

Chapter

2

Solid Part One

This chapter will cover the following to World Class standards:

- **Sketch of Solid Problem One**
- **Drawing a Rectangle**
- **Drawing a Line**
- **Copying the Sketch**
- **Dimension and Move a Circle**
- **Finish 2D Sketch of Solid Part One**
- **Extrude the Sketch**

Sketch of Solid Part One

When receiving a sketch from a professional, the CAD technician will try to identify what fundamental features are hidden in the composite solid. The basic building shapes in the three dimensional design are rectangular or cubic box, spherical shapes, conical shapes, cylindrical shapes and the triangular wedge. In your first 3D solid, you might already observe that there is a rectangular box that measures 7 by 2.957. There are two tabs off the bottom of the rectangle with holes in the center. There are four holes in a rectangular pattern on the rectangle. Whatever procedure you use to complete the exercise, most beginners find that the Rectangle, Line and Circle tool on the Sketch menu provides one of the easiest methods to draw the component.

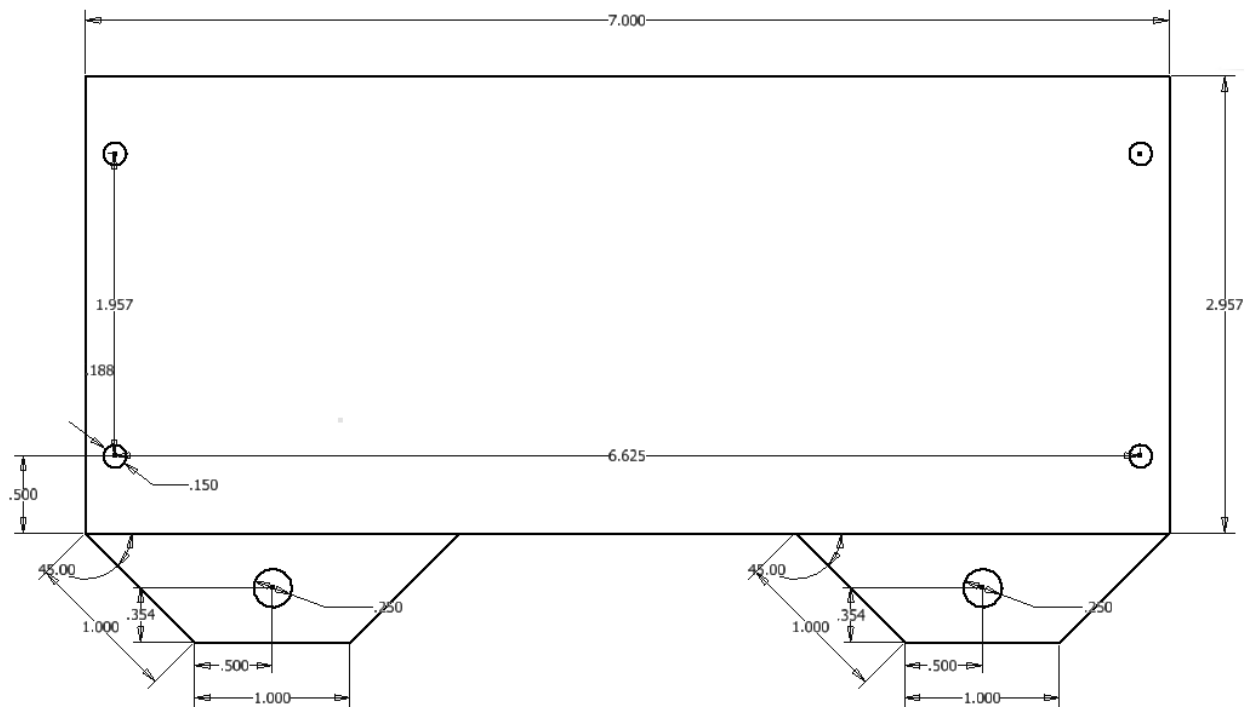


Figure 2.1 – Problem One sketch

As you have already seen from your other 2d drafting training, SolidWorks 2012 has a different interface than your previous CAD experience. Therefore, these lessons will expose us techniques that will improve our sketching skills and therefore take us to 3D modeling even faster.

