

# Chapter

# 3

## Solid Part Two

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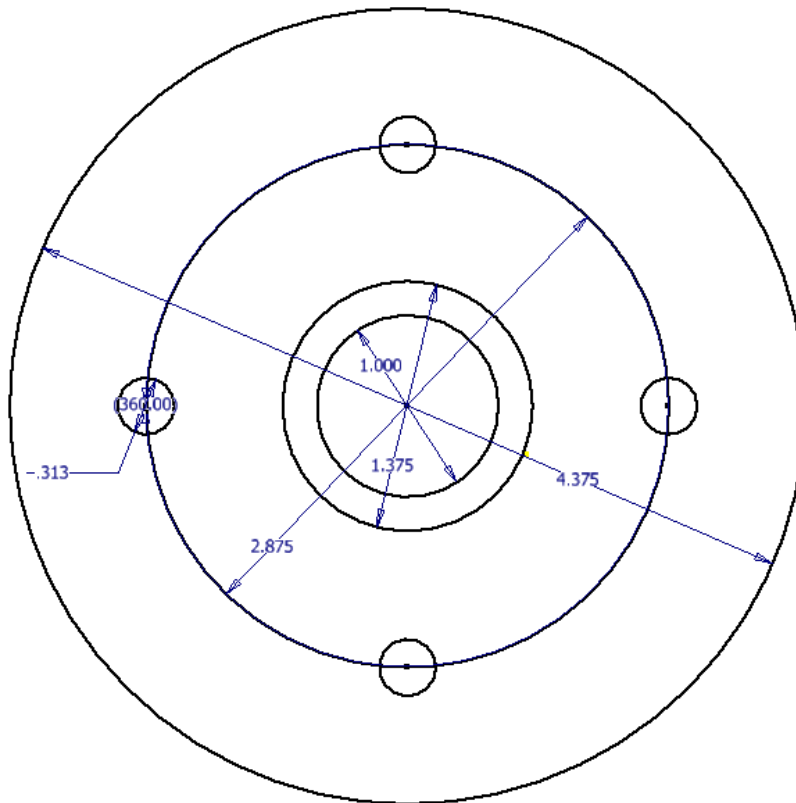
**This chapter will cover the following to World Class standards:**

- **Sketch of Solid Problem**
- **Draw a Circle**
- **Draw a Concentric Circles**
- **Draw a Circle on a Construction Line**
- **Draw a Circular Array**
- **Finish the 2D Sketch**
- **Extrude a 2D Sketch**
- **Add a Sketch on a Model**
- **Extrude another 2D Sketch**

## Sketch of a Solid Part

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Again we start any project by making a sketch, so we can efficiently produce a drawing. In part 2, we see a sketch with eight circles. The outside diameter 4.375 inches. At the center of the sketch, we will produce the geometry for a 1.375 inch hub with a 1 inch diameter. Between the hub and the outside diameter, we have four holes separated evenly on a circular array. The four holes are on a 2.875 bolt hole circle. A bolt hole circle is an imaginary diameter that passes through the center point of each hole.

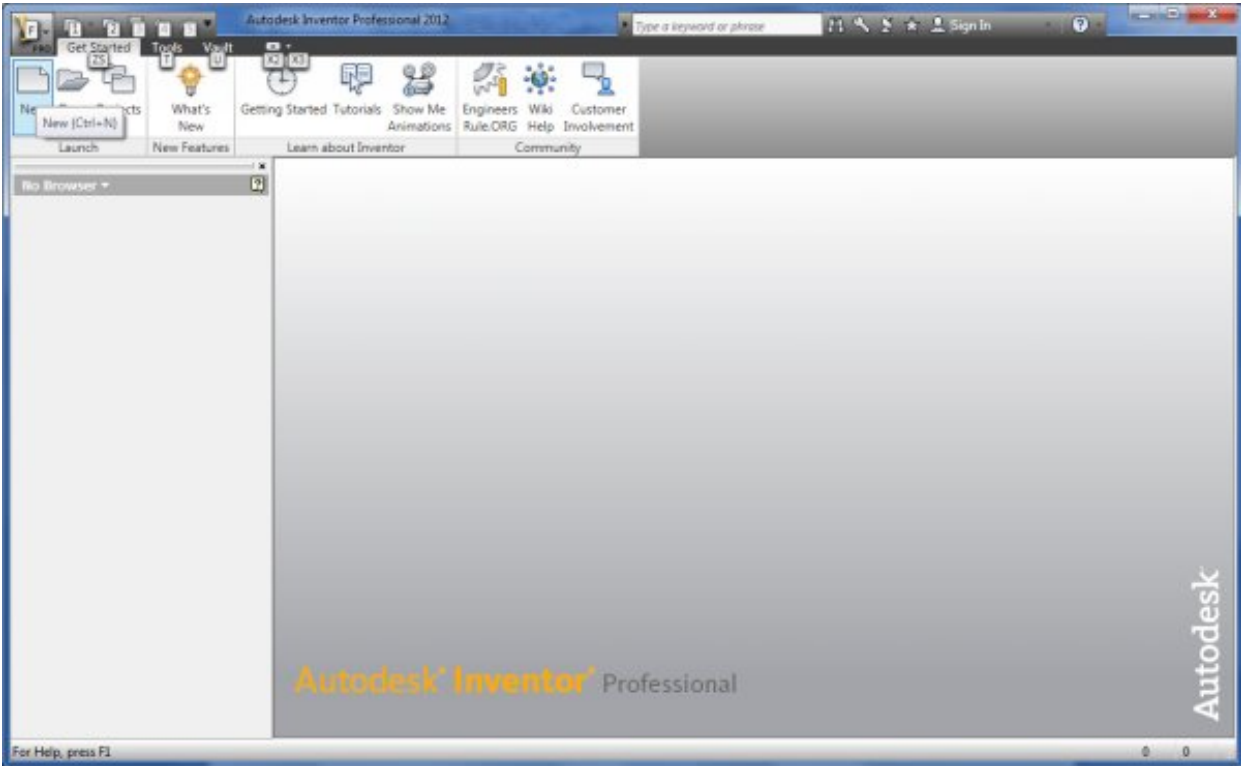


**Figure 3.1 – Problem Two Sketch**

In the second problem, we will practice techniques that we learned in the first part sketch and add some new experiences such as using construction lines.

