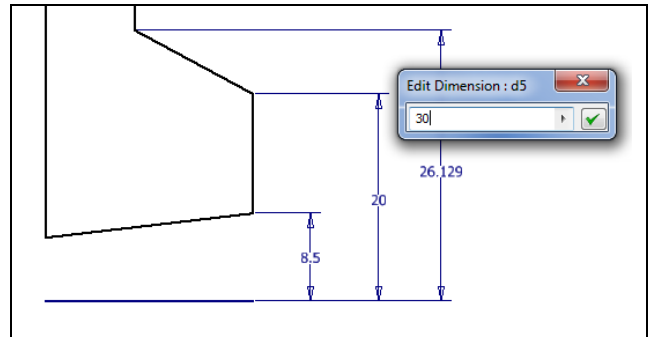
**Figure 8.12 – 8.5 mm Dimension**

We add another dimension from the centerline to the profile as shown. We will double click on the number and the Edit Dimension window will appear. We will type 20 mm and press the green check icon. The profile will change to the new constraint.



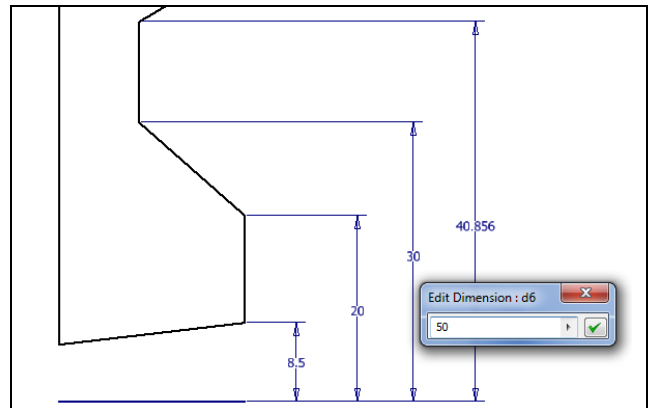
**Figure 8.13 – 20 mm Dimension**

We add our next dimension from the centerline to the profile as shown. We will double click on the number and the Edit Dimension window will appear. We will type 30 mm and press the green check icon. The profile will change to the new constraint.



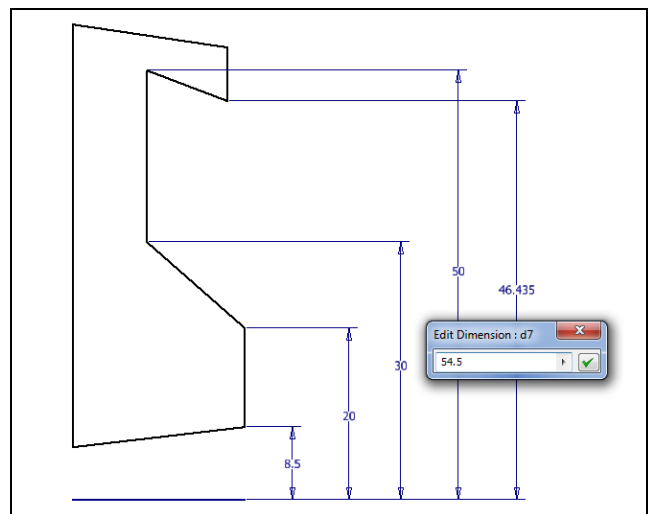
**Figure 8.14 – 30 mm Dimension**

We add our fourth dimension from the centerline to the profile as shown. We will double click on the number and the Edit Dimension window will appear. We will type 50 mm and press the green check icon. The profile will change to the new constraint.



**Figure 8.15 – 50 mm Dimension**

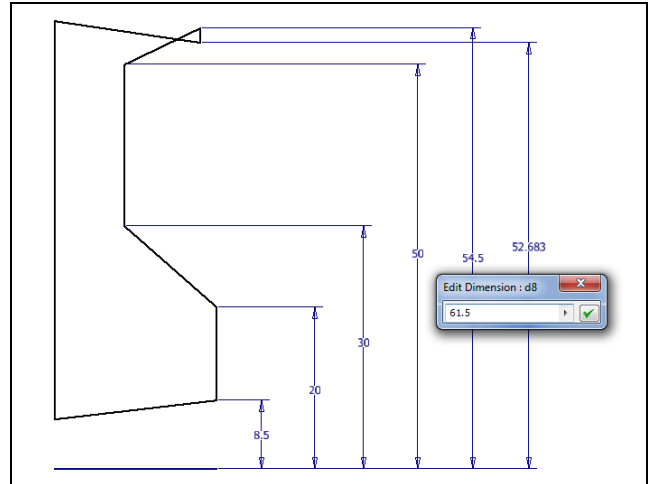
We add our fifth dimension from the centerline to the profile as shown. Notice the 50 mm dimension is above the point we are identifying, but it will move when we add the constraint. We will double click on the number and the Edit Dimension window will appear. We will type 54.5 mm and press the green check icon. The profile will change.