Some features we will enjoy using in SolidWorks 2012 are changing a dimension on an entity and then the line or circle will move along with the altered measurement. We will enjoy drawing components of the part such as this front view shown in figure 2.1 and when we start to model it in the next chapter, we will quickly have a 3D part. So let us start to use the SolidWorks 2012 program and create 3D parts and assemblies.

Starting a 3D Part Drawing Sketch

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

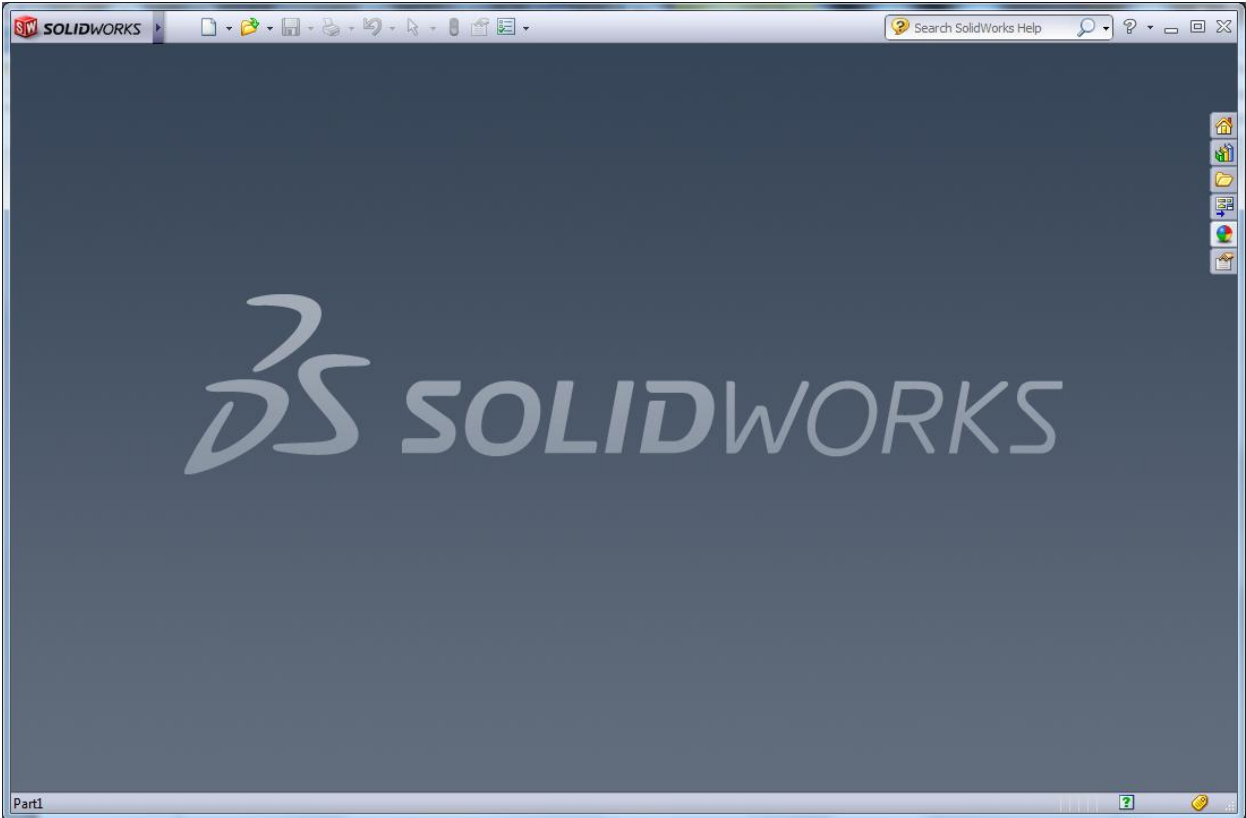


Figure 2.2 – SolidWorks 2012

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.

