Chapter

6

Solid Part Five

This chapter will cover the following to World Class standards:

- Sketch of Solid Problem
- Draw a Circle
- Draw a Concentric Circles
- Draw a Hub
- Use the Trim Function
- Draw a Circular Array
- Finish the 2D Sketch
- Extrude a 2D Sketch with a Draft Angle
- Add a Sketch on a Model
- Extrude another 2D Sketch with a Draft Angle
- Revolve a Profile and Subtract from a Solid

Sketch of a Solid Part

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We start this project by making a sketch, so we can efficiently produce a solid model. In this part, we see a sketch with lines and circles. Our strategy is to create a pattern like that shown in the front view and then we will add the additional material and subtract the 1.0-inch diameter from the inside. The drawing is of a casting, so we will extrude the profiles with a 5-degree draft angle. Depending on the direction of the extrusion, we will either use a positive or negative draft angle to achieve the desired results. For a die cast part, that gives us the ability to remove that part from the mold.
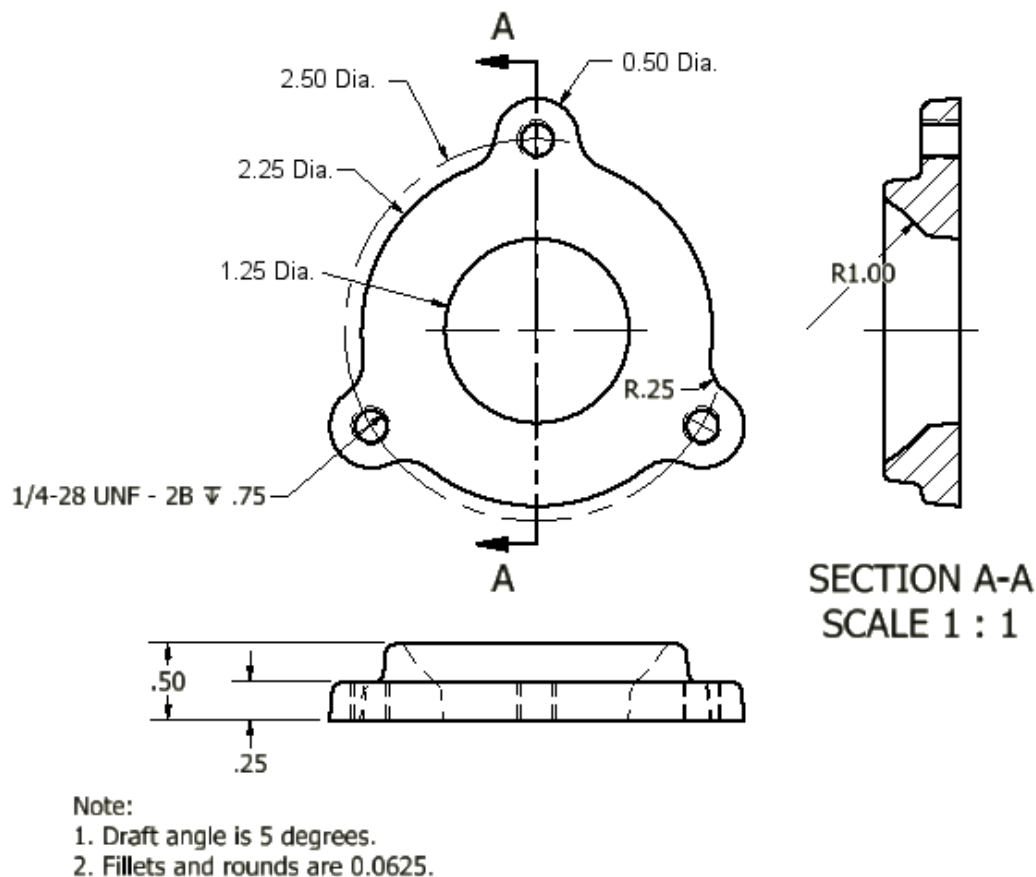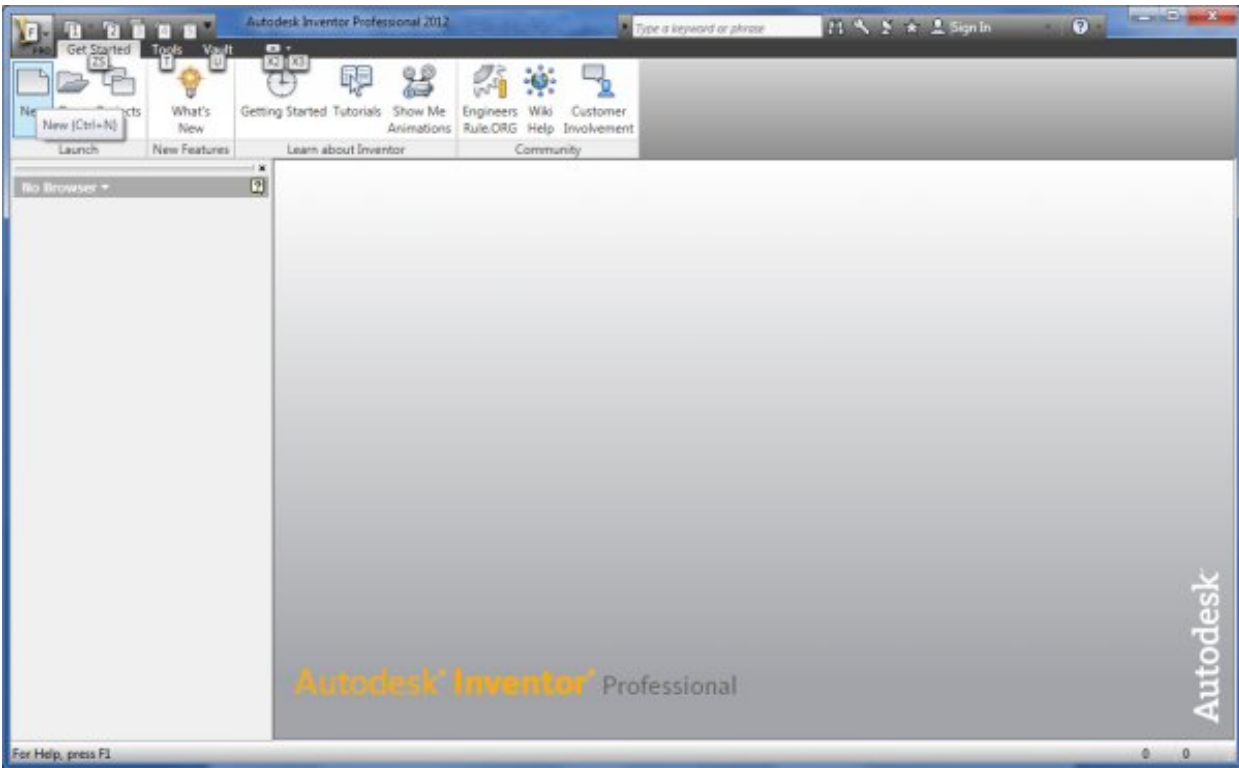

Figure 6.1 – Problem Five Sketch

In this problem, we will practice techniques that we learned in the previous solid parts and add some new experiences such as revolve, an offset work plane and fillets with continuous loops.

Starting a 3D Part Drawing Sketch

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

Figure 6.2 – 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.
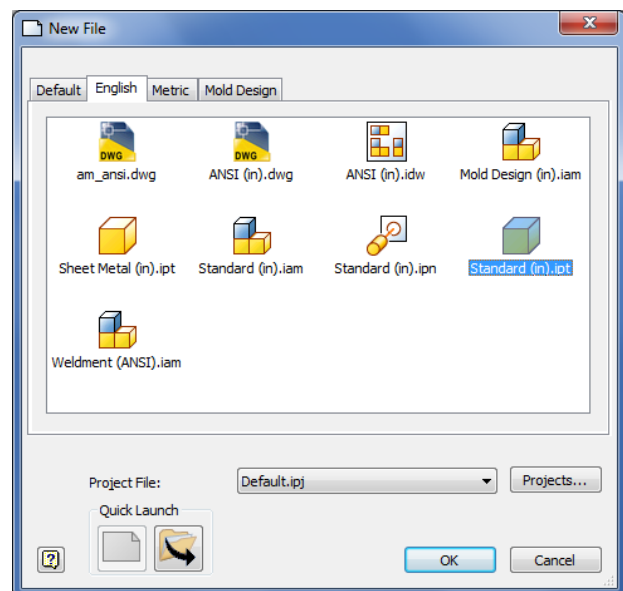
**Figure 6.3 – Starting the drawing using the Standard IPT template**