**Figure 8.16 – 54.5 mm Dimension**

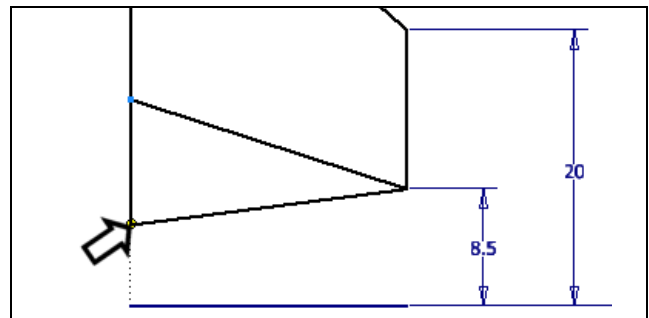


We add our sixth dimension from the centerline to the profile as shown. Again, our profile can be a little twisted, but it will move when we add the constraint. We will double click on the number and the Edit Dimension window will appear. We will type 61.5 mm and press the green check icon. The profile will change.



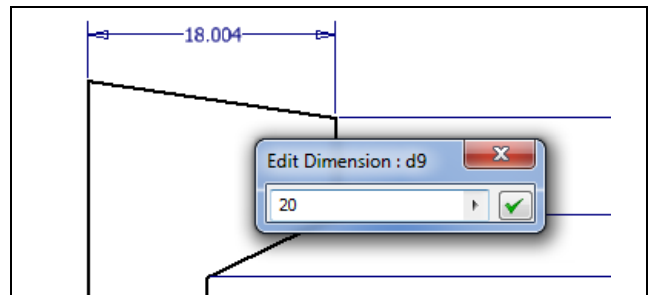
**Figure 8.17 – 61.5 mm Dimension**

On our profile, we can see the shape losing its proper proportion, so we use the cursor and we grab the lower left point and pull it down to maintain a contour that is suitable to the eye.



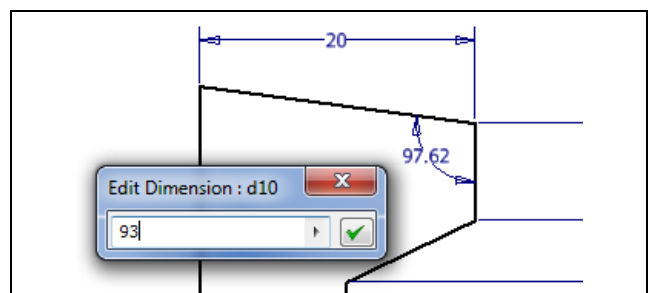
**Figure 8.18 – Moving the Shape**

We add another dimension from the left line to the front as shown. We will double click on the number and the Edit Dimension window will appear. We will type 20 mm and press the green check icon. The profile will change.



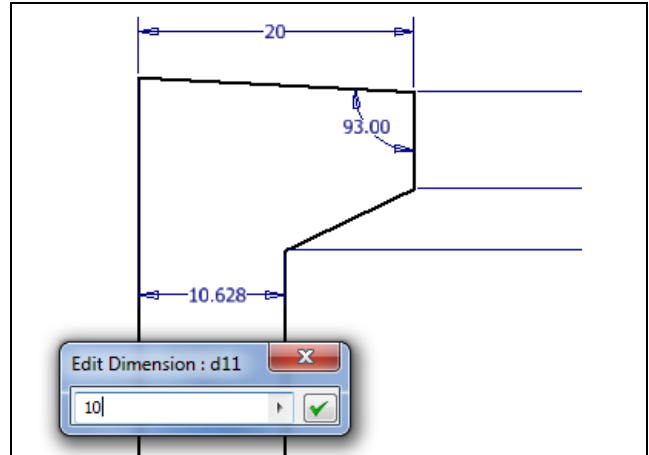
**Figure 8.19 – 20 mm Dimension**

We can add an angular dimension by first picking one line and the other line where the circular measurement will describe. We will double click on the number and the Edit Dimension window will appear. We will type 93 and press the green check icon. The angle will change.



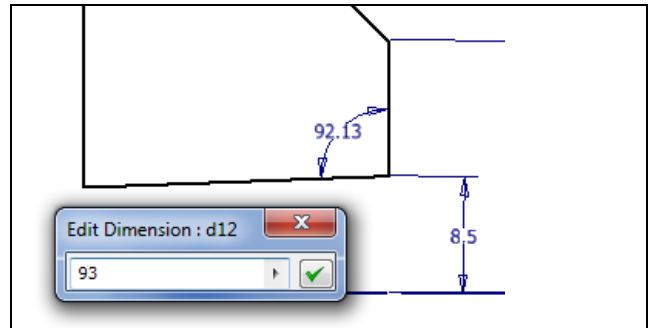
**Figure 8.20 – 93 Degree Angle**

We add another dimension from the left line to the front as shown. We will double click on the number and the Edit Dimension window will appear. We will type 10 mm and press the green check icon. The profile will change.



