

Chapter

3

Solid Part Two

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

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.

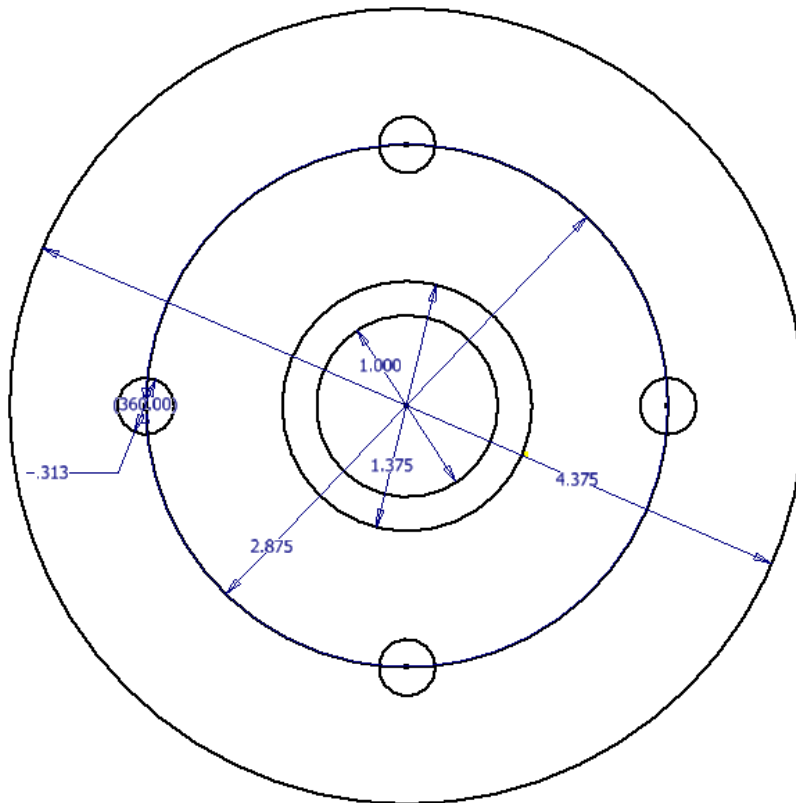


Figure 3.1 – Problem Two Sketch

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 SolidWorks 2012 application, we will select New from the menu.

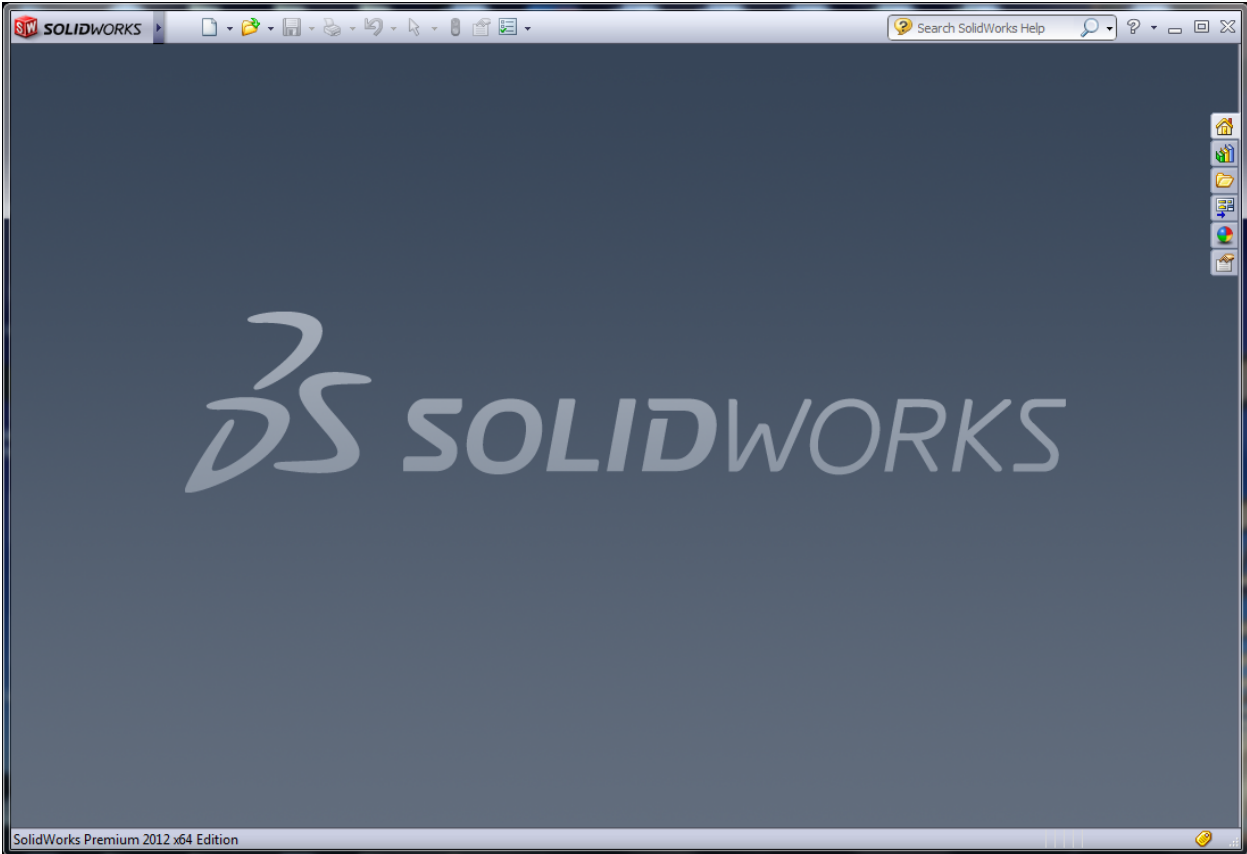


Figure 3.2 – SolidWorks 2012

The New SolidWorks Document window will appear and there are three icons on this dialogue box. They are Part, Assembly and Drawing. For this drawing, we will select the Part icon. We will press the OK button to continue.