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.
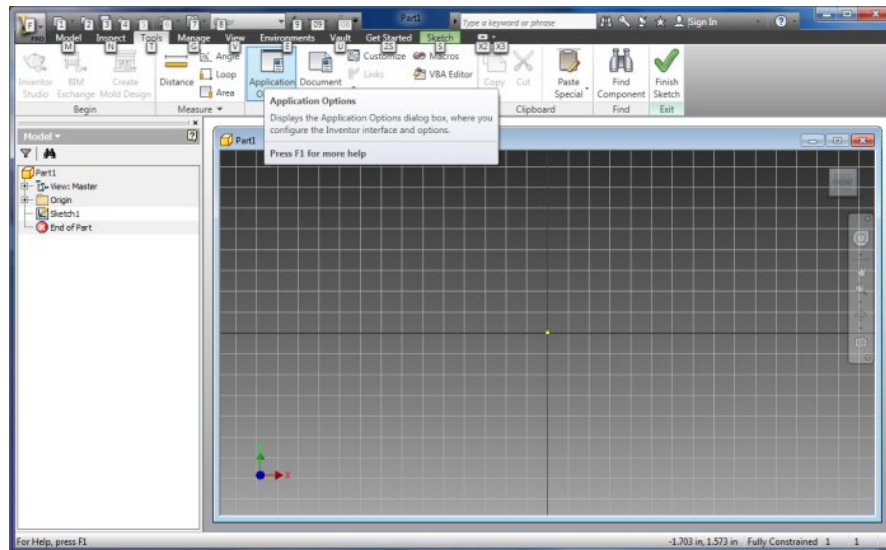
Figure 6.4 – Starting the drawing using the Standard IPT template

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.
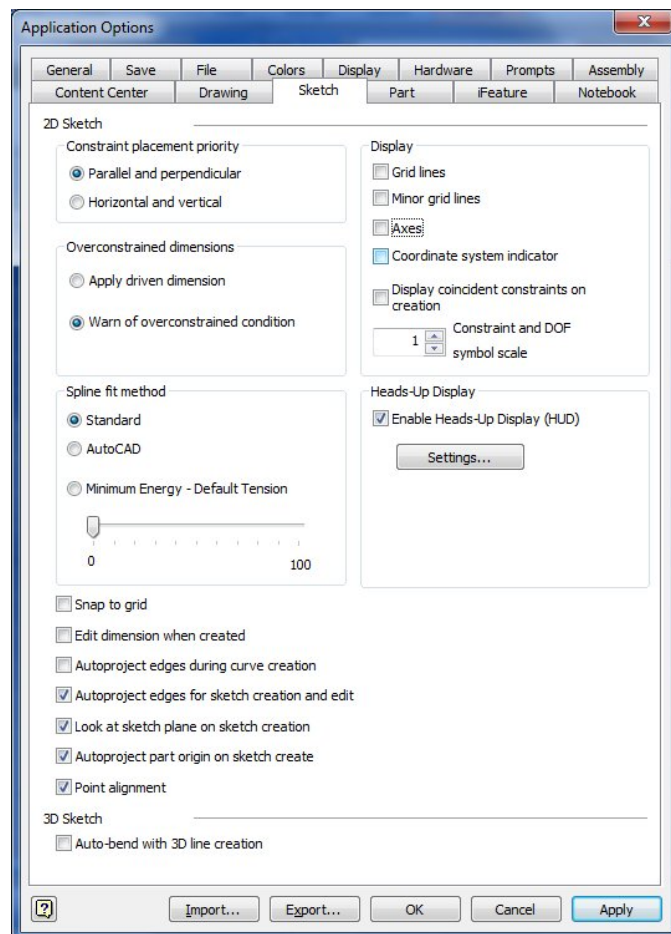
Figure 6.5 – Application Options Window

Drawing a Circle

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We can sketch our design in Inventor many different ways and achieve the same technique. The first method we will utilize is the Center Point Circle.

To draw a circle, we right click on the drawing and we can see Create Line, Center Point Circle, Two Point Rectangle and many more choices. We pick Center Point Circle and we will single click on the center portion of the graphical display at the yellow dot with the coordinates of 0,0 and then we expand the circle on the display. By using the origin 0,0, we can reference the middle of the part later in the development process.
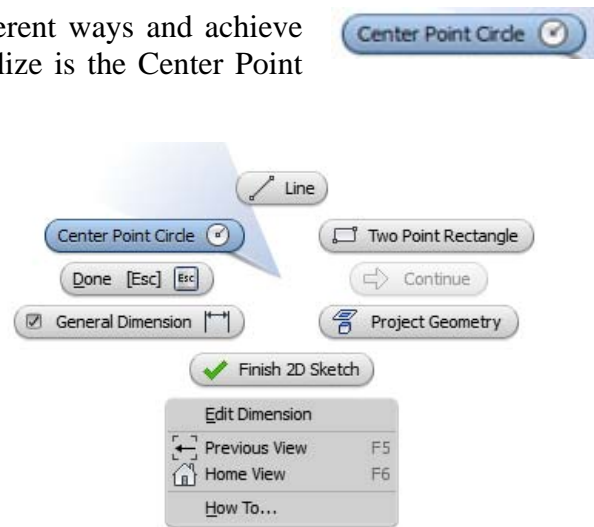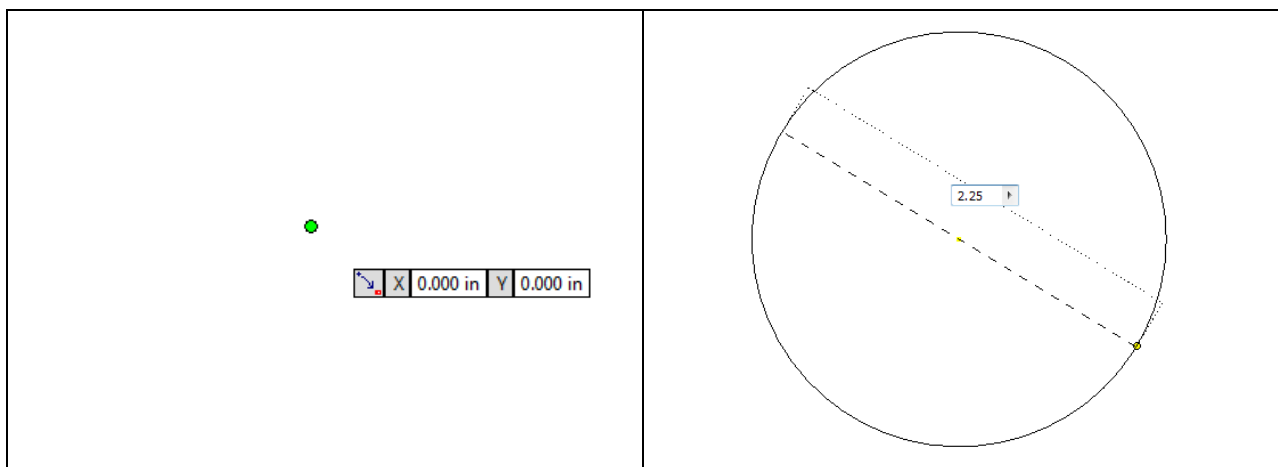
Figure 6.6 – Graphical Display Menu

**Figure 6.7 – Pick a Center Point**   **Figure 6.8 – Enter Diameter**

This any sized circle on the graphical display has one dimension. We can pick a size with the mouse or we can input the diameter using the textbox. We will type 2.25 and Enter. The circle will appear as shown in Figure 6.8.

Drawing a Concentric Circle

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Next, we will place a concentric circle on the drawing. We right click on the graphical display and we choose Center Point Circle
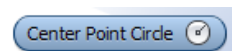
When we pick the Center Point Circle, we then point to the center of the 1.25 circle and a green dot will appear that represents the exact middle of the circle. We click on the green point and on the diameter textbox, we type 1.25 and we press Enter.
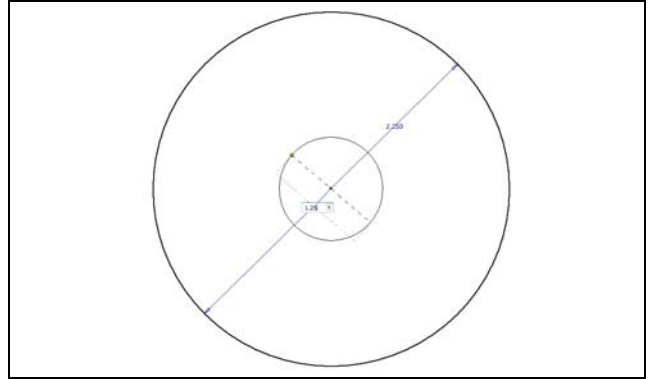