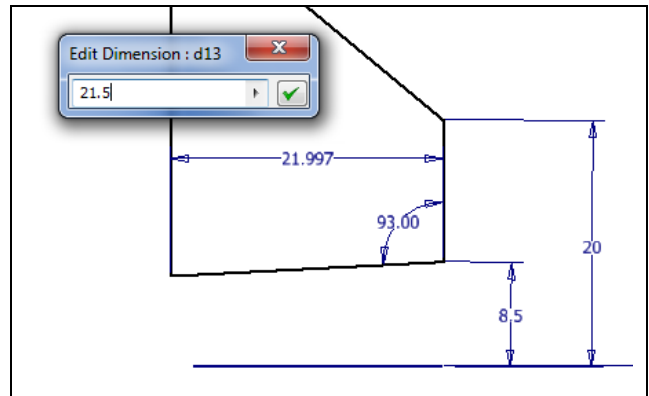
**Figure 8.21 – 10 mm Dimension**

We can add an angular dimension by first picking one line and the other line where the circular measurement will describe. We will double click on the number and the Edit Dimension window will appear. We will type 93 and press the green check icon. The angle will change.



**Figure 8.22 – 93 Degree Angle**

We add another dimension from the left line to the front as shown. We will double click on the number and the Edit Dimension window will appear. We will type 21.5 mm and press the green check icon. The profile will change.



**Figure 8.23 – 21.5 mm Dimension**

## Add Multiple Fillets

The next feature we will add to our sketch is the six 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 1.5 for the outside arcs. We will select the 20 and 61.5 lines as shown in the figure and the two lines will extend and a 1.5 radius arc will appear at the intersection of the two lines.

We will select the 21 and 8.5 lines as shown in the figure and the two lines will extend and a 1.5 radius arc will appear at the intersection of the two lines.

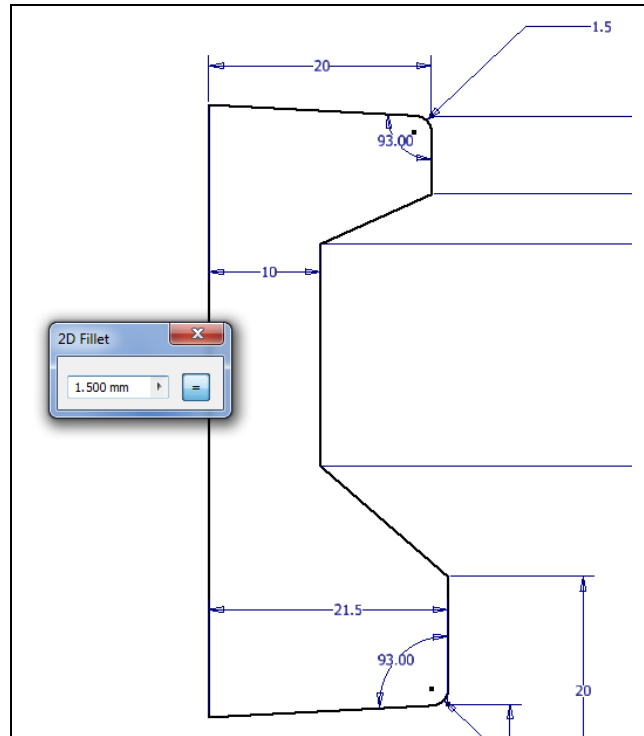


Figure 8.24 – 1.5 mm Arc

We will then set the fillet radius to 3 for the inside arcs. We will select the 4 pairs of intersecting lines as shown in the figure and the 3 mm arcs will appear each instance.