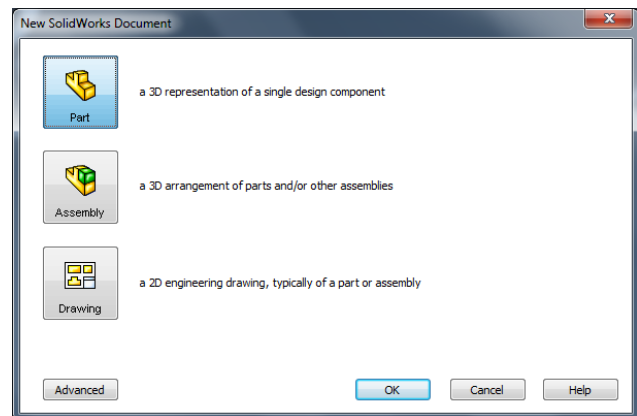


Figure 2.3 – Starting the drawing using the New SolidWorks Document window

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.

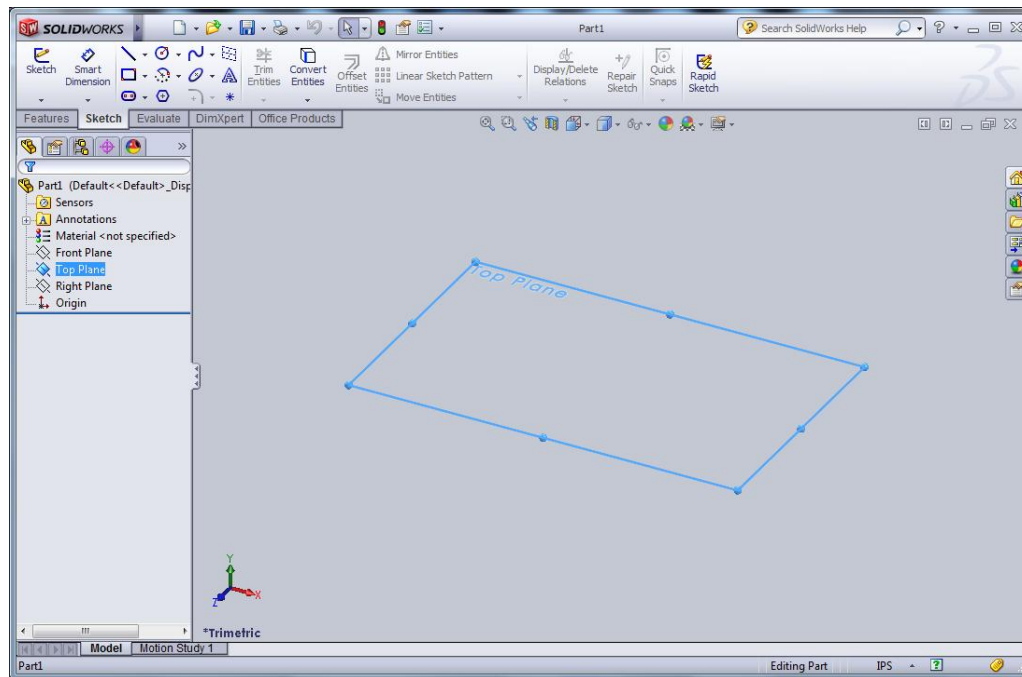


Figure 2.4 – Starting the drawing

Drawing a Rectangle

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

To draw a rectangle, we left click on the Corner Rectangle icon on the Sketch ribbon and we can see five rectangle types in the pane on the left side of the SolidWorks program. We continue with the default Rectangle where we will single click on the upper left portion of the graphical display and then pull the rectangle to the upper right section of the display.