Figure 6.10 – Select the Center Point

Add Multiple Hubs

We will add three hubs to the drawing as shown in our original sketch, so we right click on the graphical display and we choose Center Point Circle
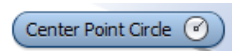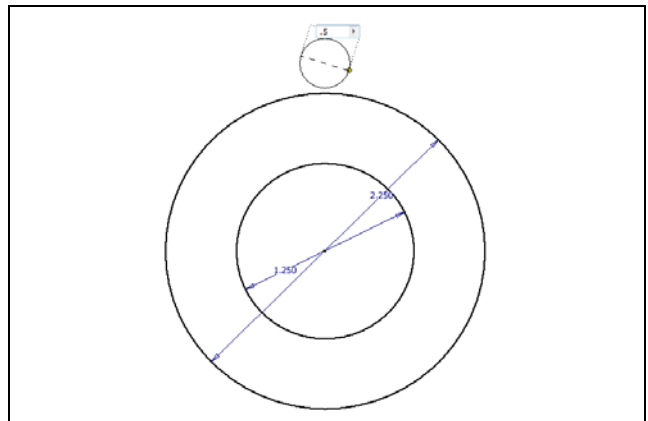

We place our cursor on the center point and then we move the mouse straight up from that point and above the top quadrant of the 2.25 circle. We will see a dotted line. We click anywhere above the circle on that line.
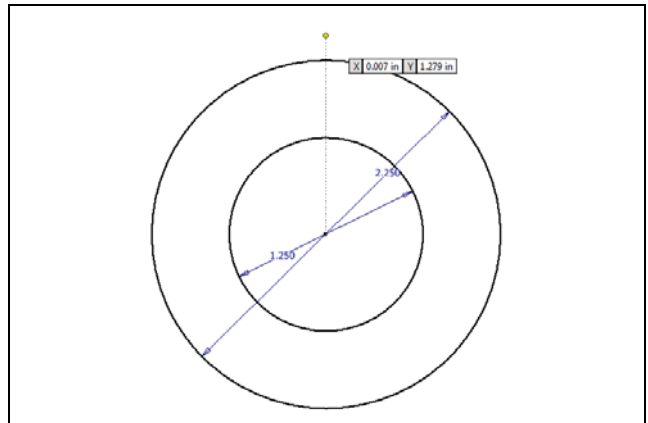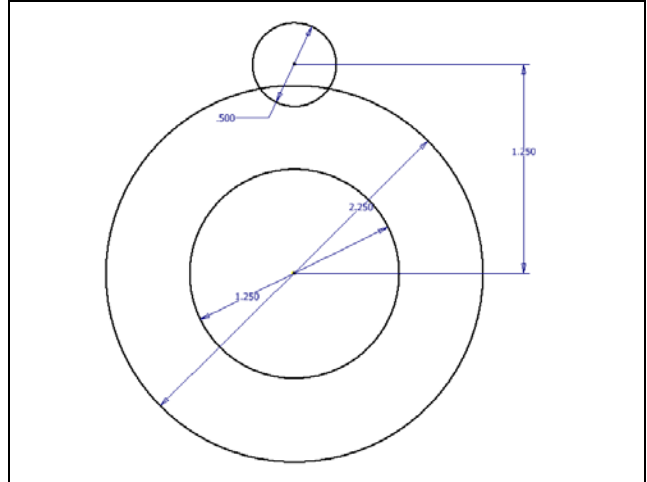

Figure 6.11 – Center of New Circle

We type 1.0 for the diameter of the circle and press Enter.

Figure 6.12 – Enter the Diameter

We add a dimension from the center of the 0.5 diameter circle to the center of the 2.25 diameter circle. We type 1.25 in the dimension textbox and the 0.5 circle will move into the position we want for it.


Figure 6.13 – Add a Dimension

We will then add a line from the west quadrant of the 0.5 diameter circle to anywhere straight down and on the inside of the 2.25 diameter circle. We do this again on the east quadrant of the 0.5 diameter circle. We should have two lines as shown in the figure.
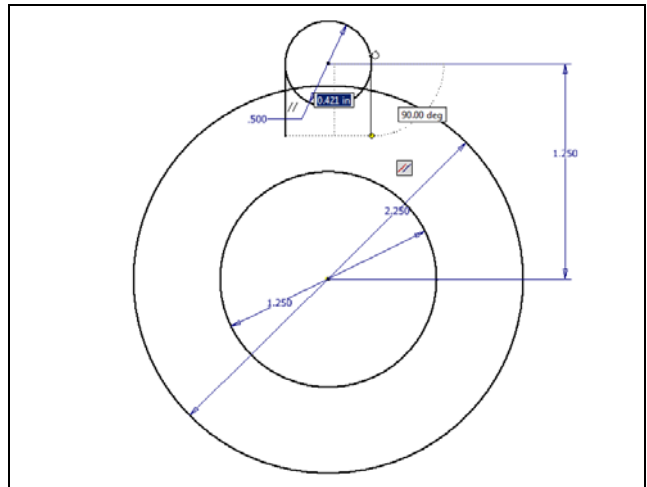

Figure 6.14 – Add Two Lines

We will trim the lines and arc segments away, so we choose the Trim button on the Inventor ribbon. By default, every entity becomes a cutting edge, we can pick the line, and it will appear as a dashed red line on the portion of the cutting edge with chose and a continuous red line on the other side. We click again on this side and the line will shorten to the edge of the circle.

We can trim both lines and remove the segments of the circles between the lines.
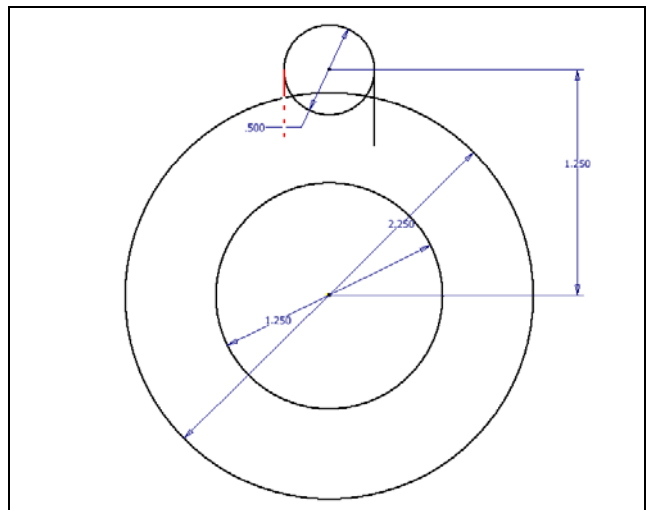
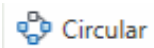**Figure 6.15 – Trim the Line**

We can see the hub now with the lines and circular segments removed. We will now array the hub so that there are three of them around the circle.
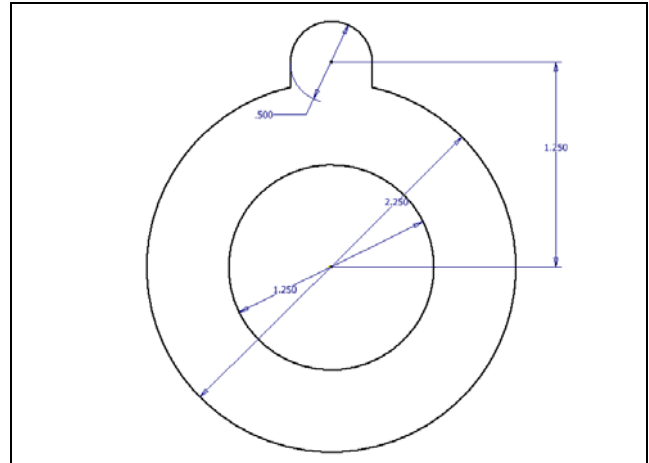

Figure 6.16 – Hub with Trimmed Lines

Array the Hub

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We want three of the hubs around the circle, so we can array the hub using a circular pattern.