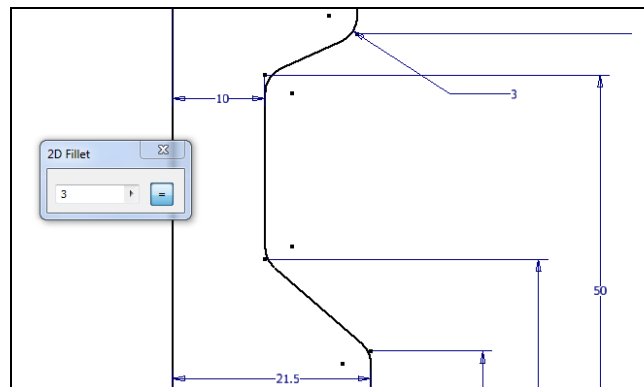
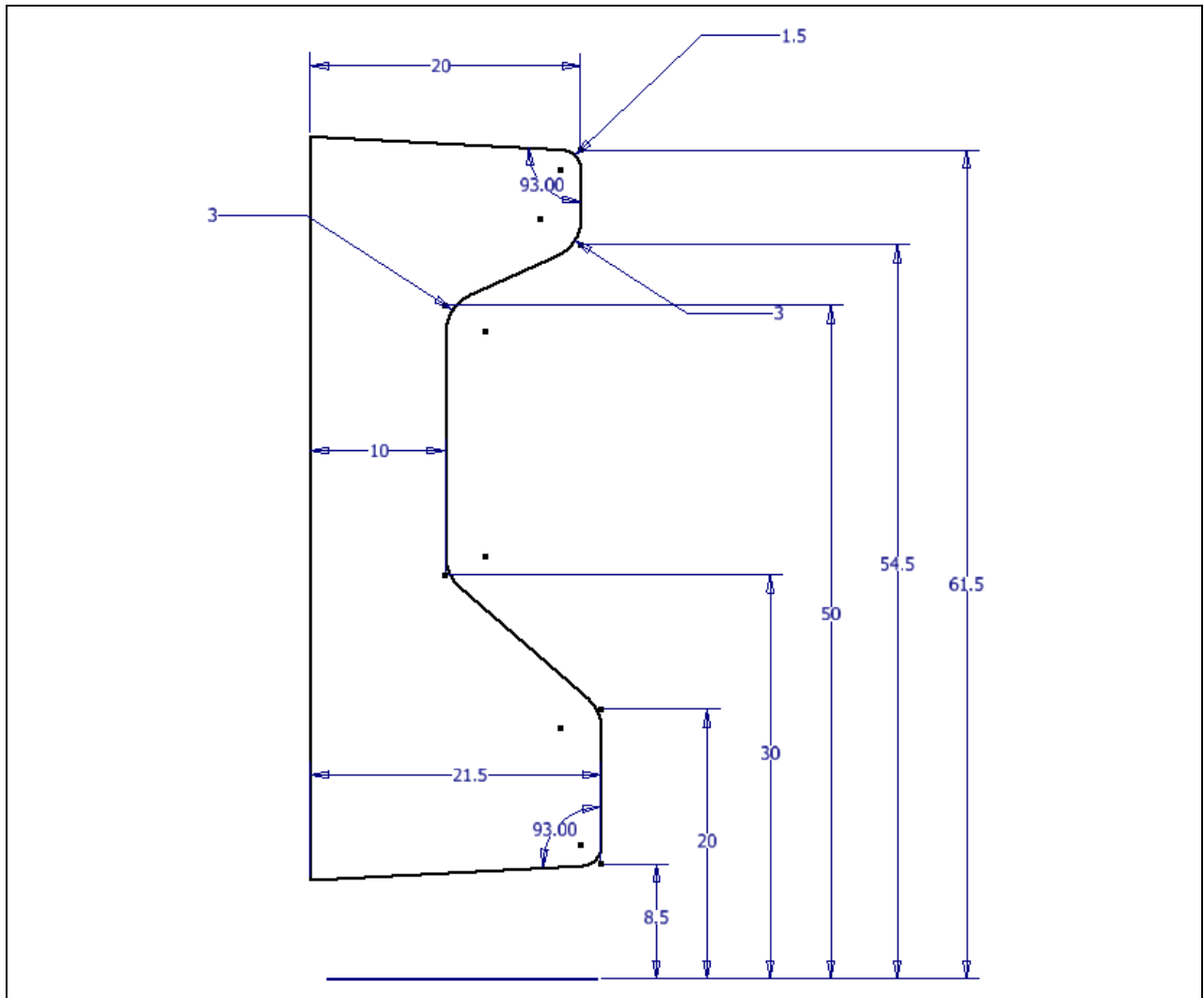


Figure 8.25 – 3 mm Arc

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 8.26 – The Finished Sketch**

**\* World Class CAD Challenge 61-14 \* - Close this drawing file. Create a New file and draw the profile with 9 lines and six 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](http://www.worldclasscad.com) to obtain your world class ranking.**

## Finish 2D Sketch of the Casting

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.

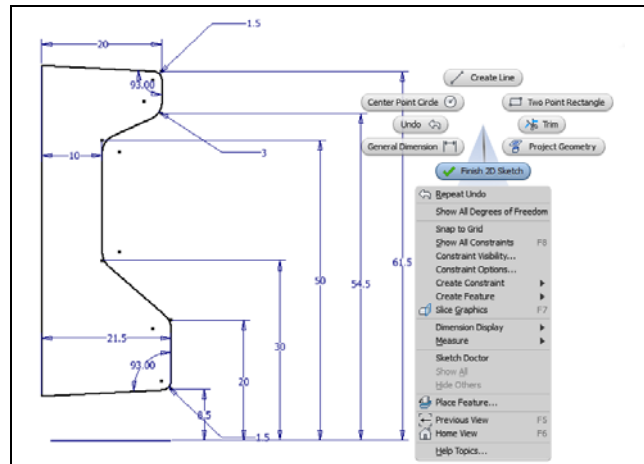


Figure 8.27 – Finished Part Two Sketch

## 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, we select the Profile icon, and we can choose the shape we just made. The inside of the profile will become shaded.

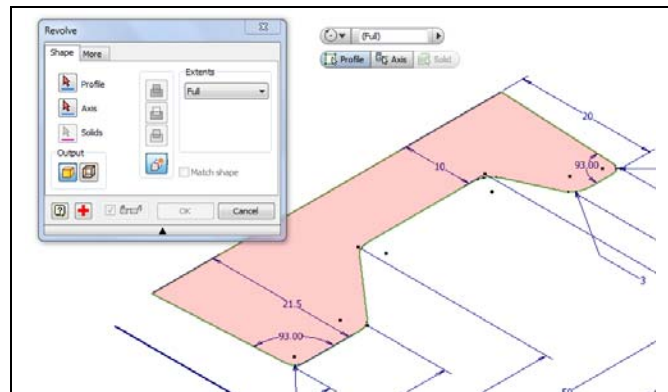
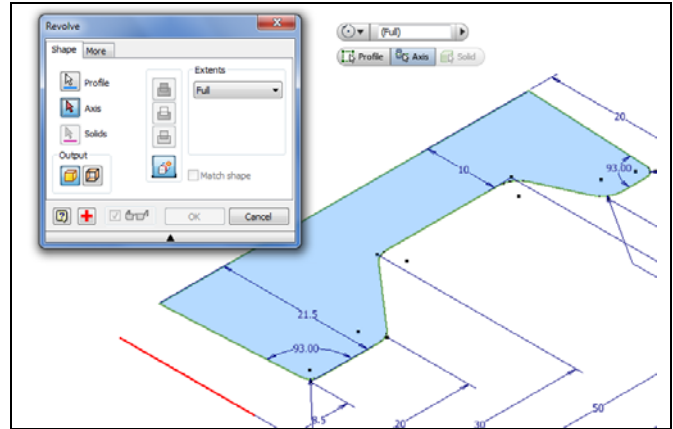