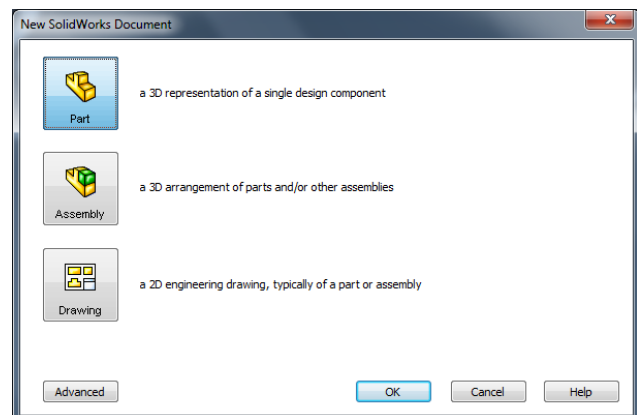


Figure 3.3 –Starting the drawing using the New SolidWorks Document window

When we launch the new drawing, we will see the Sketch ribbon at the top of the SolidWorks

window. We will draw our first part on the top plane. In the drawing properties pane, we select Top Plane and the display appears as shown.

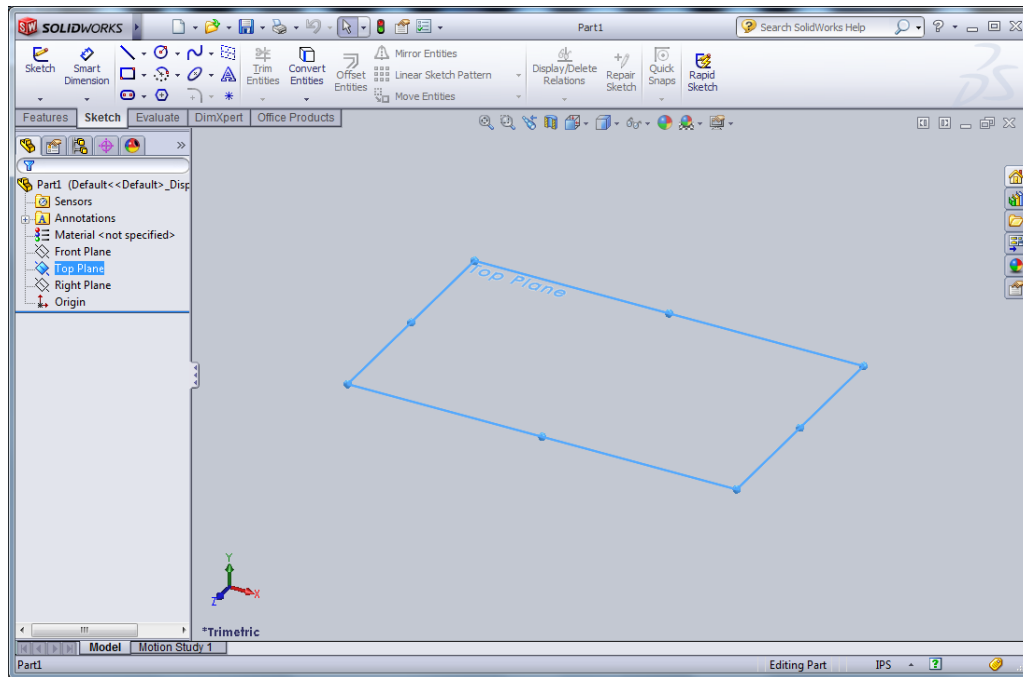


Figure 3.4 – Starting the drawing

Drawing a Circle

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



We will learn to place a circle on the drawing. We pick the Circle on the Sketch ribbon. Then we select the 0,0 coordinate on the drawing by choosing the corner of the red graphic representing the x and y axis. After picking the center point, we draw a circle.

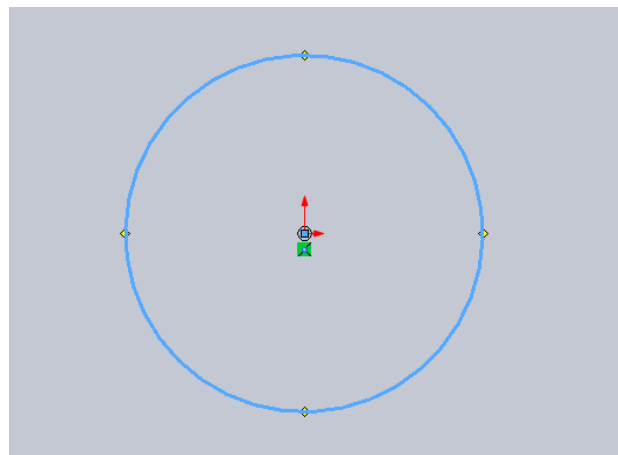


Figure 3.5 – Draw a Circle at 0,0

We see the circle options on the pane on the left side of the SolidWorks window. We enter 2.1875 for the radius of the circle. We press the green checkbox icon on the top of the pane and the circle is inserted.