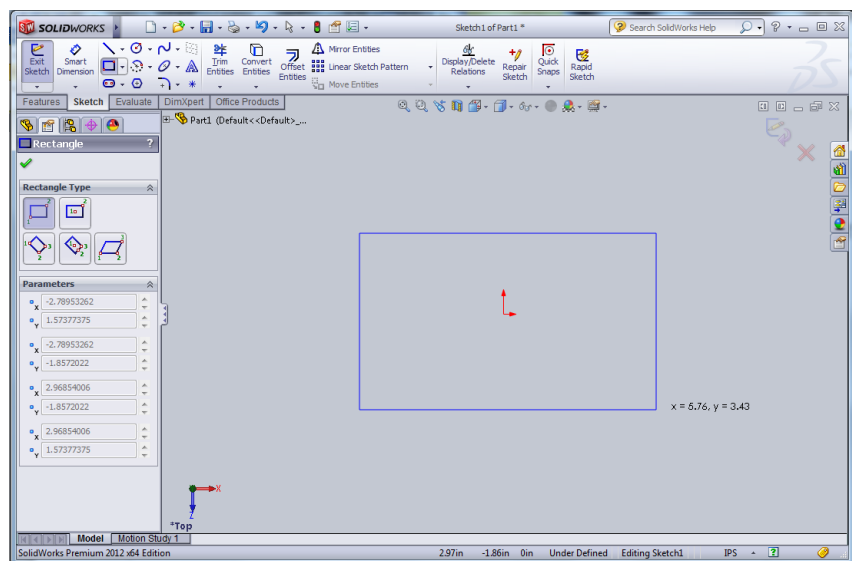


Figure 2.5 – Graphical Display Menu

This any sized rectangle on the graphical display has two dimensions. When we double click on the lower horizontal line, the dimension appears. We can pull it down below the rectangle. In the Modify window, we can type 7. We press the green checkbox icon and the rectangle will change to that measurement.

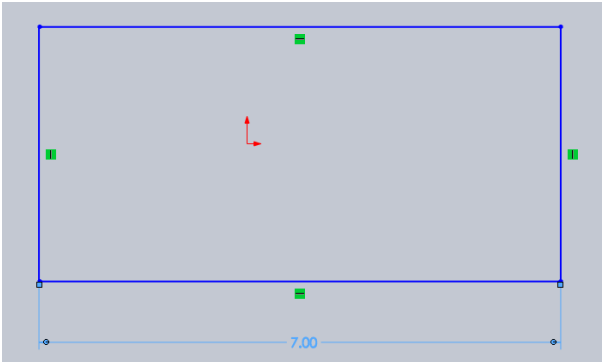


Figure 2.6 – Horizontal Dimension

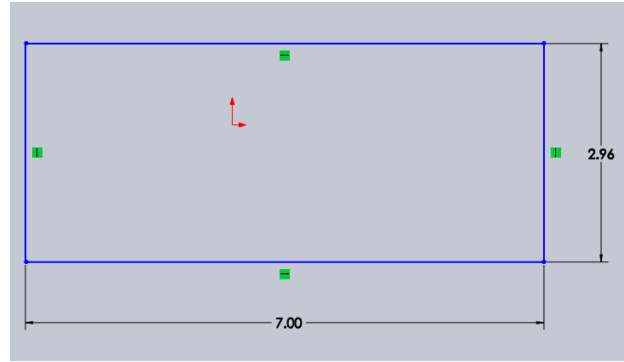


Figure 2.7 – Vertical Dimension

When we right click on the right vertical line, the menu appears and we can choose more dimension and vertical dimensions. We can click on the right vertical line and when the dimension shows up, we pull it over to the right of the rectangle. In the Modify window, we can type 2.957. We press the green checkbox icon and the rectangle will change to that measurement.