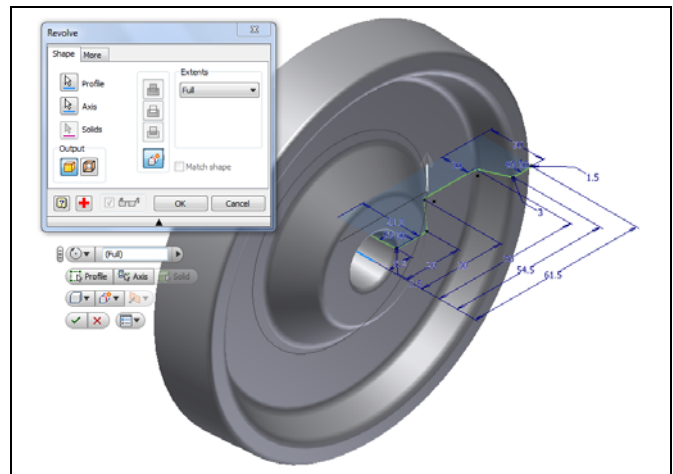
Figure 8.28 – Revolve Window

We then pick the Axis icon and then we pick the centerline we first drew. We will see the line turn red.



**Figure 8.29 – Select the Axis**

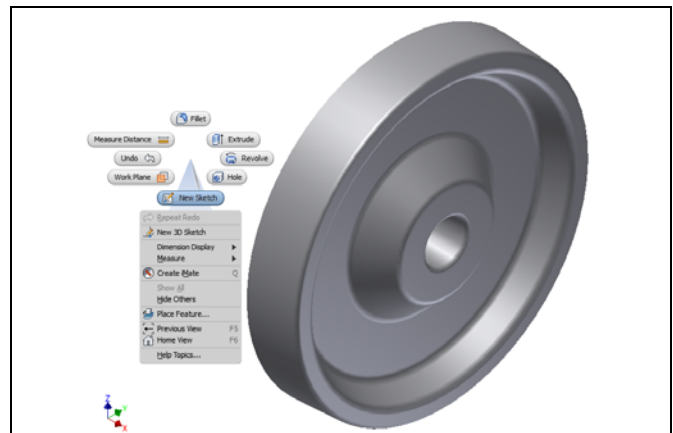
We will revolve it to the full extents or 360 degrees. We press the OK button for it to take place.



**Figure 8.30 – Revolve the Profile 360 Degrees**

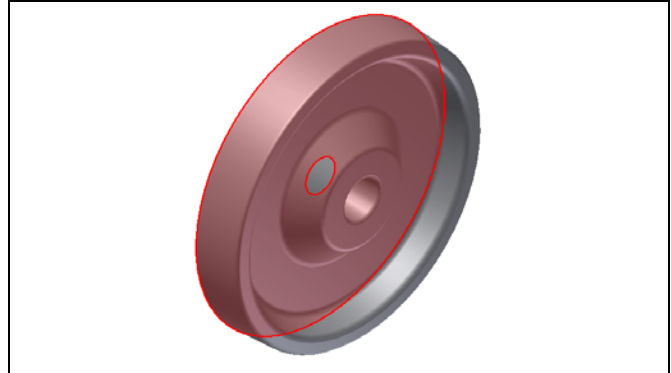
## Create Another Sketch

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



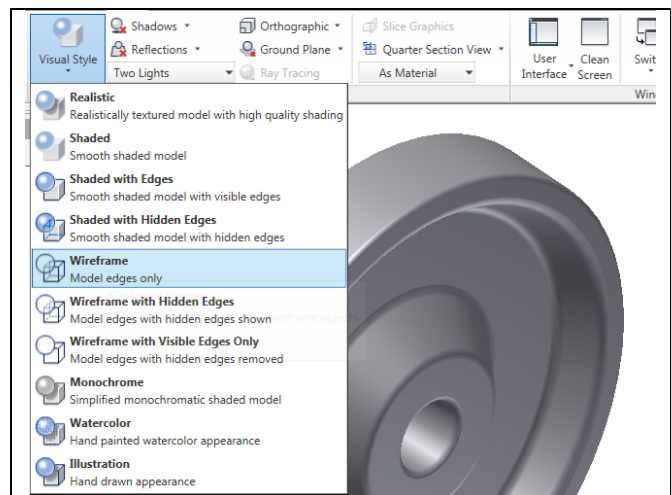
**Figure 8.31 – Start a New Sketch**

We select the plane to create the 2D sketch.