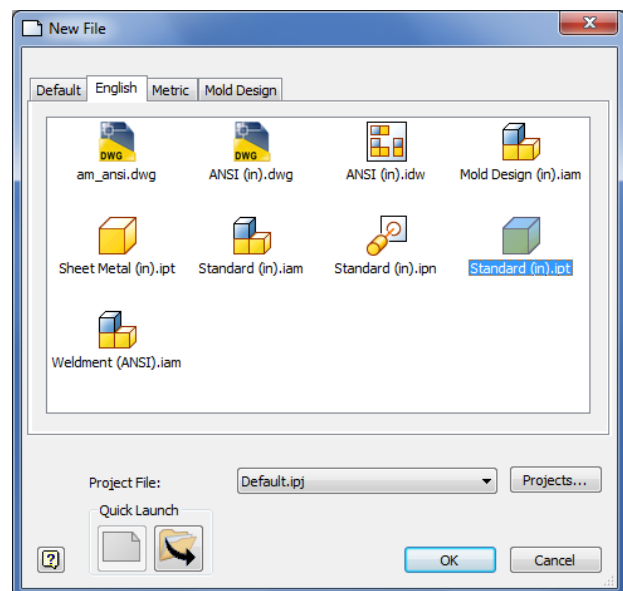
## Starting a 3D Part Drawing Sketch

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



**Figure 3.2 – 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 3.3 – 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.

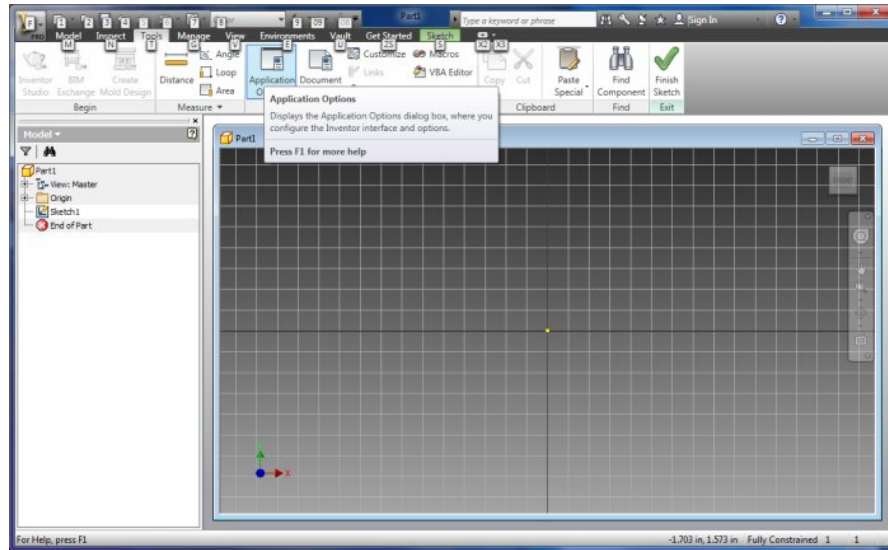


Figure 3.4 – Starting the drawing using the Standard IPT template

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.

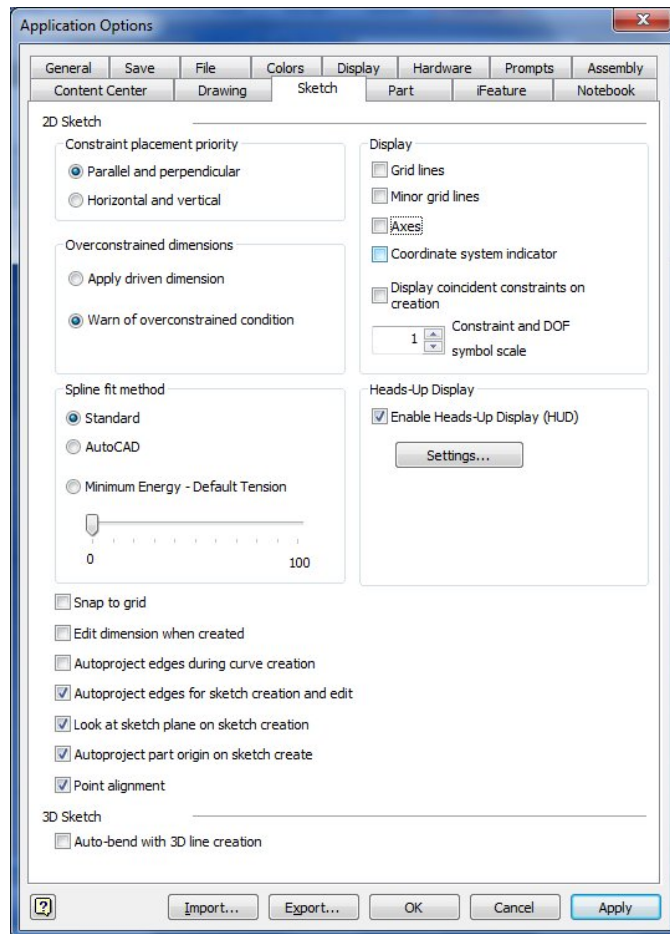


Figure 3.5 – Application Options Window

## Drawing a Circle

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 and then we expand the circle on the display.