Right click on the circle and we can select the Smart Dimension icon which is the second from the right on the menu. We choose Smart Dimension and then the perimeter of the circle and we can place the dimension on the drawing as we see in the figure.

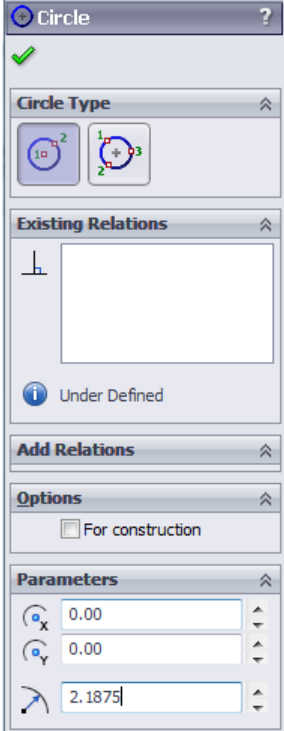


Figure 3.6 – Enter Radius

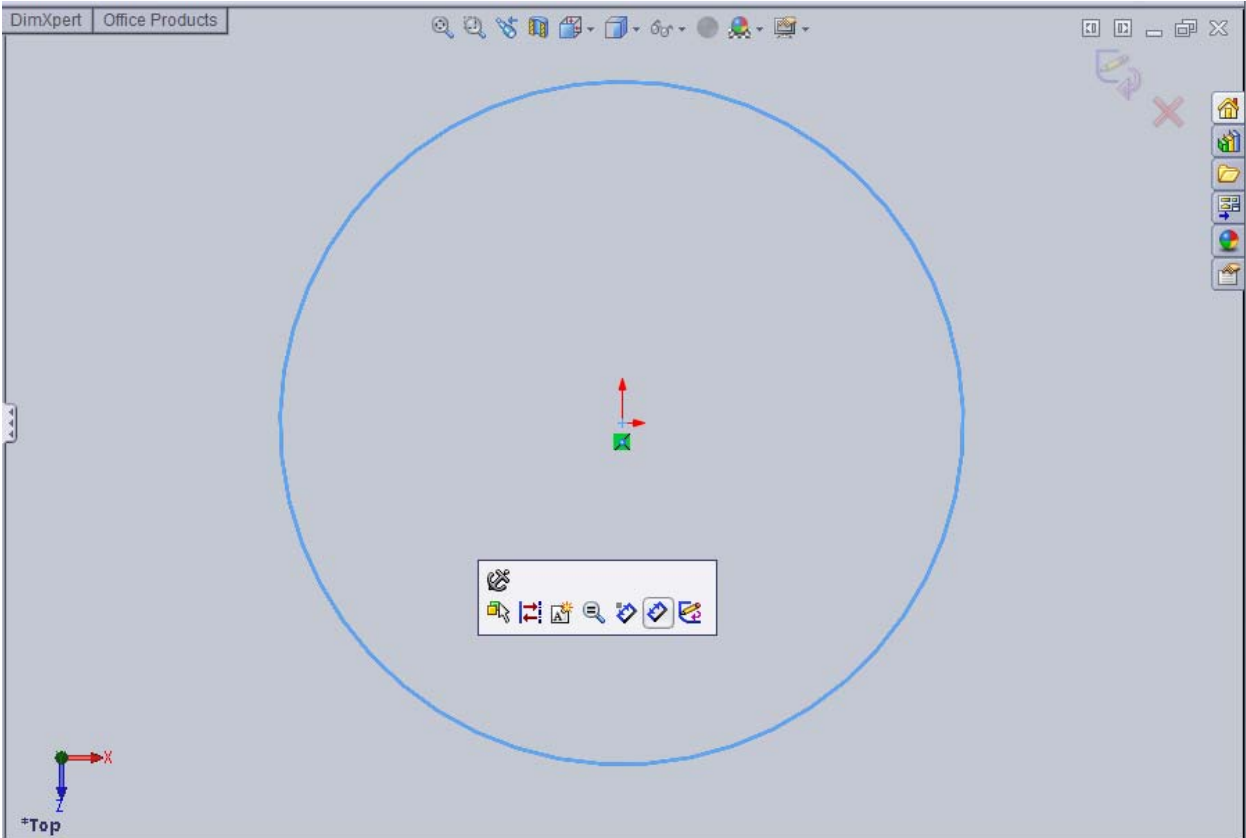


Figure 3.7 – Circle with Dimension Showing

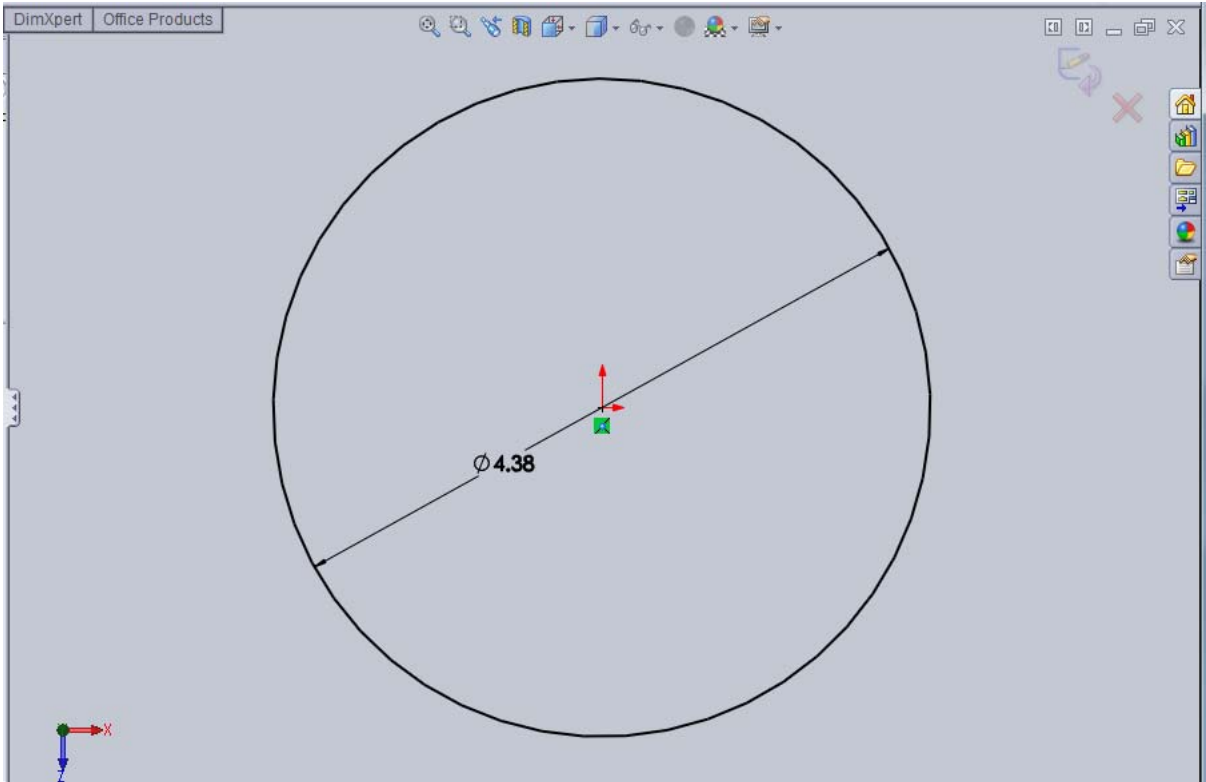


Figure 3.8 – Circle with Dimension Showing

Drawing a Concentric Circle

Next we will learn to place a concentric circle on the drawing. We pick the Circle on the Sketch ribbon.



We then point to the center of the 4.38 circle and an orange dot will appear that represents the exact middle of the circle. We click on the center point and the circle will appear. We draw any circle and in the left pane we will type 0.6875 for the radius. We press the green checkbox icon on the top of the pane and the circle is inserted.