**Figure 8.32 – Select the Back Plane**

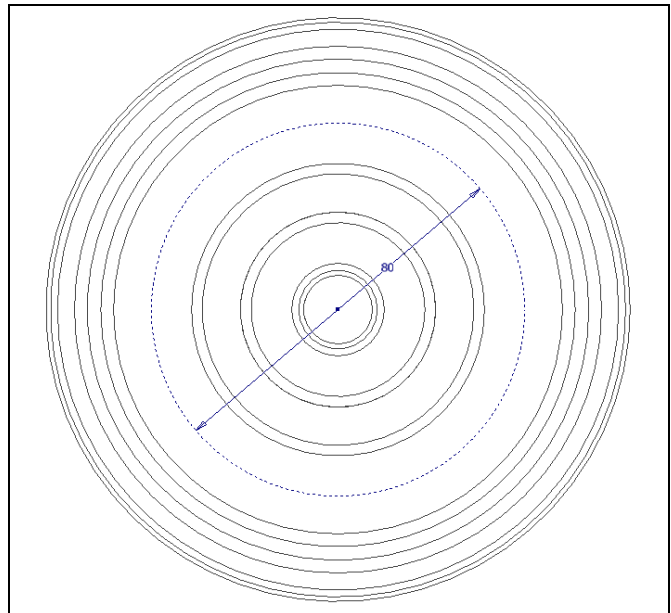
We will work off the original profile so we go to View on the Inventor ribbon and we choose Visual Style and Wireframe.



**Figure 8.33 – Change the View to Wireframe**

Next, we will draw a circle on the drawing except we will create it using a construction line. We turn on this feature by clicking on the Construction button on the right side of the Inventor Ribbon inside the Format section. Then we right click on the graphical display and we choose Center Point Circle. Remember we drew the centerline on the 0,0 point at the beginning of the exercise.

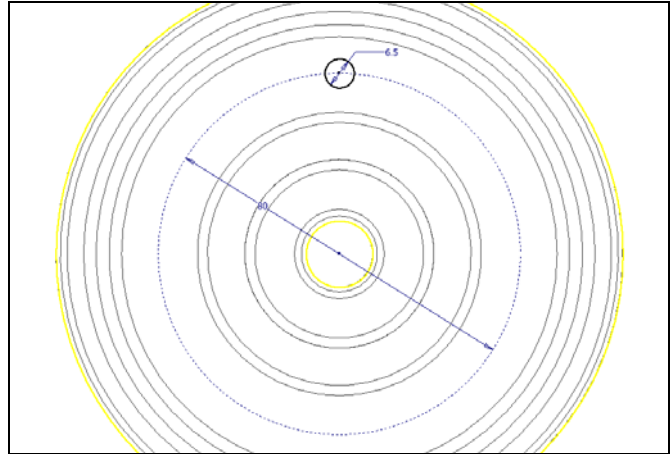
We click on the green point, on the diameter textbox, we type 80 mm, and we press Enter. We can see that the construction line is brown.



**Figure 8.34 – Revolve the Profile 360 Degrees**

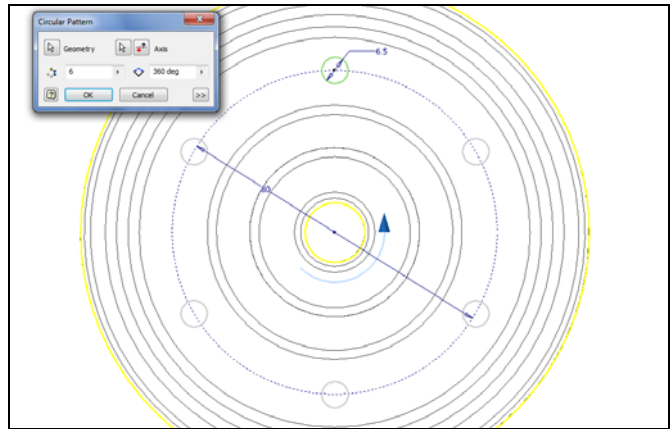
Now we will want to array 6 circles on the 80 mm circle. First, we turn off the Construction Format button on Inventor Ribbon. Then we right click on the graphical display and we choose Center Point Circle.

We then point to the top quadrant of the brown construction bolt circle and we will see a yellow dot. We click on the yellow point, on the diameter textbox, we type 6.5, and we press Enter.



**Figure 8.35 –80 mm Construction Circle**

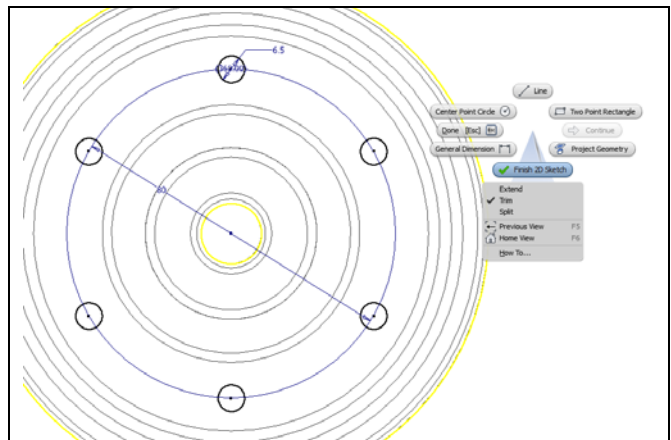
We can array the last entity using a circular pattern. We select Circular on the Sketch Ribbon, we will see the Circular Pattern Window appear on the graphical display. We first press the Geometry arrow and select the 6.5 circle to array. Then we pick the Direction arrow and we choose the brown circle. An arrow will appear showing the direction of the array.