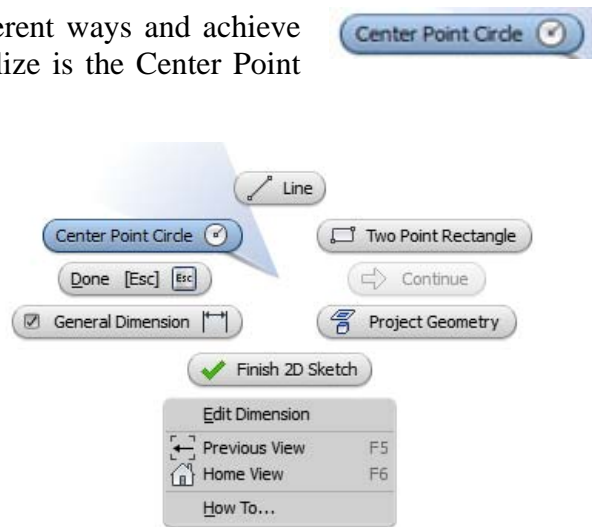


Figure 3.6 – Graphical Display Menu

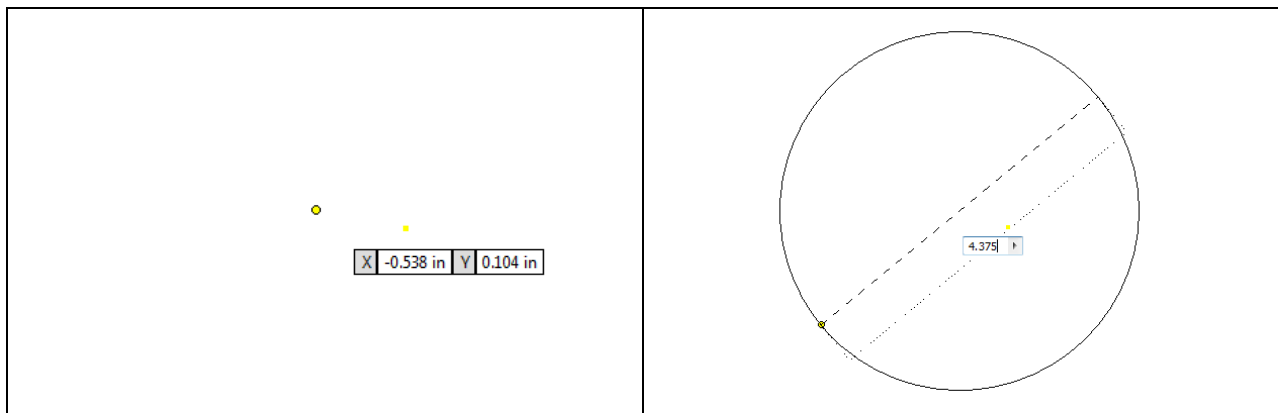


Figure 3.7 – Pick a Center Point

Figure 3.8 – 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 4.375 and Enter. The circle will appear as shown in figure 3.8.

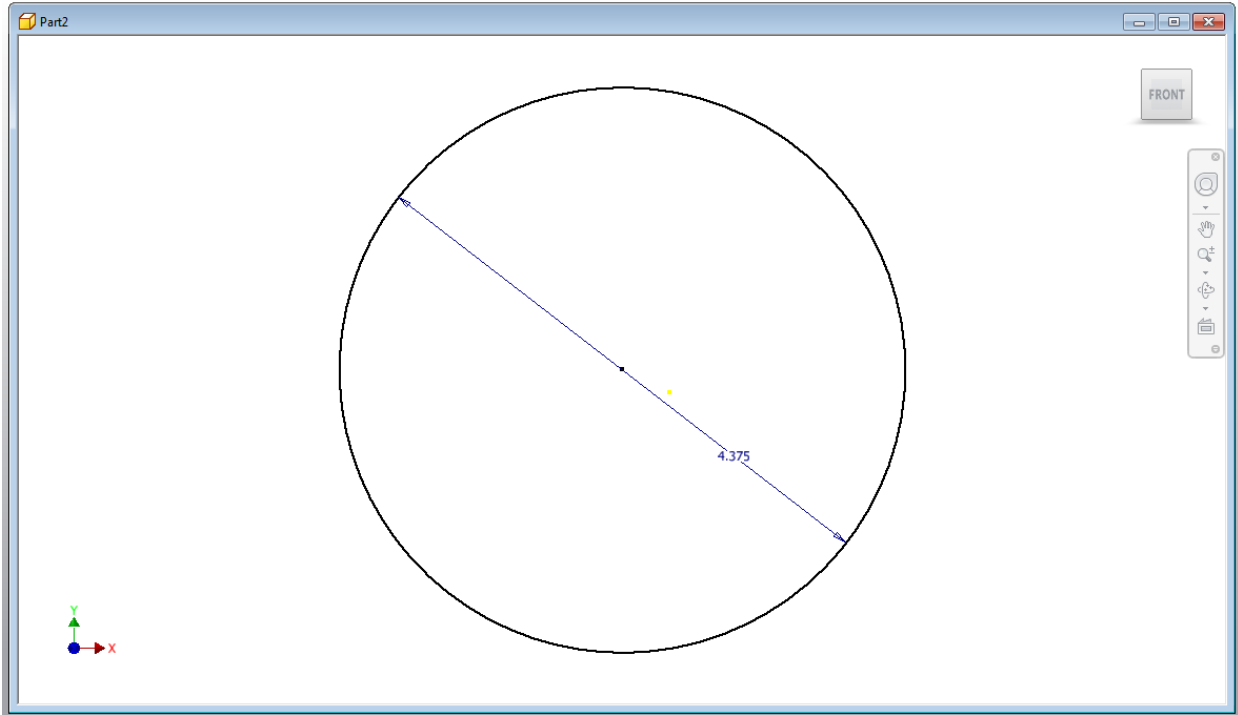
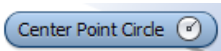


Figure 3.9 – Circle with Dimension Showing

## Drawing a Concentric Circle

Next we will learn to 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 4.375 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 1.0 and we press Enter.