Right click on the circle and we choose the Smart Dimension icon and then the perimeter of the circle and we can place the dimension on the drawing as we see in the figure.

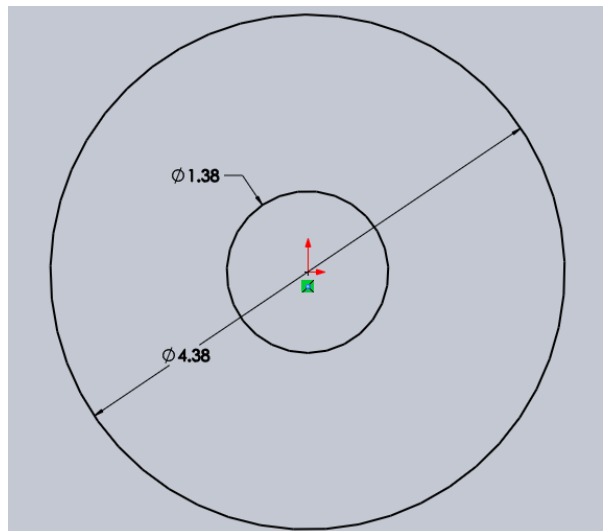


Figure 3.9 – The 1.38 Diameter Circle

Again we point to the center of the 4.38 circle and an orange dot will appear that represents the exact center of the circle. We click on the center point and the circle will appear. We draw any circle and in the left pane we will type 0.5 for the radius. We press the green checkbox icon on the top of the pane and the circle is inserted.

Right click on the circle and we choose the Smart Dimension icon and then the perimeter of the circle and we can place the dimension on the drawing as we see in the figure.

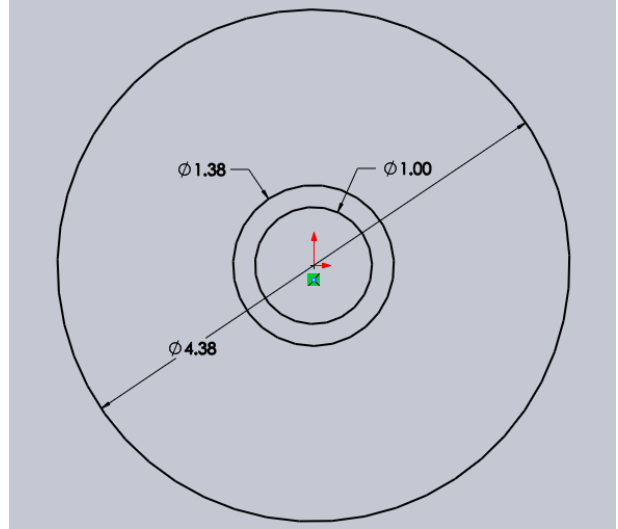


Figure 3.10 – Concentric Hub

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 annotating the Construction check box on the left side of the SolidWorks display in the Circle pane.