**Figure 8.36 – Draw a 6.5 mm Circle**

We will type 6 for the number of holes in the circular array on the left textbox of the window and we will input 360 degrees on the left textbox. With 360 degrees, we will have 60 degrees between each 6.5 diameter circle. We press the OK button to add the entities.

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.

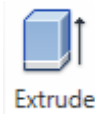


**Figure 8.37 – Circular Array**



## Extrude the Hole Pattern

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. We will pick the Solid output on the left. We select the Cut icon so we can remove the material for the holes from the casting. We will keep the Extents distance 10 mm.

Underneath the distance textbox, we will pick the Direction 2 button (second from the left), so that the holes will extrude into the casting.

We select the six circles carefully.

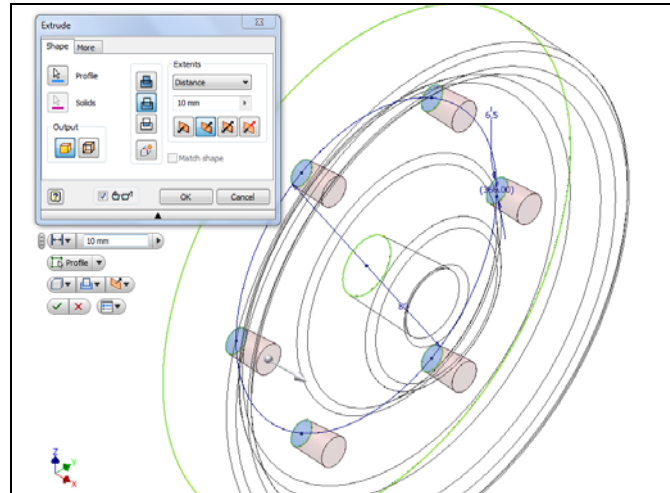


Figure 8.38 – The Extrusion Window

On the More tab, we change the Taper textbox to 3 degrees.

We then press the OK button and six holes will appear in the casting.

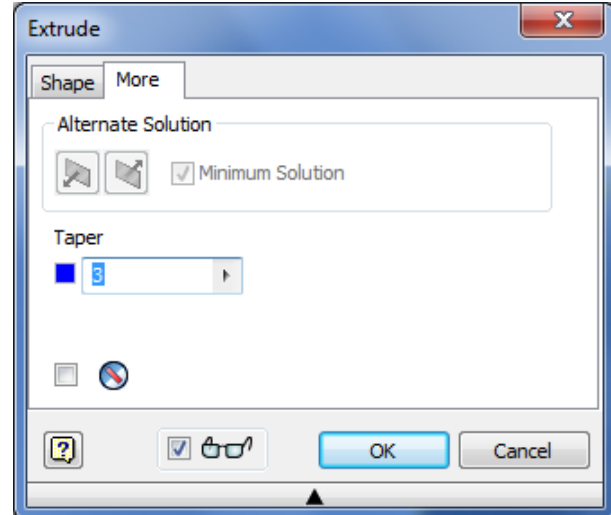


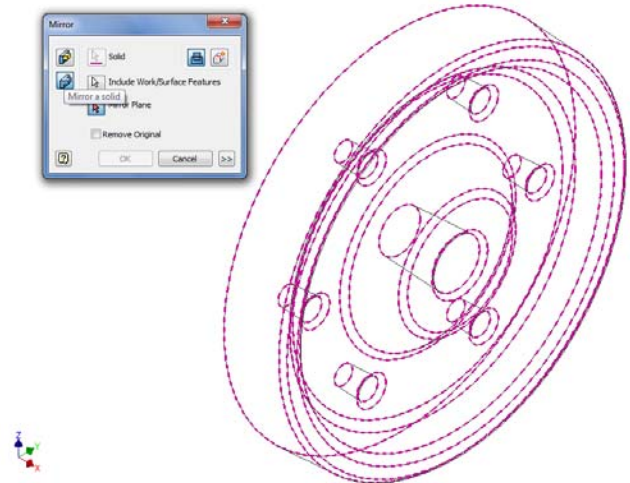
Figure 8.39 – The 3 Degree Taper

## Mirror the Casting

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

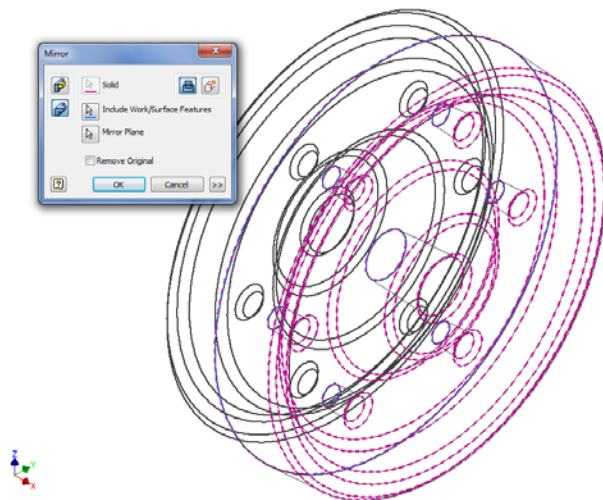


We pick the Solid icon and the entire solid is selected.