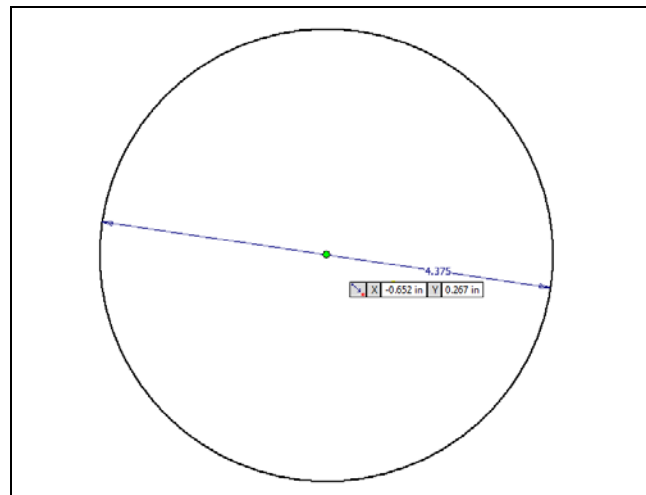


Figure 3.10 – Select the Center Point

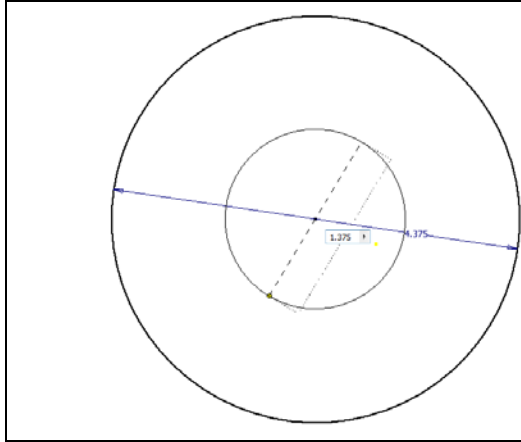


Figure 3.11 – Diameter Textbox

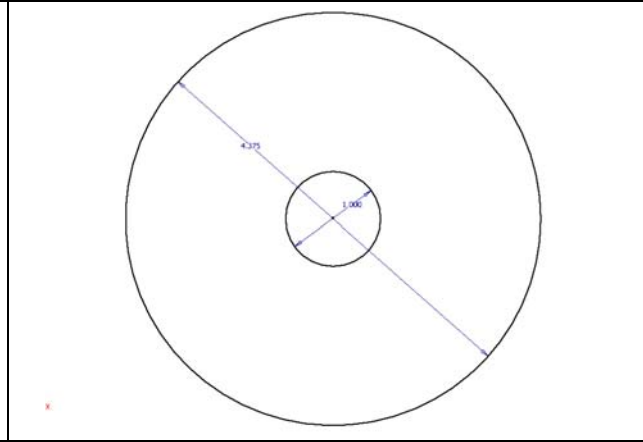
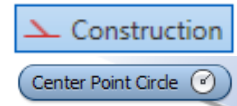


Figure 3.12 – Concentric Hub

## Drawing a Circle using a Construction Line

Next we will repeat drawing 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



We then point to the center of the 4.375 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 2.875 and we press Enter. We can see that the construction line is brown.

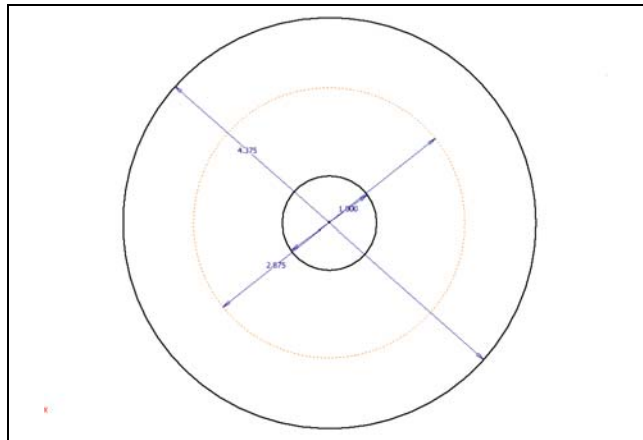
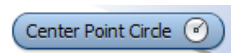


Figure 3.13 – A Construction Line

## Add a Bolt Hole

The last set of circles begins with making a single 0.3125 diameter bolt hole. 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 left quadrant of the brown construction bolt circle. When we place the cursor on the west quadrant we will see a yellow dot. We click on the yellow point and on the diameter textbox we type 0.3125 and we press Enter.

There are four quadrants on a circle in the north, east, south and west positions.

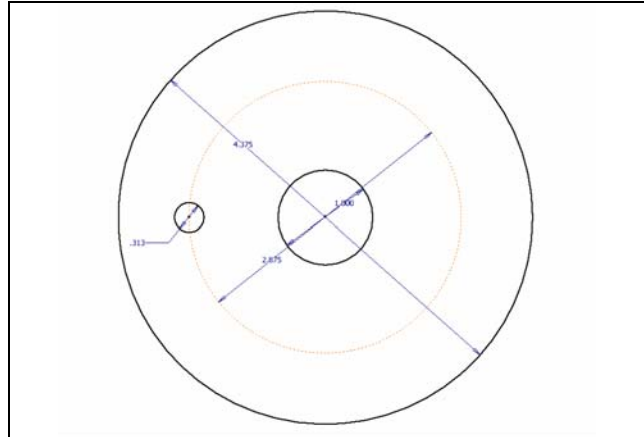
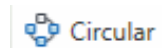


Figure 3.14 – Change the Dimension

## Array the Circle

We can array the last entity using a rectangular or circular pattern. This array we will do is the circular configuration.