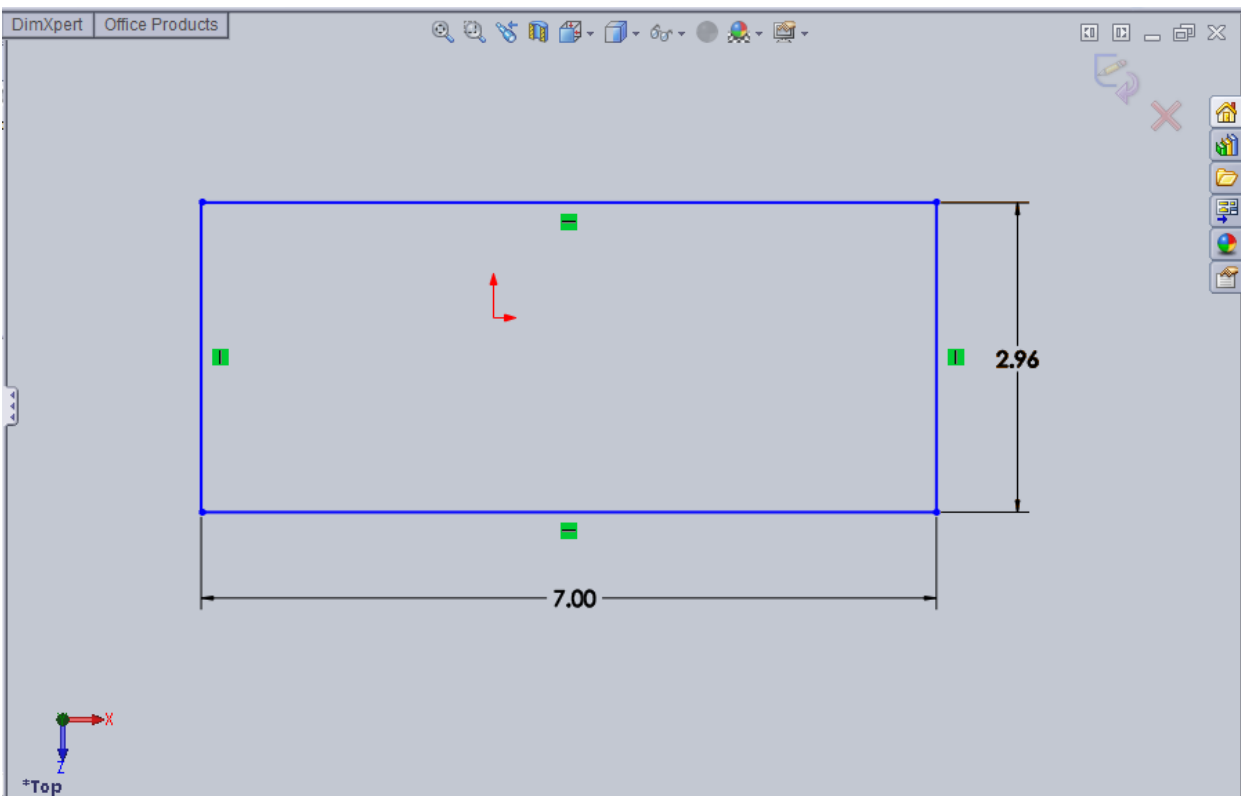


Figure 2.8 – Rectangle with Two Dimensions Showing

Drawing your First Line

The next entity we will learn to draw in SolidWorks is a line. We left click on the Line icon on the Sketch ribbon.



We see the line options on the pane on the left side of the SolidWorks window. We pick the Angle option and annotate the Add dimensions checkbox. This is a polar dimension, which has a length and an angle, so we input 1.00 for the length and -45 degrees for the angle. We then choose the lower left corner of the 7 by 2.957 rectangle. We can observe the line with the two dimensions as we pick the starting point. We press the green checkbox icon on the top of the pane and the line is inserted.