**Figure 8.40 – Select the Entire Solid**

Then we select the Mirror Plane icon. We pick the back plane as shown in figure 8.41. The casting will appear on the other side of the back plane. We press the OK button to unite the two solids.



**Figure 8.41 – Select the Mirror Plane**

## Add Fillets

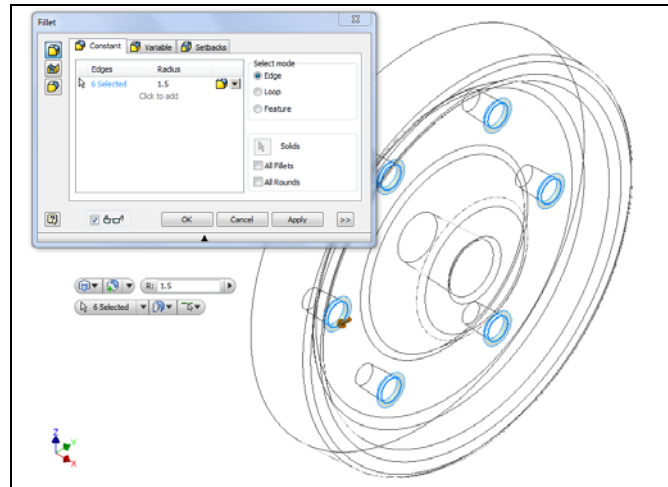
The next feature we will add to our bracket is the four fillets. We choose the Fillet button on the Inventor ribbon and the Fillet window will appear on the graphical display.



We set the fillet radius to 1.5 mm. When we select the top edge of the hole, the pointed edge will change to a 1.5 mm rounded corner.

We should select the other five straight edges and the pointed edges will change to a 1.5 mm rounded corners.

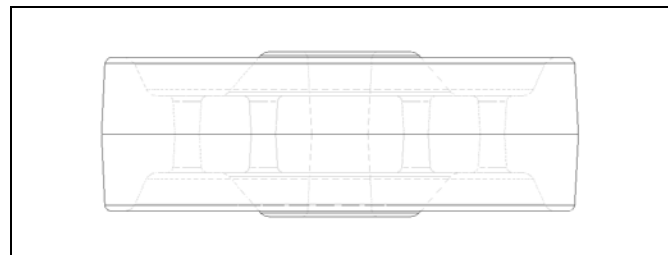
To make the placements permanent, we press the OK button.



**Figure 8.42 – Fillet Window**

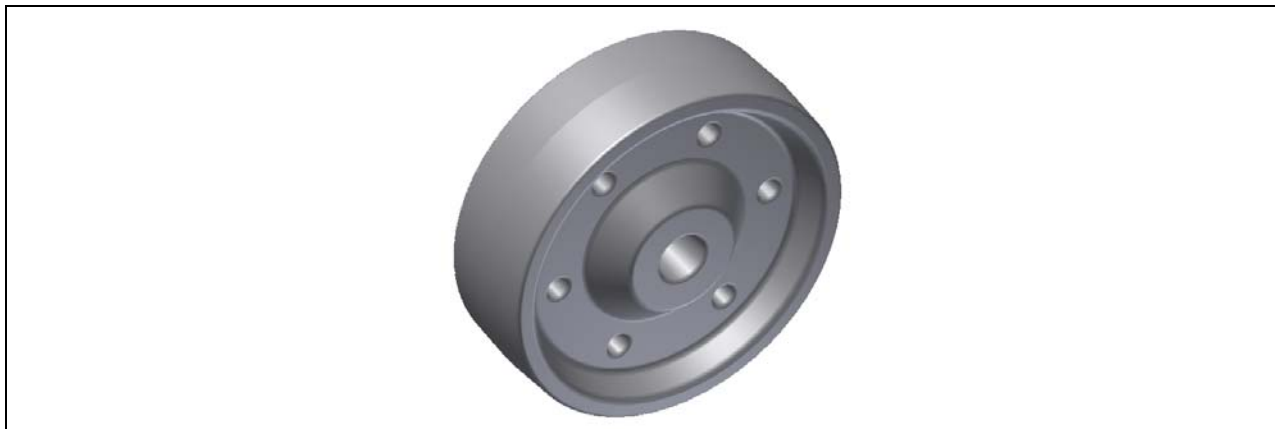
We can change the view of the casting and rotate it in the wireframe with hidden lines shown option. We can see the detail that we created.

Go ahead and change the view back to shaded.



**Figure 8.43 – Fillet Top Outside Lip**

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



**Figure 8.44 – Finished Solid**

**\* World Class CAD Challenge 61-15 \* - Close this drawing file. Create a New file and draw the nine lines and six arcs. Revolve the profile into a solid. Make a sketch showing the six holes and cut the holes from the casting. Add the fillets to the top of the holes and mirror the casting. 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.**

**\* World Class CAD Challenge \* - Report your best times to World Class CAD at [www.worldclasscad.com](http://www.worldclasscad.com) to obtain your world class ranking.**