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 0.3125 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.

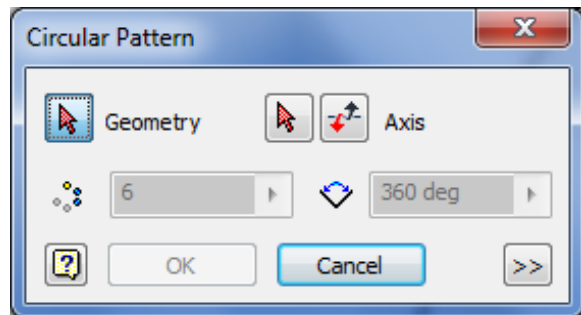


Figure 3.15 – Circular Pattern Window

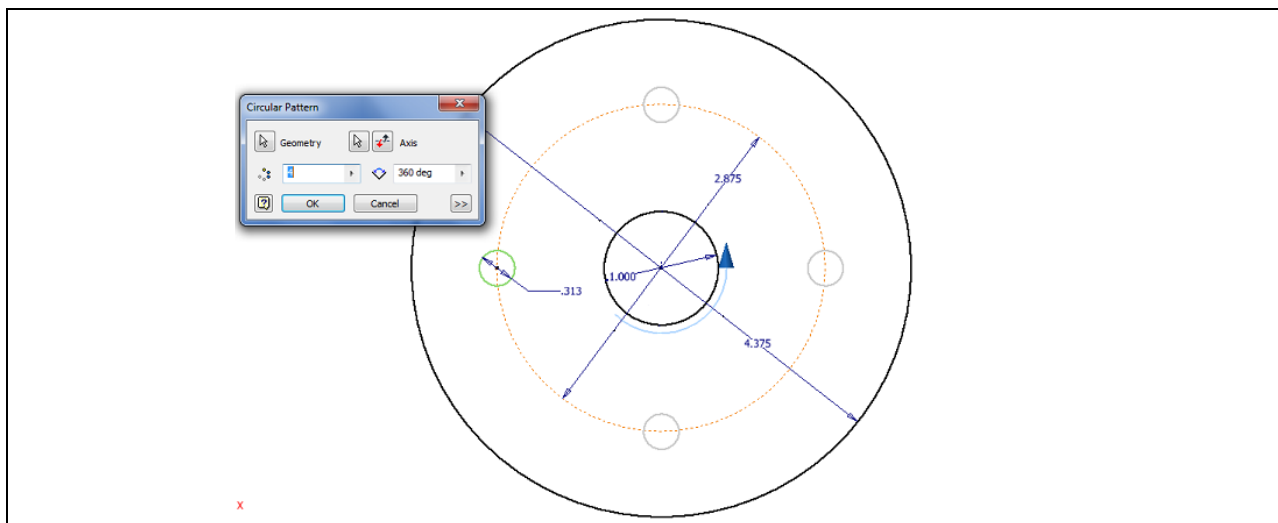
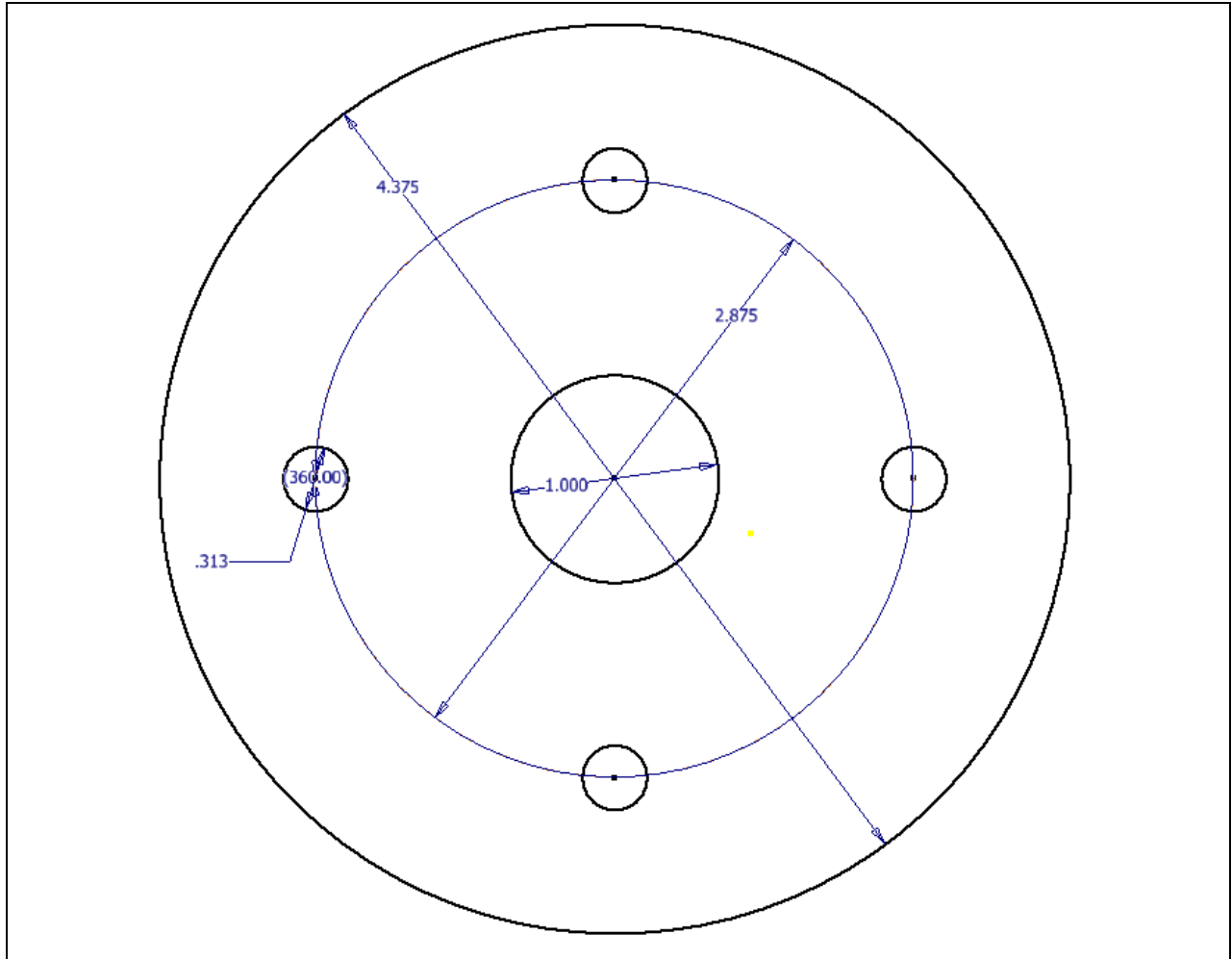