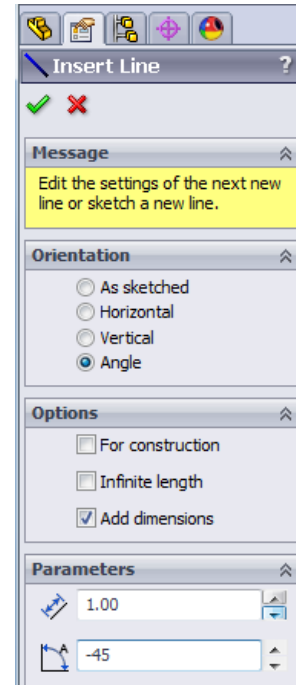


Figure 2.9 – Drawing a Line

We can see the line. To draw a second line, again we choose the Line icon on the Sketch ribbon. We pick the Horizontal option and annotate the Add dimensions checkbox. We type 1.0 for the length of the line.

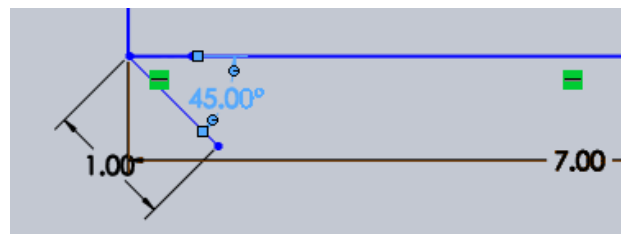


Figure 2.10 – Length and Angle

We then choose the right end of the first line. We press the green checkbox icon on the top of the pane and the line is inserted. We add the horizontal dimension to the line.

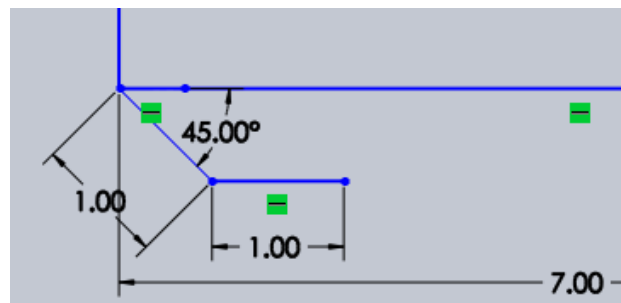


Figure 2.11 – Second Line Segment

For the last line segment, we select Line icon on the Sketch ribbon and start the line on the right endpoint of the second line. When we pull the third segment to the bottom of the rectangle, we will automatically get a 45-degree line. We then draw a fourth segment to the starting point of the first line.

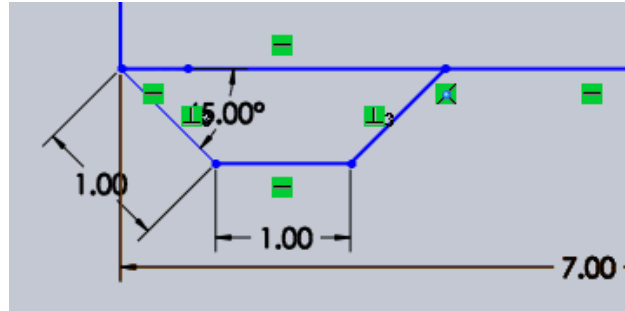


Figure 2.12 – Third Line Segment

Drawing a Circle

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



We then point and not click on the midpoint of the 1-inch line. A dotted line appears as we move the mouse upward.

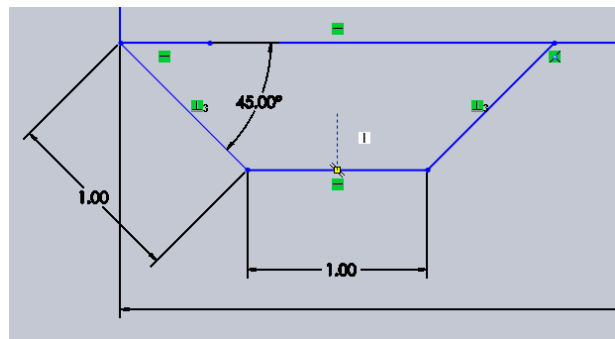


Figure 2.13 – Point to the Midpoint

We then move the mouse to the midpoint of the 45-degree line to the left and we will see two dotted lines, one horizontal and one vertical. We then can pick the intersection and the circle will be in the middle of the tab.