When 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 two lines and the 0.5 diameter arc. Then we pick the Direction arrow and we choose the 2.25 diameter circle. An arrow will appear showing the direction of the array.
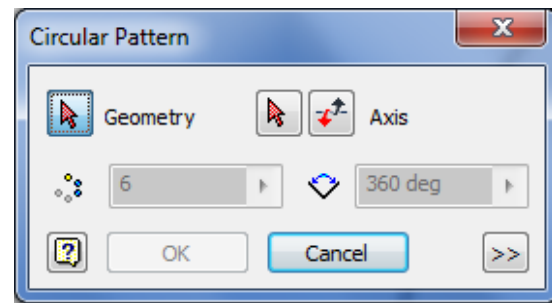
Figure 6.17 – Circular Pattern Window

We type 3 for the number of items in the pattern and we keep the angular dimension at 360 degrees. We can see the light features of the array and we press the OK button to make them permanent.
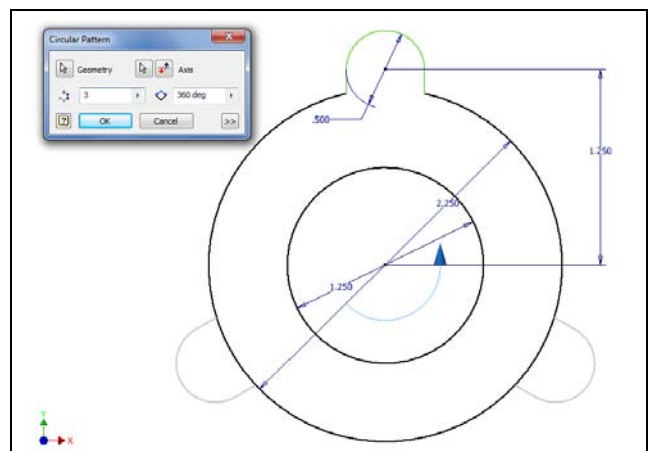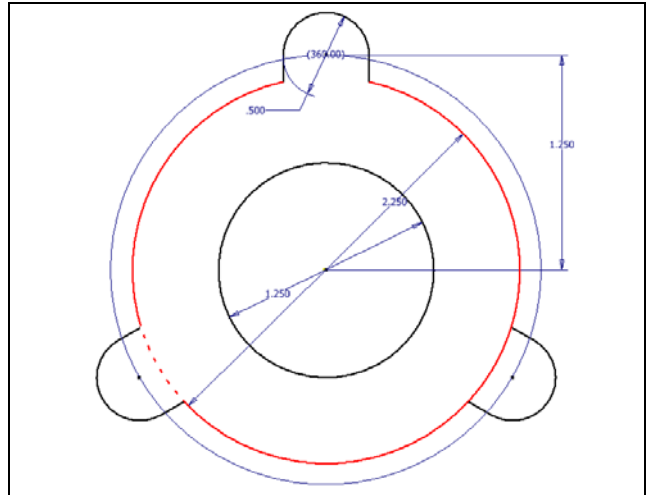

**Figure 6.18 – Circular Pattern Window**

We will trim the arc segments away, so again we choose the Trim button on the Inventor ribbon. By default, every entity becomes a cutting edge, we can pick the arc segment, and it will appear as a dashed red line on the portion of the cutting edge with chose and a continuous red arc on the other side. We click again on this side and the slice of arc will disappear.

We can trim both arc segments of the 2.25 circle between the lines.

Figure 6.19 – Circular Pattern Window

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.
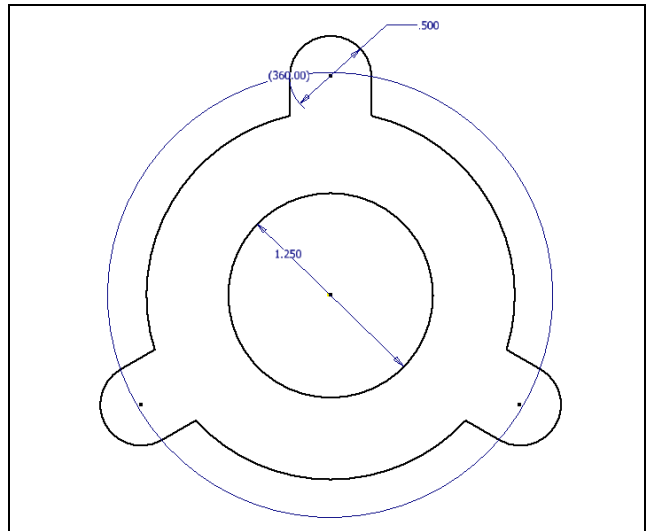
Figure 6.20 – First Sketch Complete

* World Class CAD Challenge 61-11 * - Close this drawing file. Create a New file and draw the circle, the concentric circle and the three hubs. 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

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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.
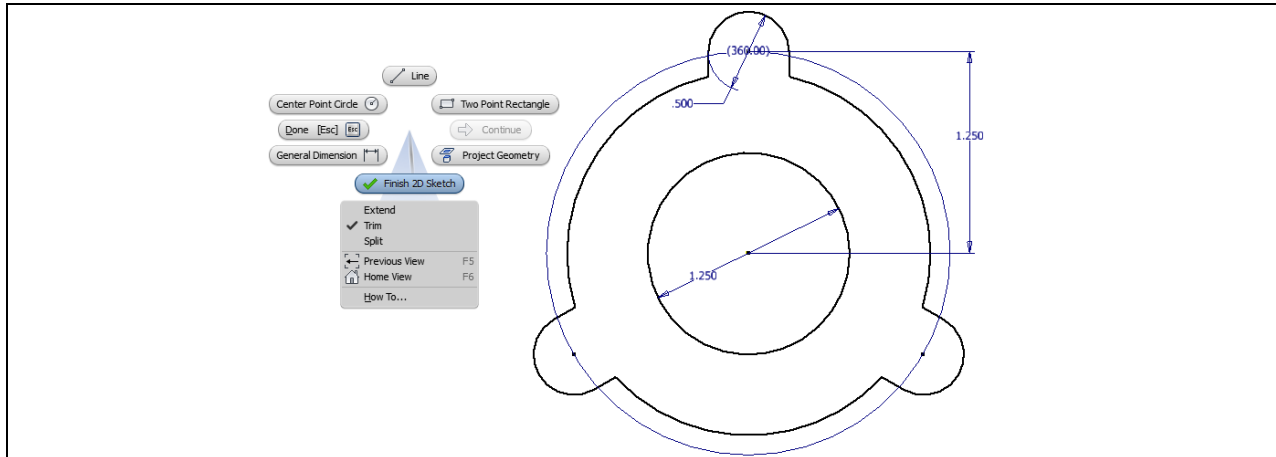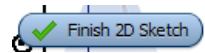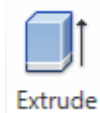

Figure 6.21 – Finished Part Two Sketch

Extruding a 3D Sketch

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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 choose solid. Next, we will change the Extents distance from 1.0 to 0.25. We change the direction of the extrusion to the right.
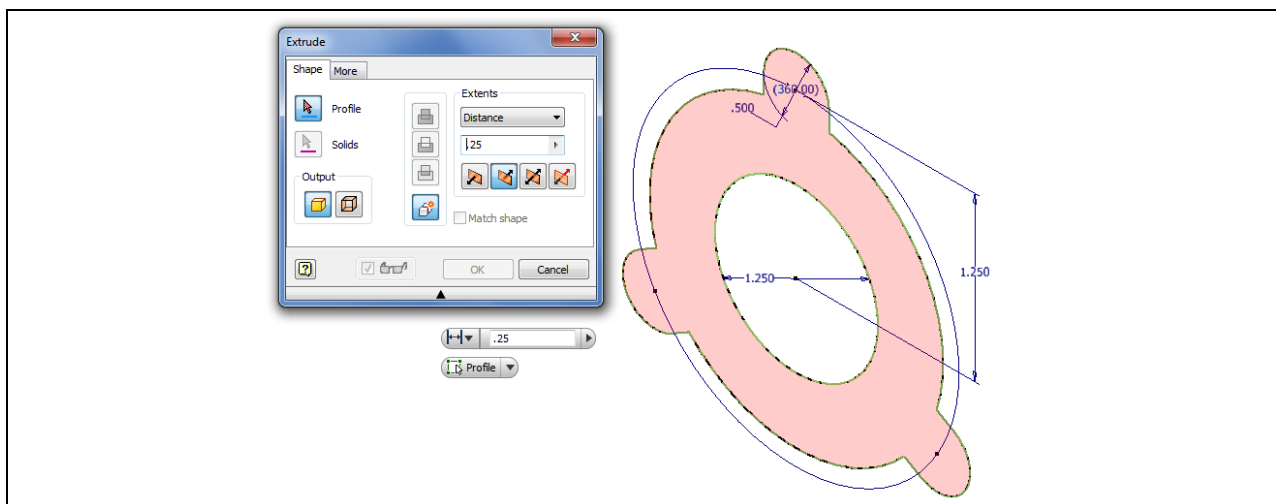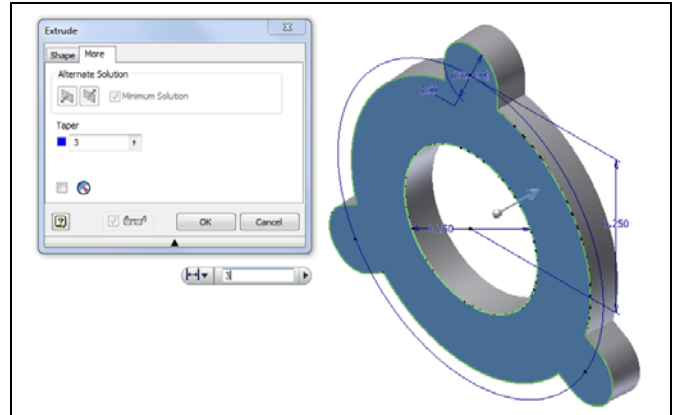