Figure 3.16 – Array the Circle



We will type 4 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 90 degrees between each 0.3125 diameter circle. We press the OK button to add the entities.

The sketch now meets the requirements given at the beginning of the chapter.



**Figure 3.17 – The Finished Part**

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.

**\* World Class CAD Challenge 61-04 \* - Close this drawing file. Create a New file and draw the circle, the two concentric circle and the four holes on the bolt circle. 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 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.

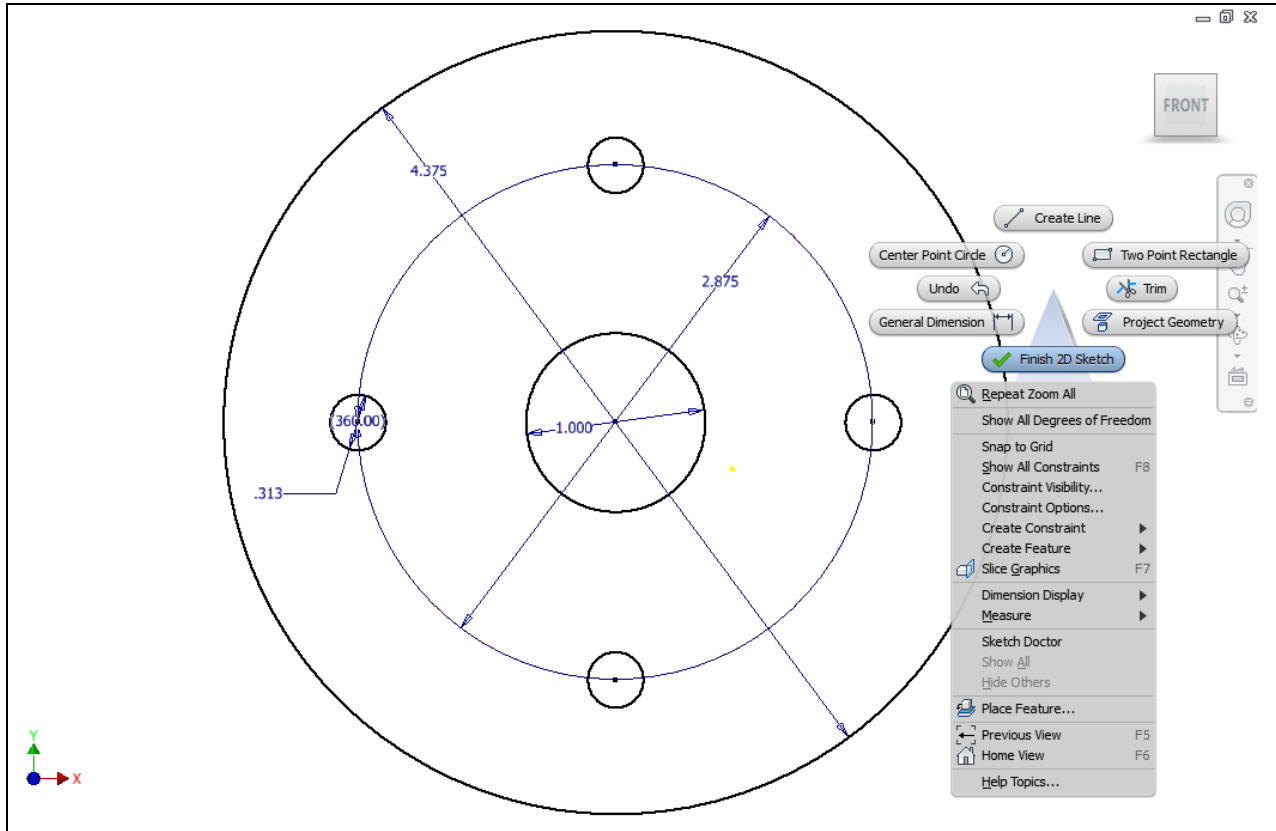
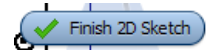
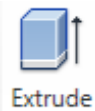