We start by selecting Circle on the Sketch ribbon. We point to the center of the 4.38 circle and an orange dot will appear that represents the exact center of the circle. We click on the center point and the circle will appear. We draw any circle and in the left pane we will type 1.4375 for the radius. Again we mark the construction checkbox and we press the green checkbox icon on the top of the pane and the circle is inserted.

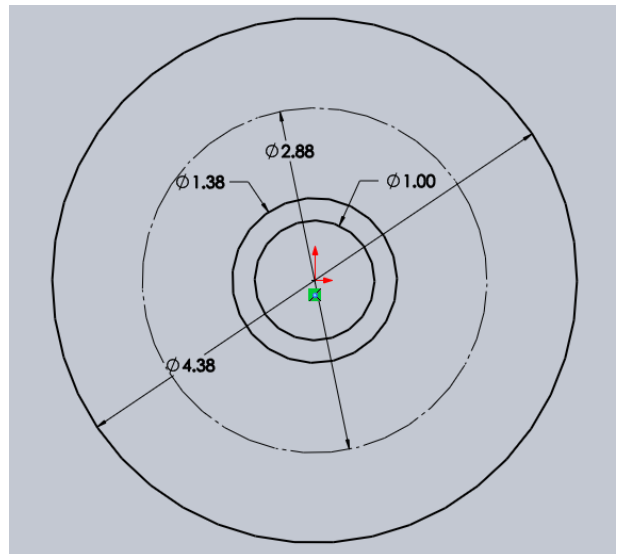


Figure 3.11 – A Construction Line

Right click on the circle and we choose the Smart Dimension icon and then the perimeter of the circle and we can place the dimension on the drawing as we see in the figure.

Add a Bolt Hole

The last set of circles begins with making a single 0.3125 diameter bolt hole. We pick the Circle on the Sketch ribbon.



We then point to the left quadrant of the construction bolt circle. When we place the cursor on the west quadrant we will see an orange diamond. We click on that point and we draw a small circle. In the left pane, we will type 0.15625 for the radius. We press the green checkbox icon on the top of the pane and the circle is inserted.

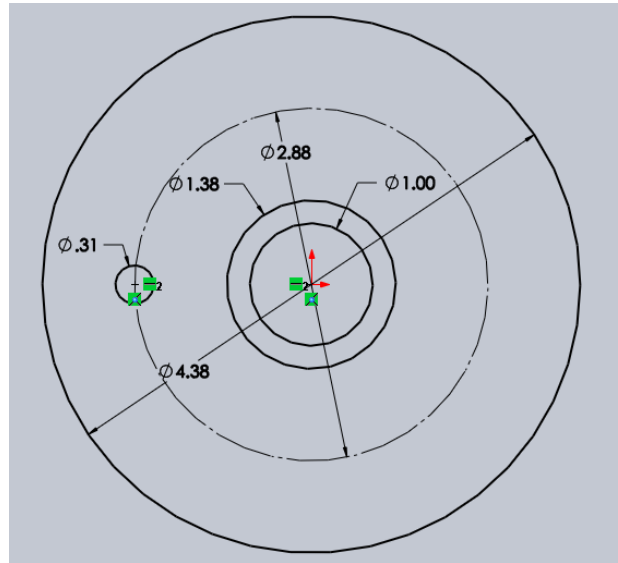
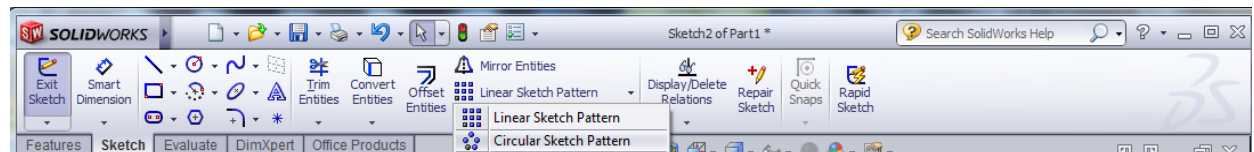
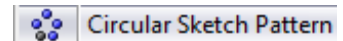


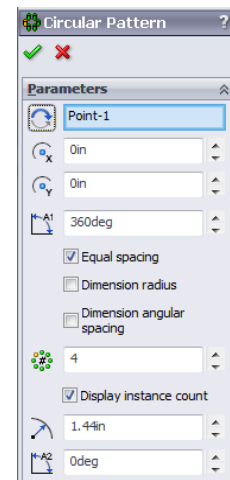
Figure 3.12 – The 0.31 Diameter Hole

Array the Circle

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



We select the 0.31 diameter hole and we select Circular Sketch Pattern on the Sketch Ribbon. We will see the Circular Pattern window appear on the left side of the graphical display. The default is equal spacing on a 360 degree circle with four holes. This happens to be the exact specification from our sketch. We can see representation of our circular array in figure.