Figure 6.22 – The Extrude Window

This solid part will be cast, so we need to add a draft angle to the edge of the part. This allows the die cast or molded solid to be removed easily from a mold. We select the More tab on the Extrusion window and we change the taper from 0 to 5 degrees.
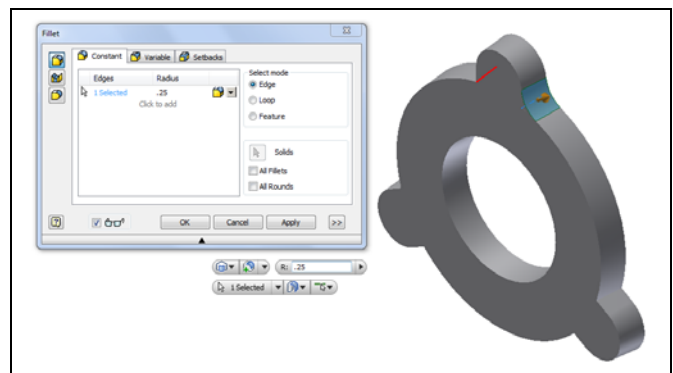

Figure 6.23 – Draft Angle of 5 Degrees

Add Multiple Fillets

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The next feature we will add to our solid is the six fillets. We choose the Fillet button on the Inventor ribbon and the Fillet window will appear on the graphical display.
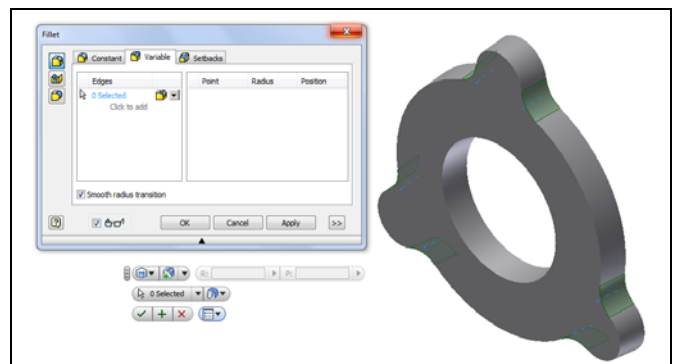
We set the fillet radius to 0.25. When we select the straight edged corner, the pointed edge will change to a 0.25 inch rounded corner.

Figure 6.24 – Fillet Window

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

Figure 6.25 – Six 0.25 Radius Fillets

Add another Extrusion
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We will need to add a smaller extrusion onto the first one to achieve the feature from our initial sketch. We begin the process by making a new sketch.
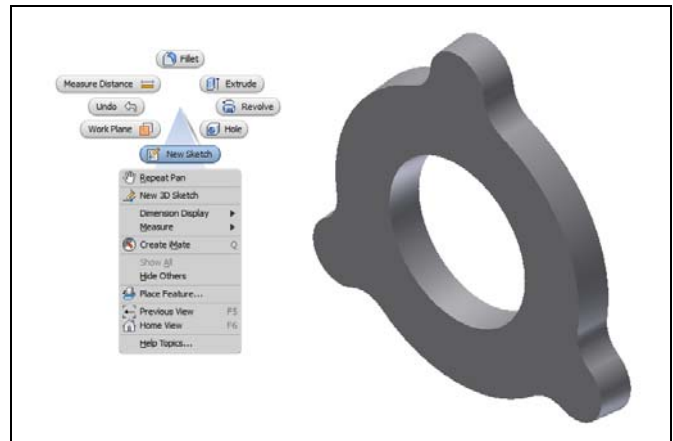

Figure 6.26 – A New Sketch

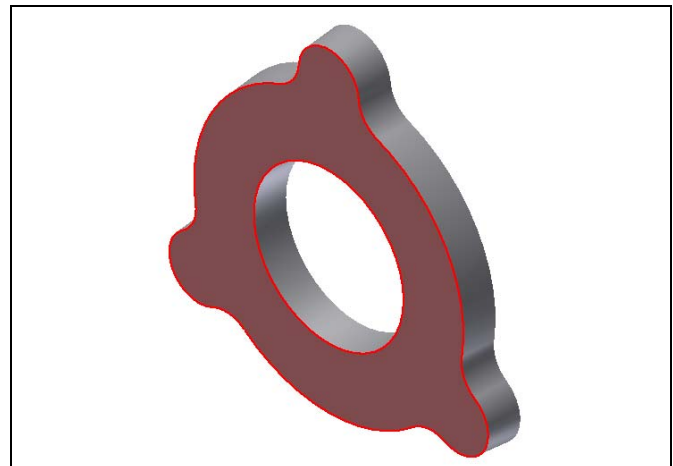We then choose the plane for the new sketch, so we pick the surface as shown in Figure 6.27.

Figure 6.27 – Select the Sketch Plane

We now pick the Center Point Circle, we then point to the center of the 2.25 circle and a green dot will appear that represents the exact middle of the circle. We click on the green point and in the diameter textbox, we type 1.25 and we press Enter.
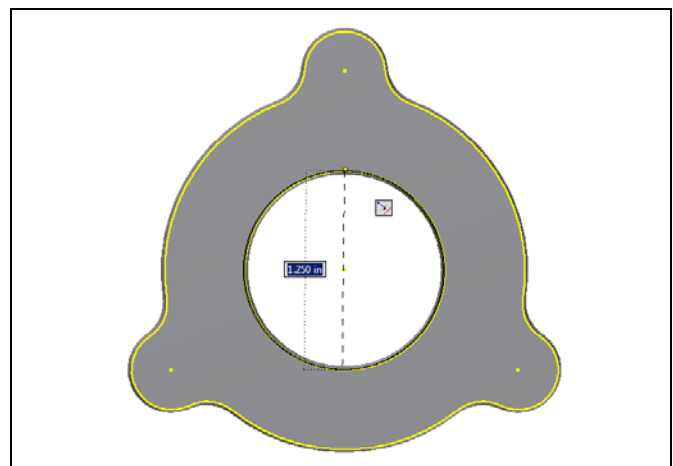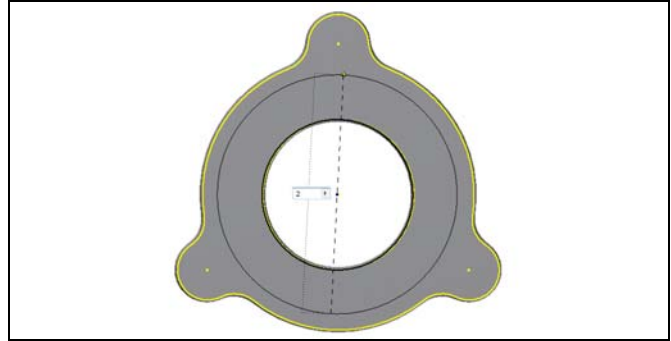

Figure 6.28 – Add the Internal Circle

To add another circle, we then pick the Center Point Circle, we point to the center of the 2.25 circle and a green dot will appear that represents the exact middle of the circle. We click on the green point, on the diameter textbox, we type 2.0, and we press Enter.


Figure 6.29 – Add the External Circle

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.
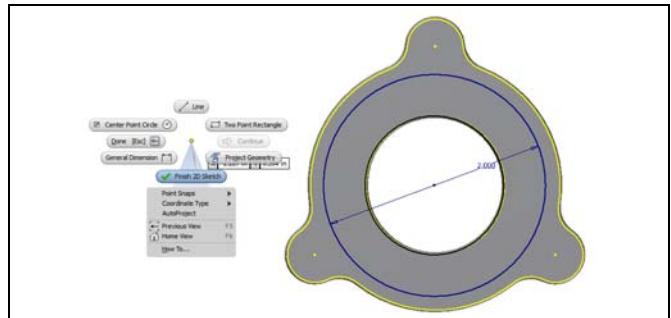

Figure 6.30 – Finish the 2d Sketch

To add this solid to the first one, 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 choose solid. Next, we will change the Extents distance from 1.0 to 0.25. We change the direction of the extrusion to the left.
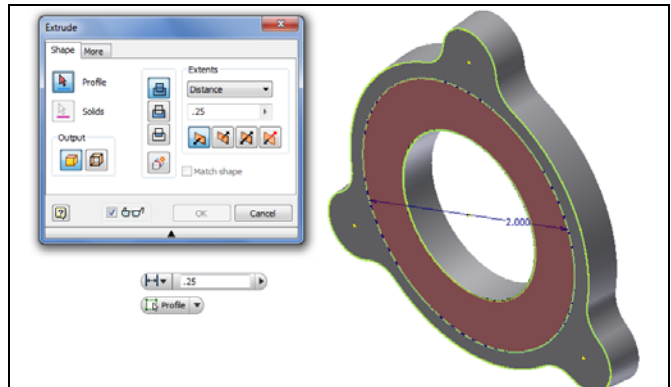