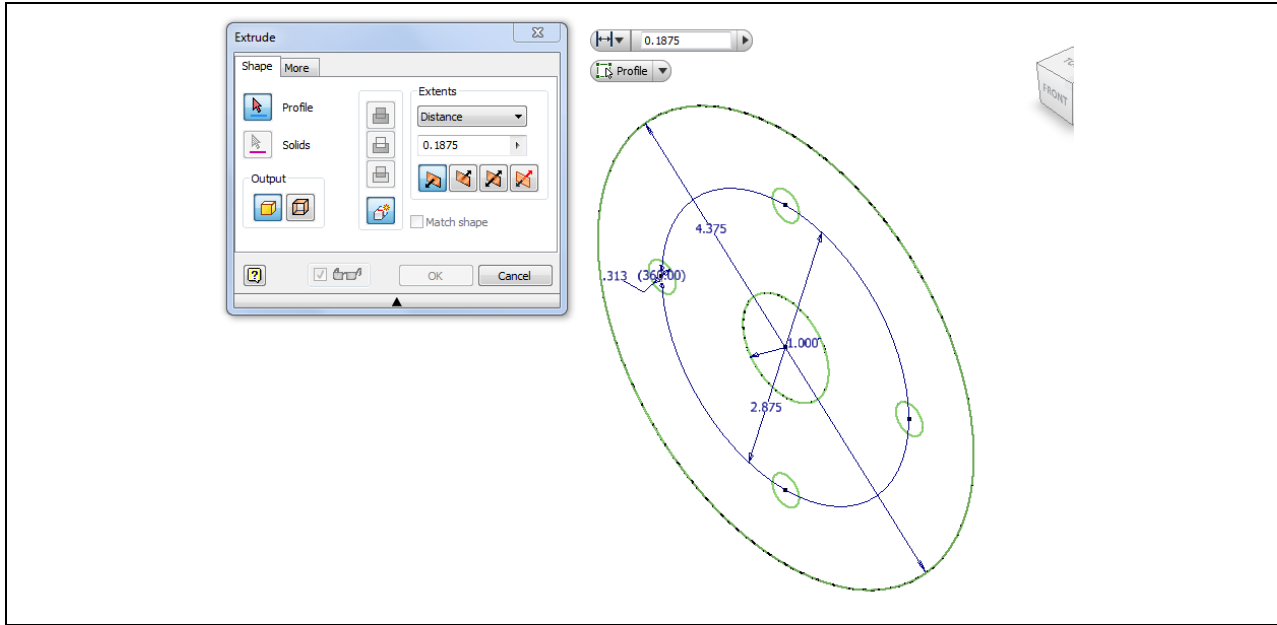
Figure 3.18 – Finished Part Two Sketch

## Extruding a 3D 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 Inventor ribbon. The Extrude window will appear on the display.

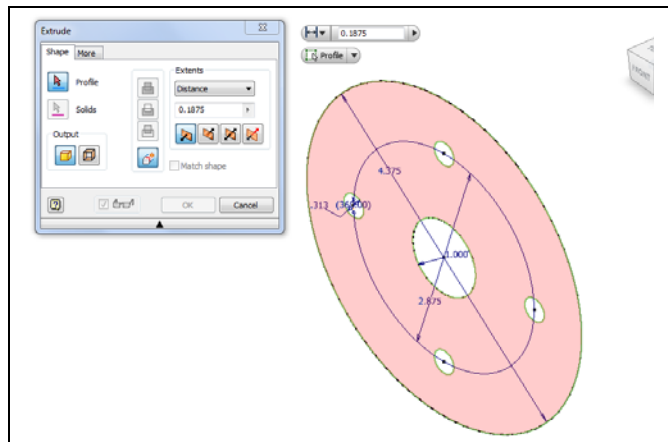


On the Extrude window, we can output either a solid or surface. The difference between the two are that the first is like a hard piece of aluminum and the second choice is similar to a box. We will pick the Solid output on the left. Next, our part will be made from finished aluminum, so we will change the Extents distance from 1.0 to 0.1875.



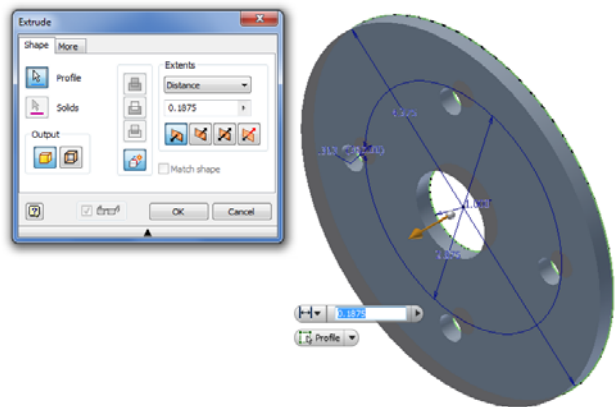
**Figure 3.19 – The Extrude Window**

Next, we will click inside the large circle and it will become highlighted in red as shown. Click on the red area again and the 2D polygon will become a solid with a hole where the circle was located.



**Figure 3.20 – Highlight Disk**

Then we will see the solid cylinder.



**Figure 3.21 – Highlight Disk**

## Create and Extrude a New 2D Sketch

We right click on the graphic display and we choose the New Sketch button. We will be prompted to select what plane to make our sketch on and we will pick the top plane. This is the one closest to us.

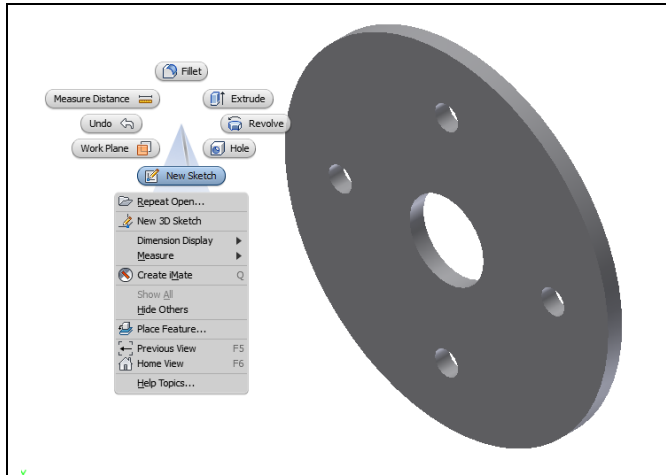


Figure 3.24 – Highlight Left Tab

We then will draw two concentric circles on the center point of the disk.

One circle will be 1.375 diameter.

The inside circle will be 1.0 diameter.