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

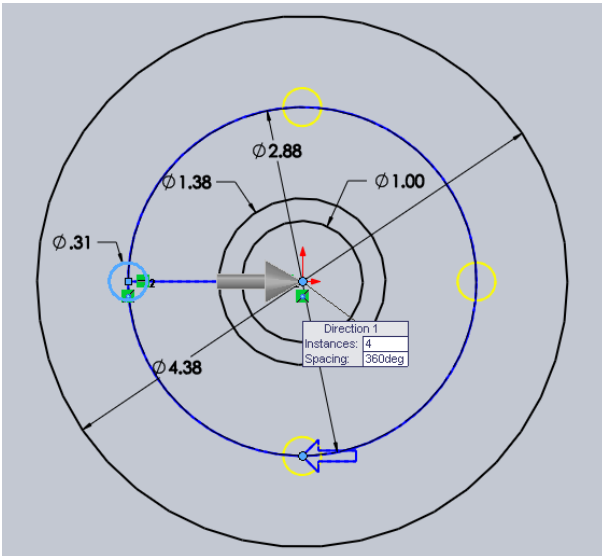


Figure 3.13 – Circular Pattern Choice

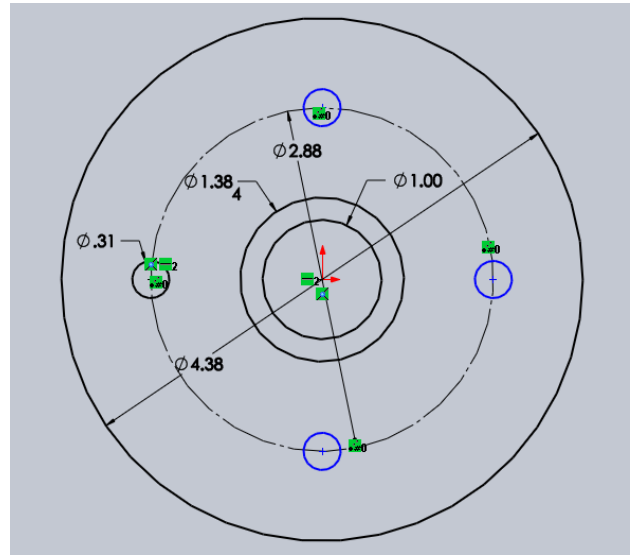


Figure 3.14 – Finished Sketch

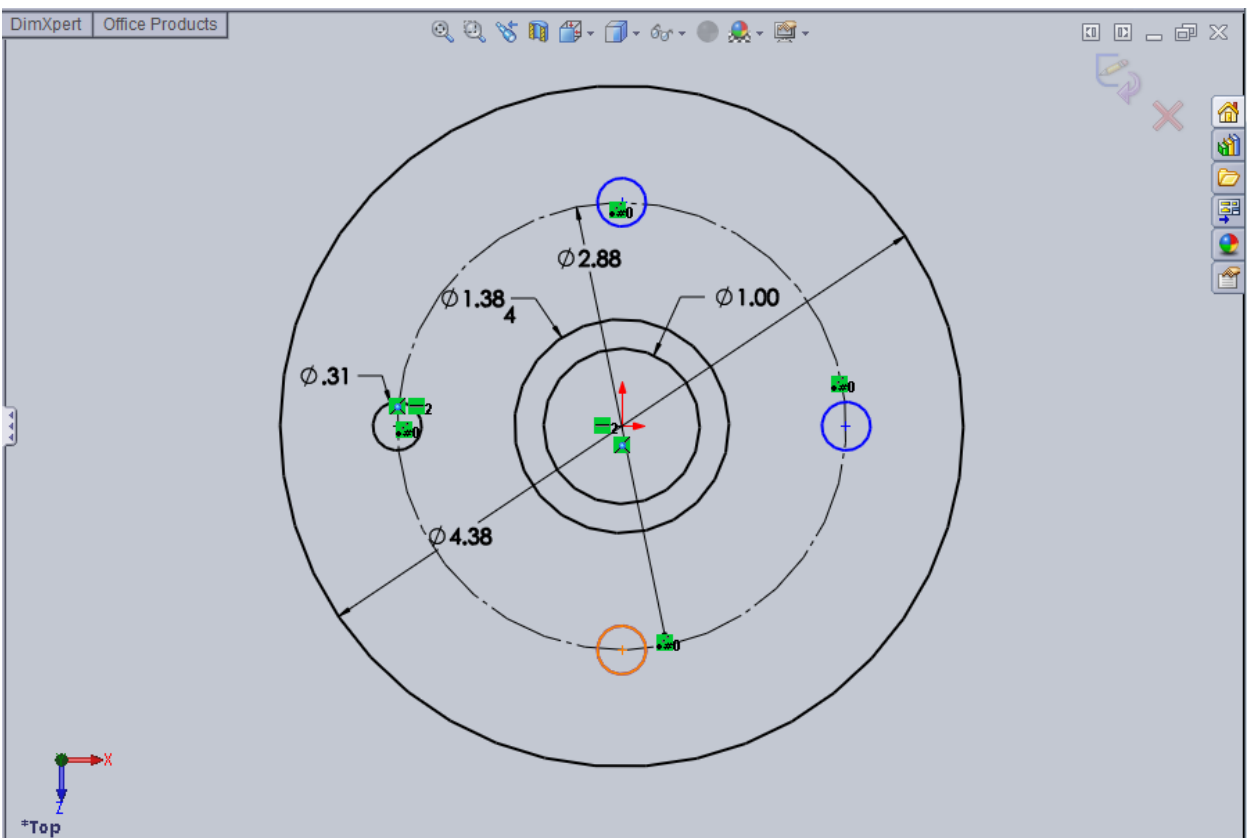


Figure 3.15 – The Finished Part

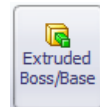
Save the drawing and we will now make a model off the sketch. To improve our sketching abilities, we should practice these drills until we know each function thoroughly.

* 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 to obtain your world class ranking.

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 Features ribbon. The Extrude pane will appear on the left side of the graphical display.



In the Extrude pane, we see the from the Sketch plane and the direction will be blind. We will change the 0.1 to 0.1875 for the extrusion thickness.