Figure 6.31 – The Extrusion Window

This solid part will be cast, so we need to add a draft angle to the edge of the part. This allows the die cast or molded solid to be removed easily from a mold. We select the More tab on the Extrusion window and we change the taper from 0 to -5 degrees.
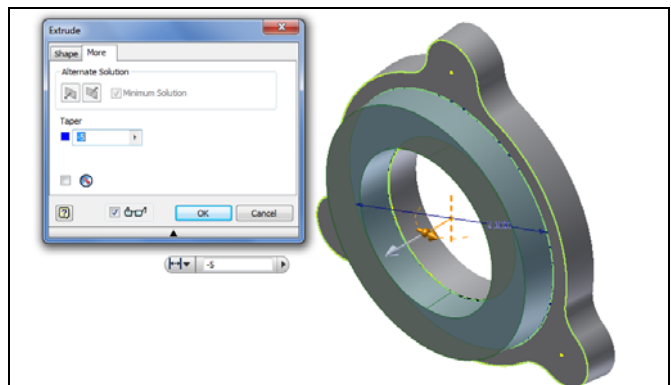

Figure 6.32 – Add the Draft Angle

Subtract another Extrusion

In this new section of training, we will create another solid and subtract its feature from the first solid. We begin by creating a plane that is offset from the front of the solid by 0.5 inch. We pick the Plane button on the Inventor ribbon and we choose Offset from Plane.
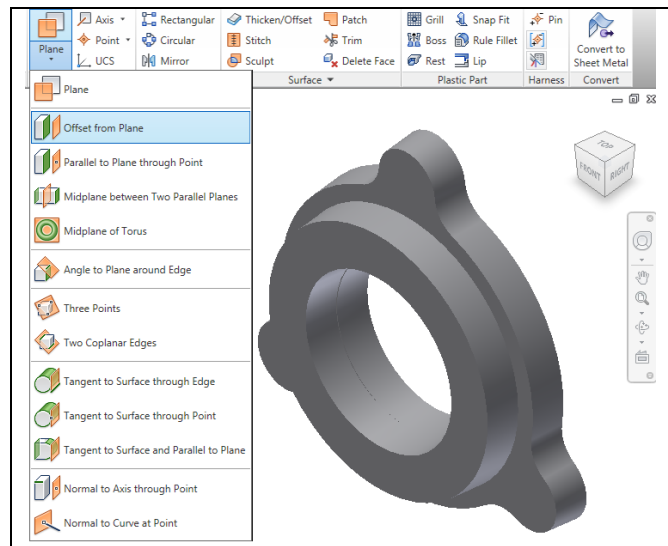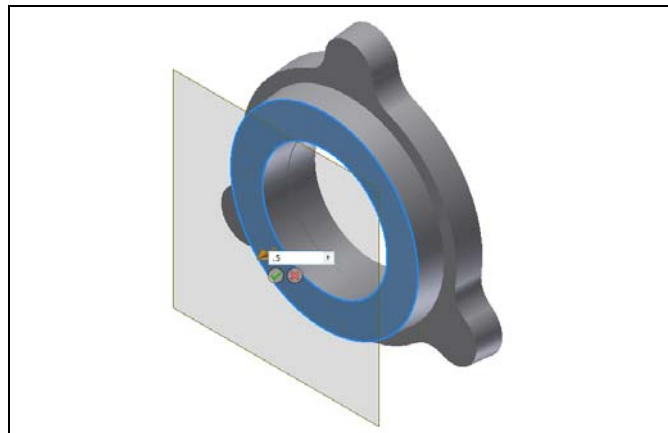
Figure 6.33 – Offset from Plane

We then pick the front plane as shown in the figure to reference.
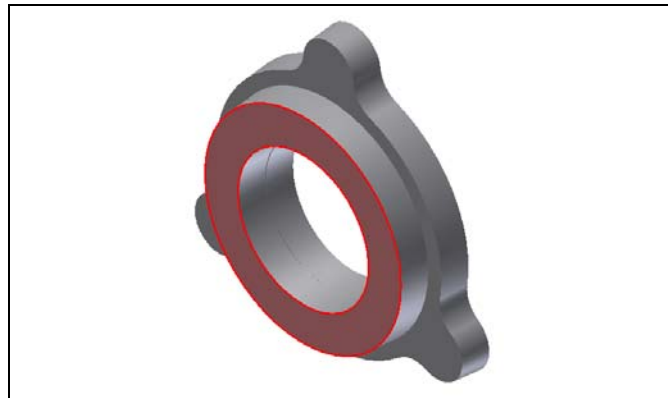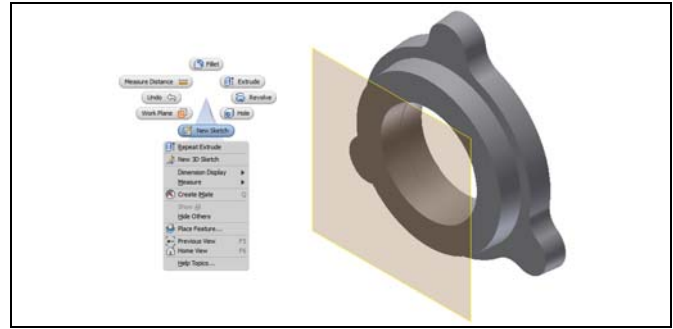
Figure 6.34 – Select the Reference Plane

We input 0.5 in the measurement textbox and the new plane will acquire that position in front of the solid part.
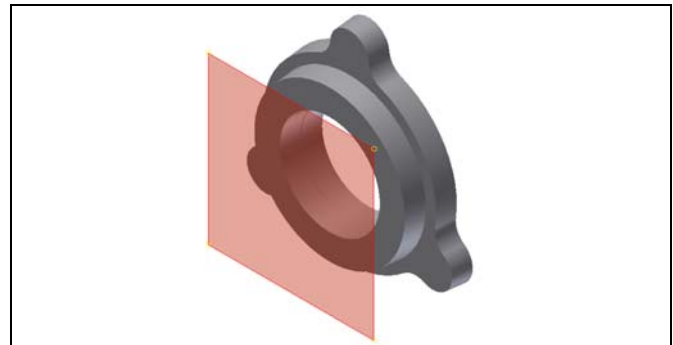
Figure 6.35 – Position the New Work Plane

We will need to create a spherical solid on this plane and to subtract it from the solid part. We begin the process by making a new sketch.


Figure 6.36 – New 2D Sketch

We then choose the plane for the new sketch, so we pick the new work plane surface as shown in Figure 6.37.


Figure 6.37 – Select the Plane

We now pick the Center Point Circle, we then point to the center of the 2.25 circle and a green dot will appear that represents the exact middle of the circle. We click on the green point and on the diameter textbox, we type 2 and we press Enter.
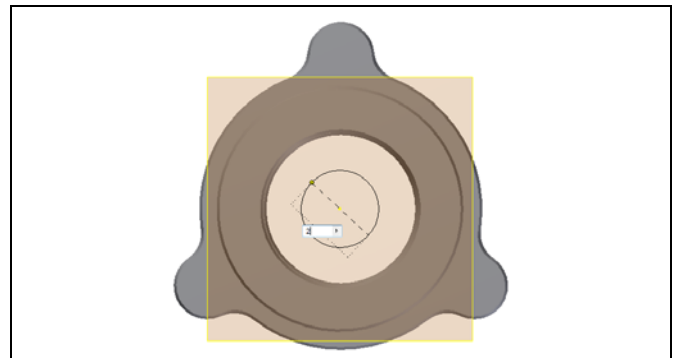

Figure 6.38 – Draw a 2.0 Diameter Circle

We draw a line from the top quadrant of 2-inch diameter circle to the bottom quadrant.
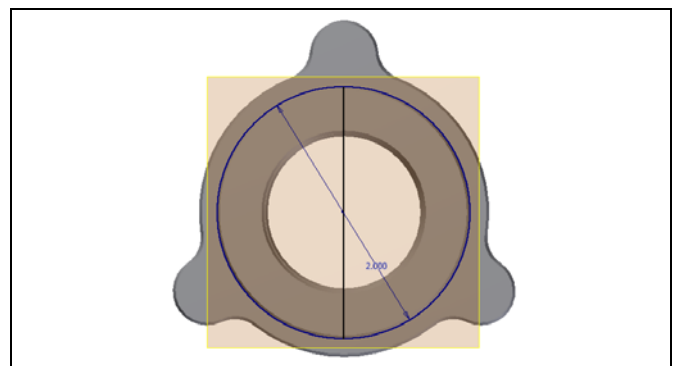
Figure 6.39 – Draw a 2.0 Line