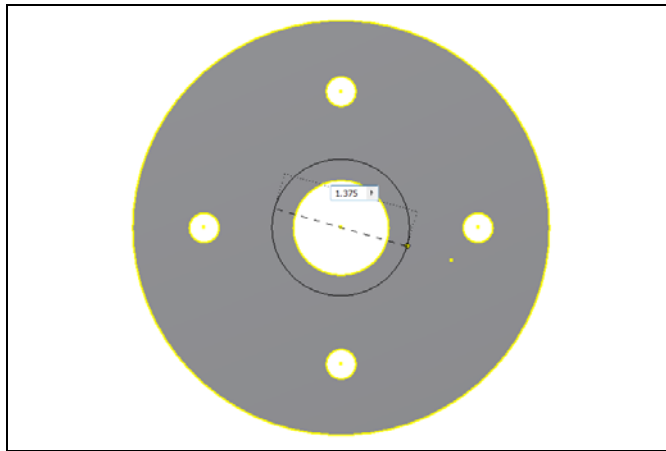


Figure 3.25 – Highlight Plate

Now that we have a finished sketch, we need to extrude the new section. 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.

Then we will click in between the two circles and it will become highlighted. Click on the area again and the 2D polygon will become a solid with a hole where the circle was located. We input 0.8125 for the distance and we press the OK button.

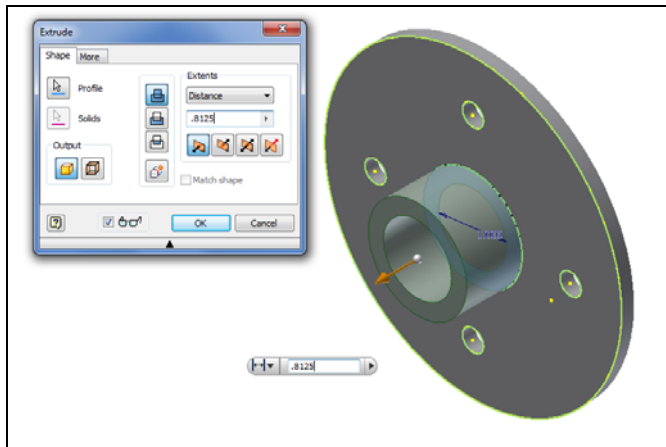
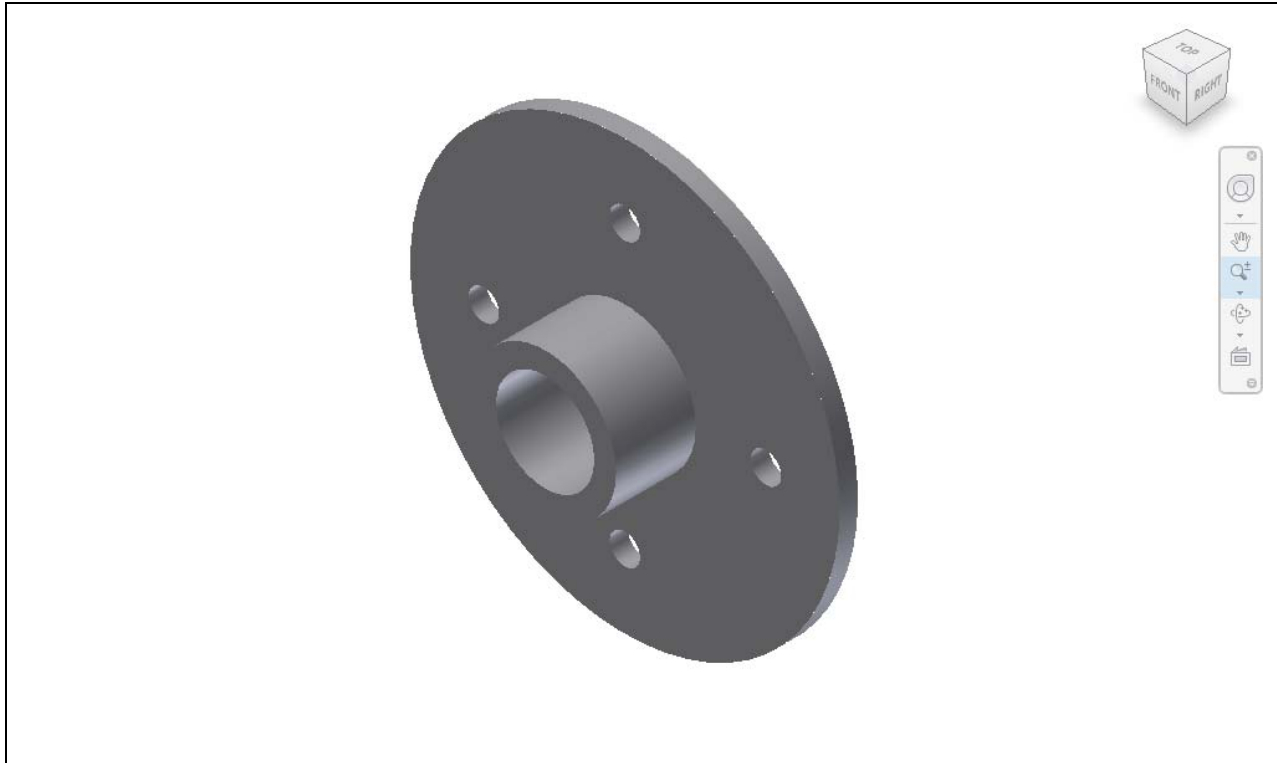


Figure 3.26 – The Solid Part

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



**Figure 3.27 – Finished Solid**

**\* World Class CAD Challenge 61-05 \* - Close this drawing file. Create a New file and draw the circle, the two concentric circle and the four holes on the bolt circle. Extrude the part. Complete the task in less than 7 minutes. Continue this drill four times, each time completing the drawing under 7 minutes to maintain your World Class ranking.**

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