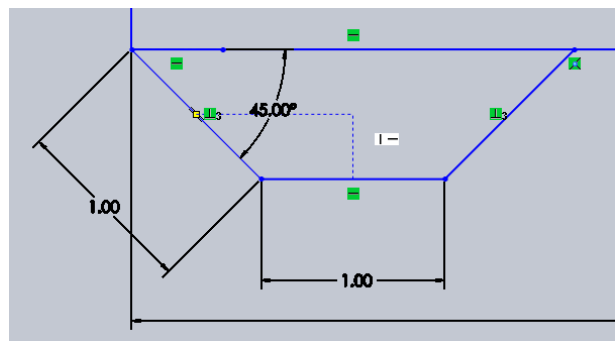


Figure 2.14 – Point to the Vertical Midpoint

Then we can input the radius of the circle as 0.125 in the pane to the left. We press the green checkbox icon on the top of the pane and the circle is finished.

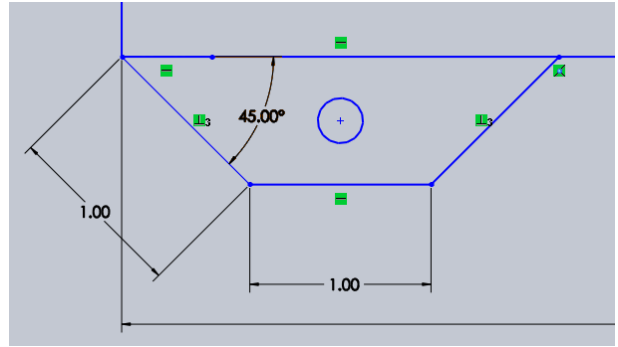


Figure 2.15 – Input the Diameter

Adding Dimensions

Sometimes when we add an entity, we will not see all the dimensions, so we will load them. We can choose Smart Dimension on the Sketch ribbon.



We will pick the circle and the left corner of the tab and the 0.50 dimension will appear. Click on the green check on the Modify window to make it permanent.

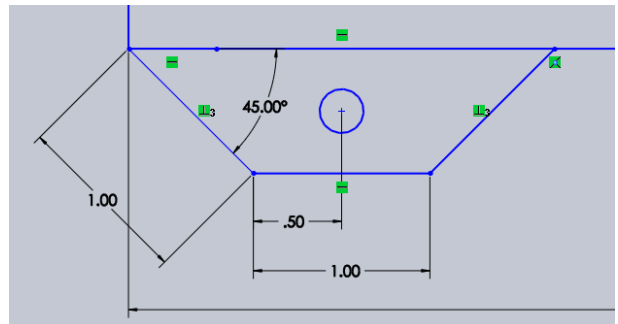


Figure 2.16 – Horizontal Dimension

We will pick the circle and the left corner of the tab and this time we will pull the dimension to the left so the 0.354 dimension will appear. Click on the green check on the Modify window to make it permanent.

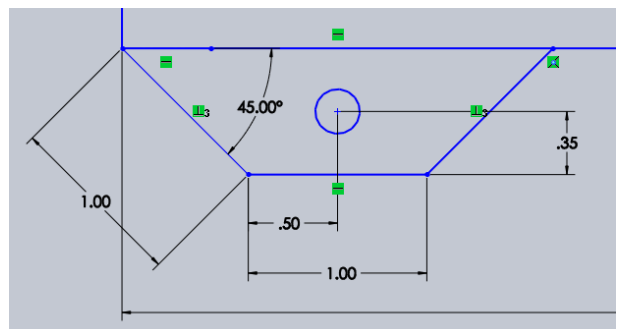


Figure 2.17 – Vertical Dimension

Copying in a SolidWorks Sketch

After we have drawn a single or series of entities, we can improve our efficiency by copying them to repeat the shape.