We will trim the right arc segment away, so again we choose the Trim button on the Inventor ribbon. By default, every entity becomes a cutting edge, we can pick the arc segment, and it will appear as a dashed red line on the portion of the cutting edge with chose and a continuous red arc on the other side. We click again on this side and the slice of arc will disappear.
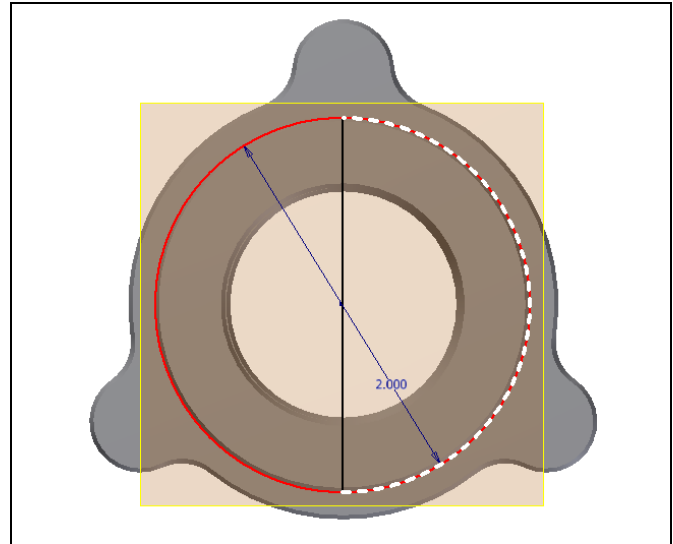

Figure 6.40 – Trim Away the Right Arc

Before we revolve the sphere, we need to right click on the graphical display and on the menu; we choose the Finish 2D Sketch button.
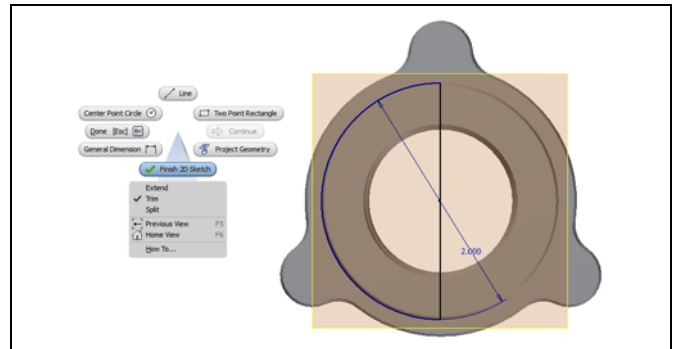
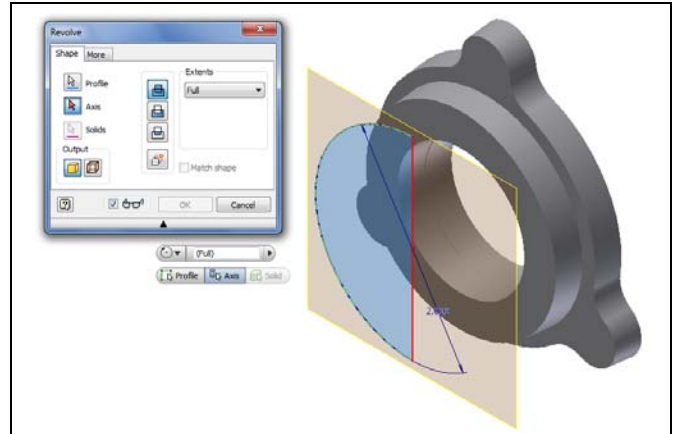**Figure 6.41 – Finish the 2D Sketch**

To subtract this solid from the first one, 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.
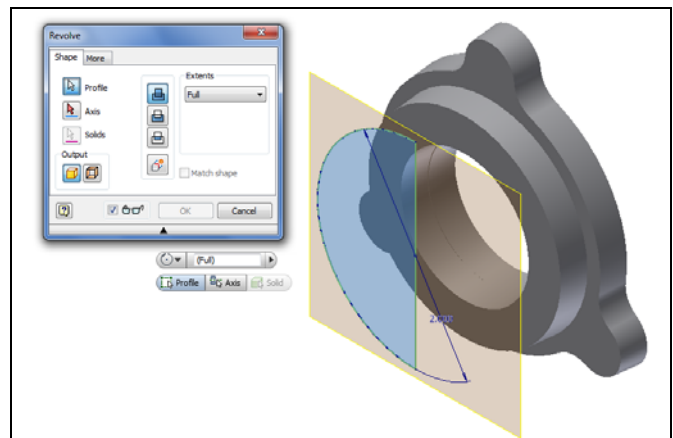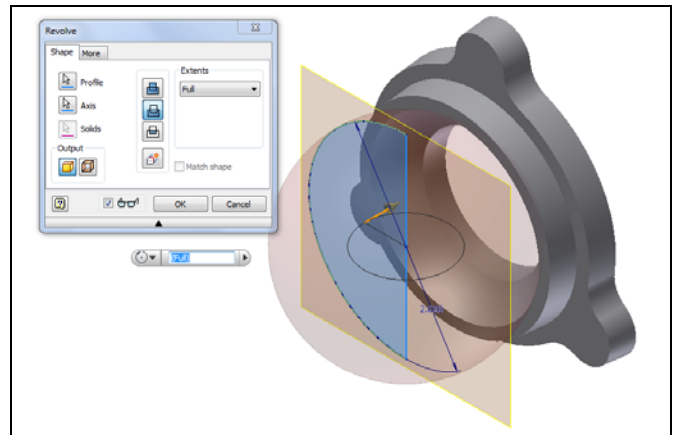

Figure 6.42 – The Revolve Window

Next, we will pick the Axis button and select the 2.0 line on the offset work plane.
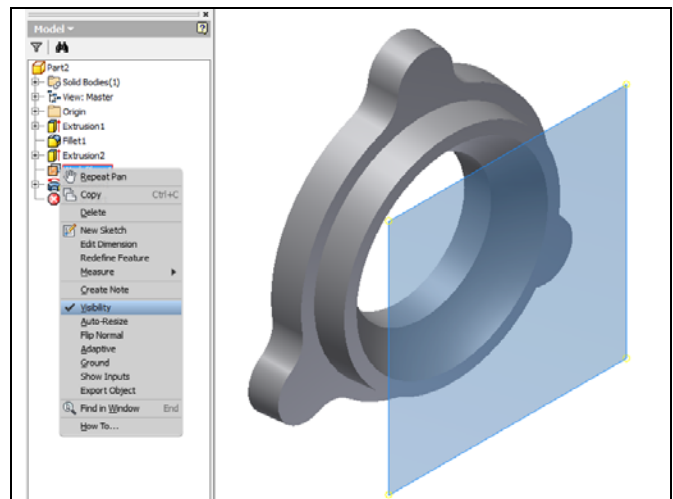

Figure 6.43 – Pick the Revolve Axis

Now, we will pick the Cut button. We can see the material that will be deleted from the original solid. We press the OK button to make the alteration permanent.

Figure 6.44 – Remove the Solid

We are done with the offset work plane, so we right click on the work plane in the left pane and we pick Visibility to uncheck it.

Figure 6.45 – Make the Work Plane Invisible

Draw a Tapped Hole

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We will need to add 3 tapped holes onto the solid, so we begin the process by making a new sketch.
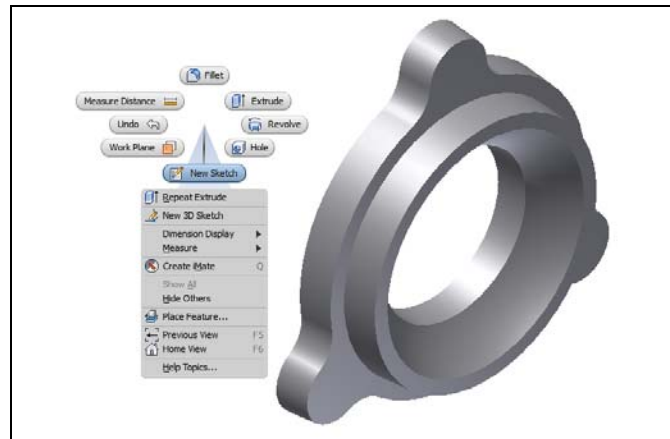

Figure 6.46 – New 2D Sketch

We then choose the plane for the new sketch, so we pick the surface as shown in Figure 6.47.
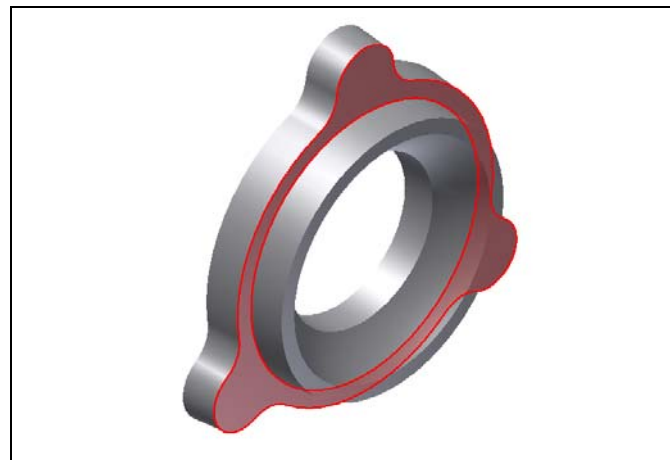