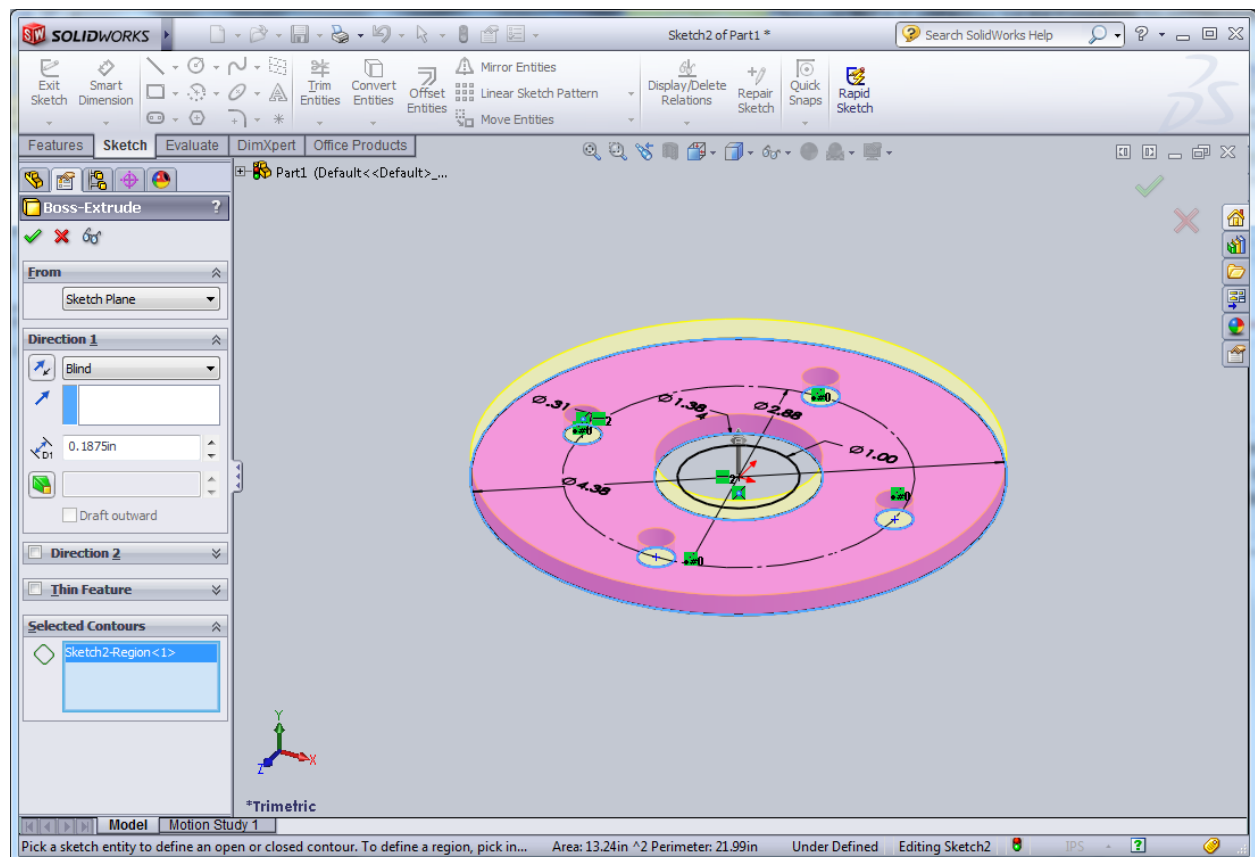


Figure 3.16 – The Extrude Window

We click on the green check mark and the solid becomes permanent.

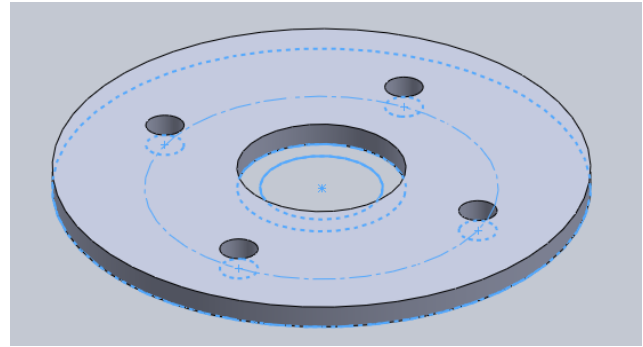


Figure 3.17 – Finished Disk

Sketching on the 3D Model

To obtain the next part of the solid, we will draw two circles. We choose the Top Plane on the left pane and we select Sketch on the Sketch ribbon. We draw two circles on the center of the disk. The first one has a 0.6875 radius and the second circle has a 0.5 radius.

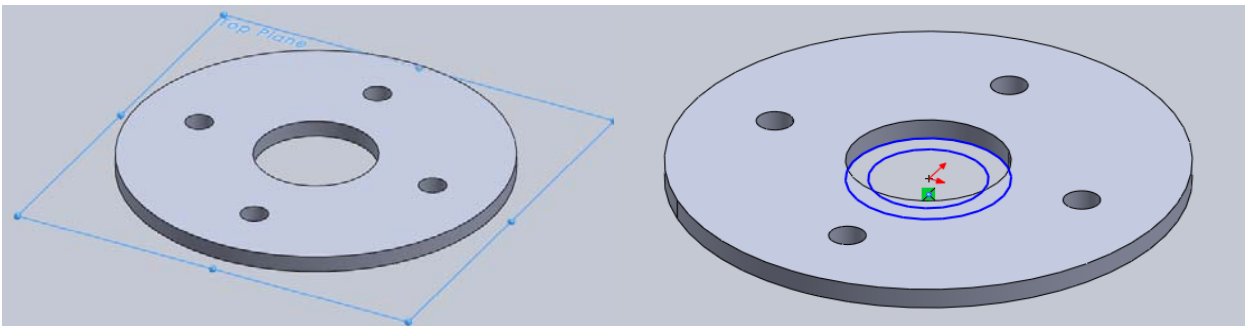


Figure 3.18 – Choose the Plane

Figure 3.19 – Draw Two Circles

We can go ahead and pick the Extrude icon on the Features ribbon. The Extrude pane will appear on the left side of the display.