Figure 6.47 – Choose the Plane

Now, we will select Point on the Sketch tab of the Inventor ribbon. Then we place the cursor over the yellow point that is the center points for each hub as shown in the figure.
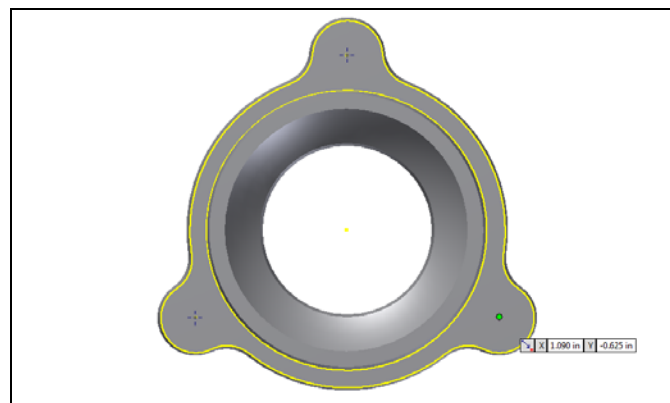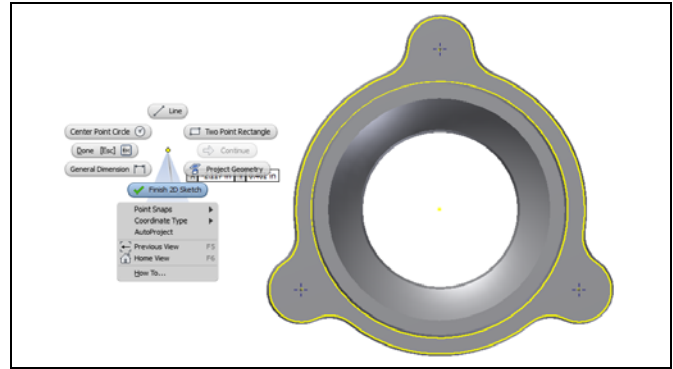

Figure 6.48 – Add Center Points

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

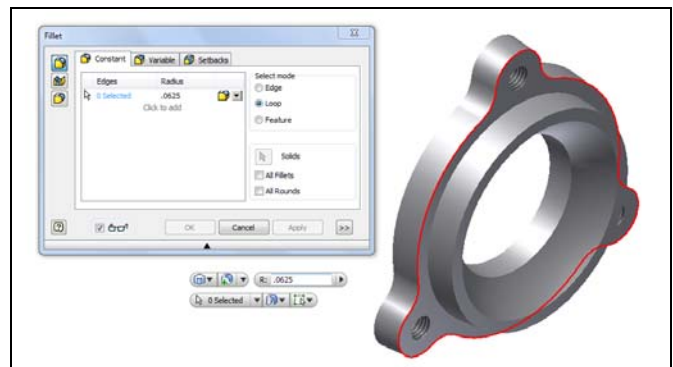**Figure 6.49 – The Solid Part**

We then select the Hole button on the Inventor ribbon and the Hole window will appear on the graphical display. We are making a ¼-28 tapped hole, so we choose the tapped hole icon. We pick ANSI Unified Screw Threads, 0.25 for the size, 2B for the class and ¼-28 UNF for the designation. The hole that automatically appeared will appear tapped and we press the OK button to retain the feature.
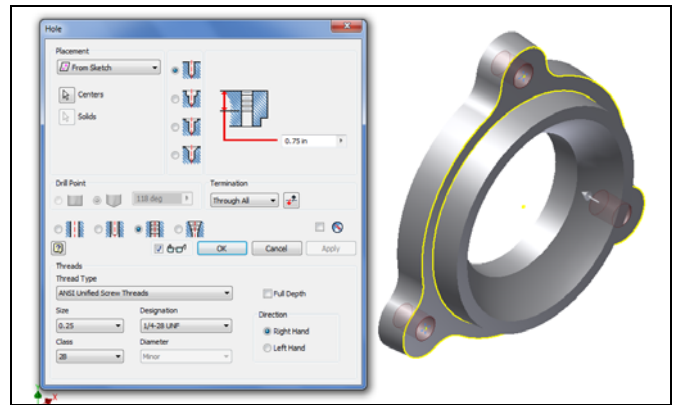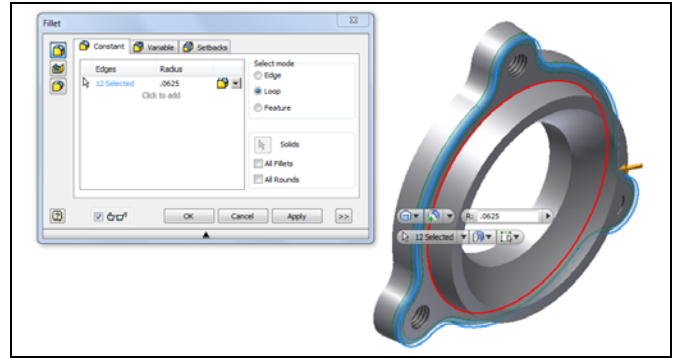

Figure 6.50 – The Solid Part

Add Fillets on the Loop

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The next feature we will add to our solid is the five fillets on the continuous loop. We choose the Fillet button on the Inventor ribbon and the Fillet window will appear on the graphical display. We make the fillet radius 0.0625 and the mode to loop. We pick the edge as shown in the figure and we can see the 0.0625 rounded sections appear.
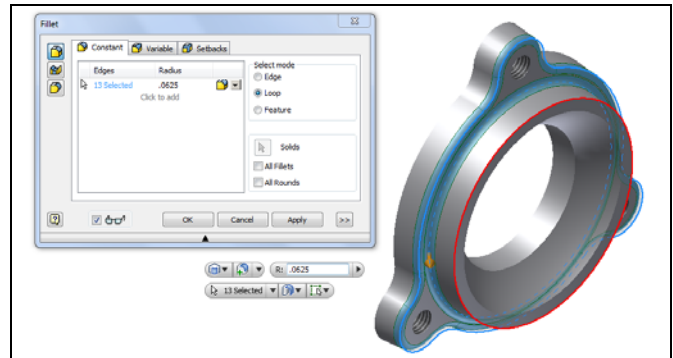

Figure 6.51 – The Solid Part

We pick the inside edge as shown in the figure and we can see the 0.0625 rounded section appear.
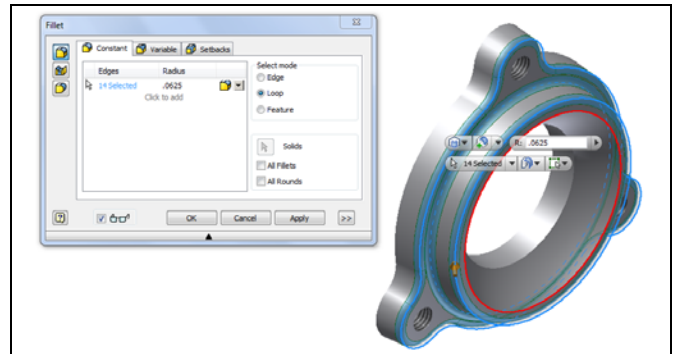

Figure 6.52 – The Solid Part

We pick the top outside edge as shown in the figure and we can see the 0.0625 rounded section appear.
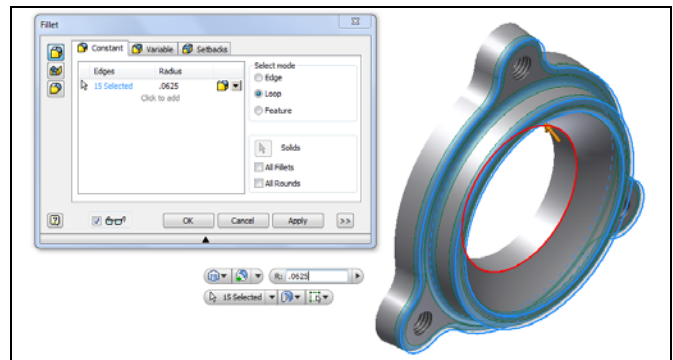

Figure 6.53 – The Solid Part

We pick the top inside edge as shown in the figure and we can see the 0.0625 rounded sections appear.

Figure 6.54 – The Solid Part

We pick the inside edge as shown in the figure and we can see the 0.0625 rounded section appear.

We press the OK button and the fillets become permanent.

Figure 6.55 – The Solid Part

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

Figure 6.56 – Finished Solid

* World Class CAD Challenge 61-11 * - Close this drawing file. Create a New file and draw the circle, the concentric circle and the three hubs. Extrude the profile and add the fillets. Add the next extrusion and subtract the sphere. Add the three tapped and holes. Add the continuous 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.

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