Then we will click in between the two circles and it will become highlighted. Click on the area again and the 2D shape will become a solid with a hole where the circle was located. We input 0.8125 for the distance and we press the green checkmark to make it permanent.

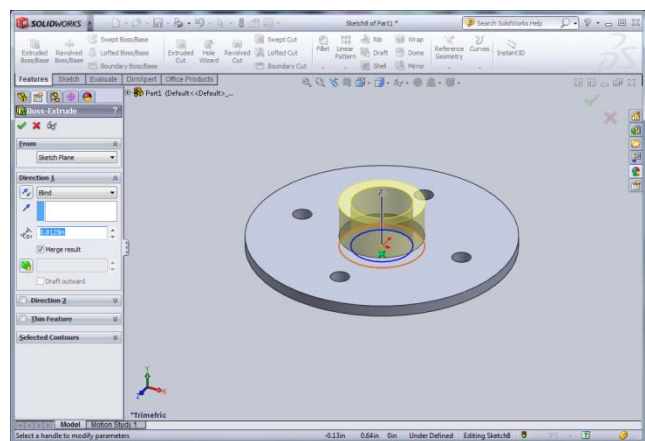


Figure 3.20 – Extrude the Part

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

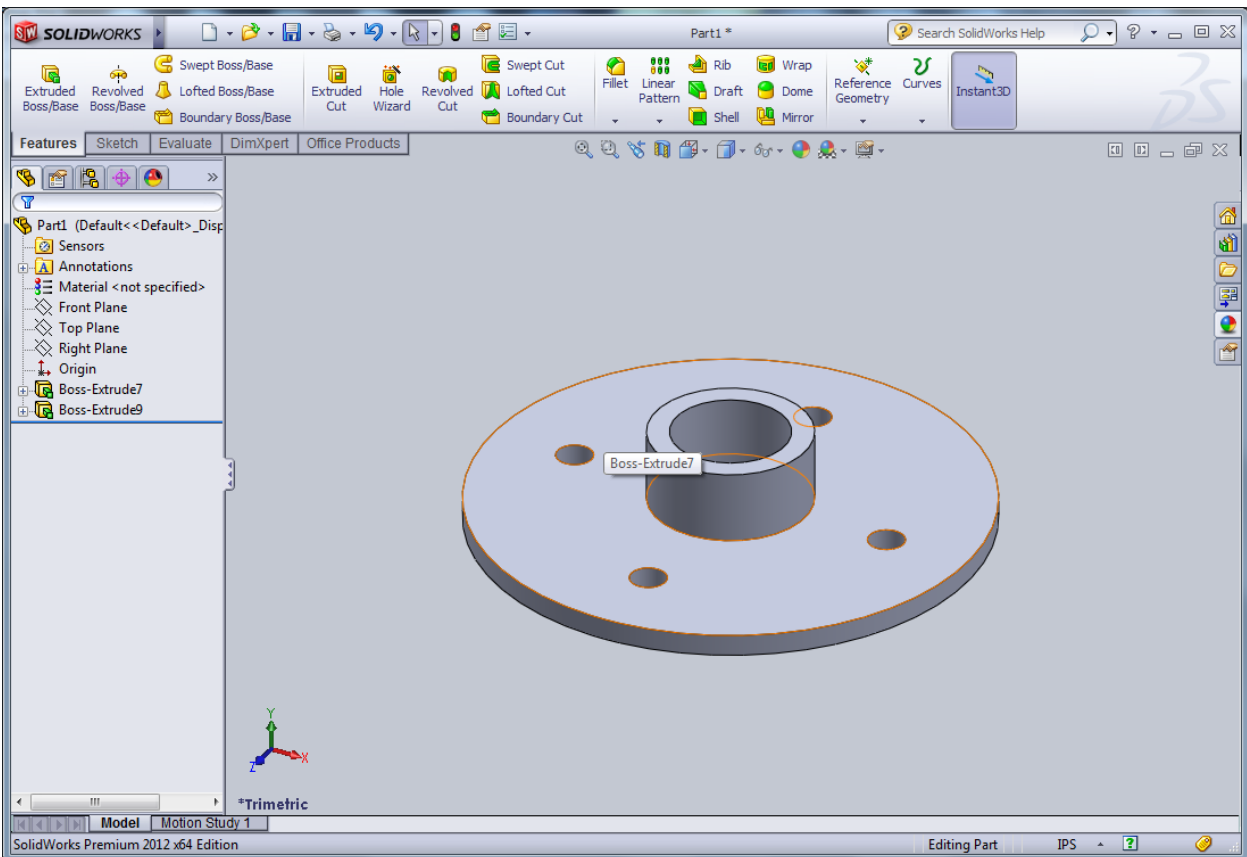


Figure 3.21 – Finished Solid

*** 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.**

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