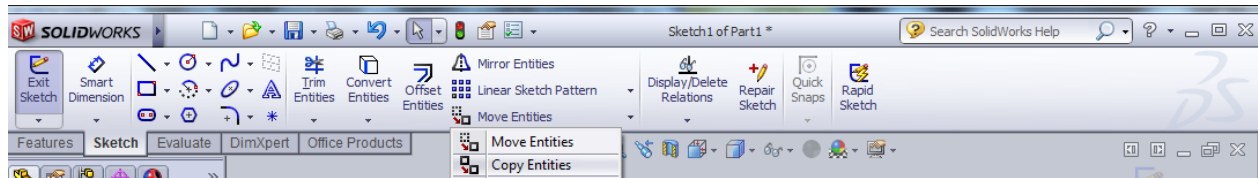
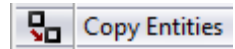


Figure 2.18 – Copy on the Sketch Ribbon

We select a point on the graphical display to the left of the rectangle being sure not to click on a line or dimension. We hold down the left mouse button a blue rectangle will cover the entities we want to choose. To accept the selection set, we right click the mouse.

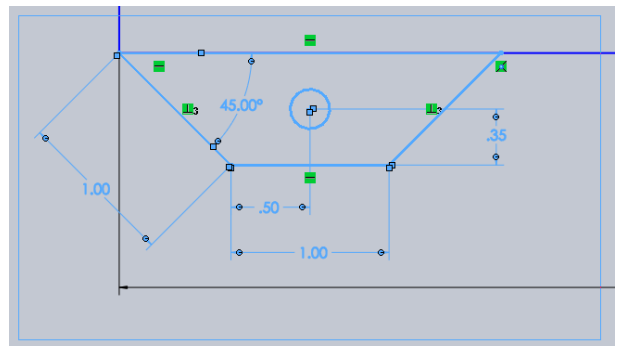


Figure 2.19 – Highlight the Entities

We then pick intersection of the right side of the tab and the bottom of the rectangle. We then pick the right corner of the rectangle with the mouse to show where the new entities will be drawn.

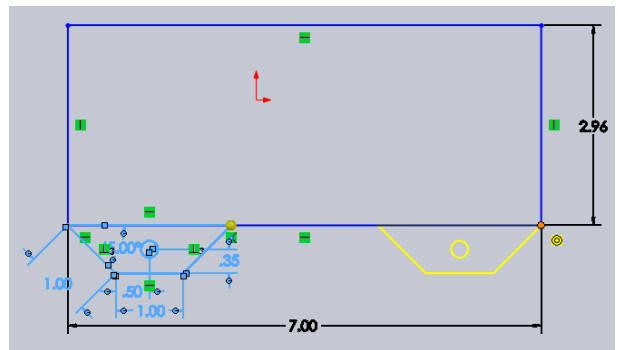


Figure 2.20 – Highlight the Entities

All five entities and dimensions are now copied and shown in figure 2.21.

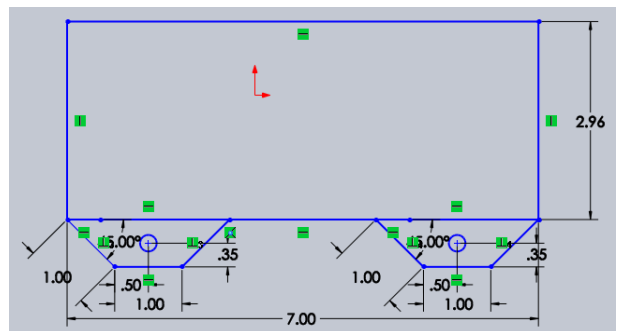


Figure 2.21 – Base Point of Displacement

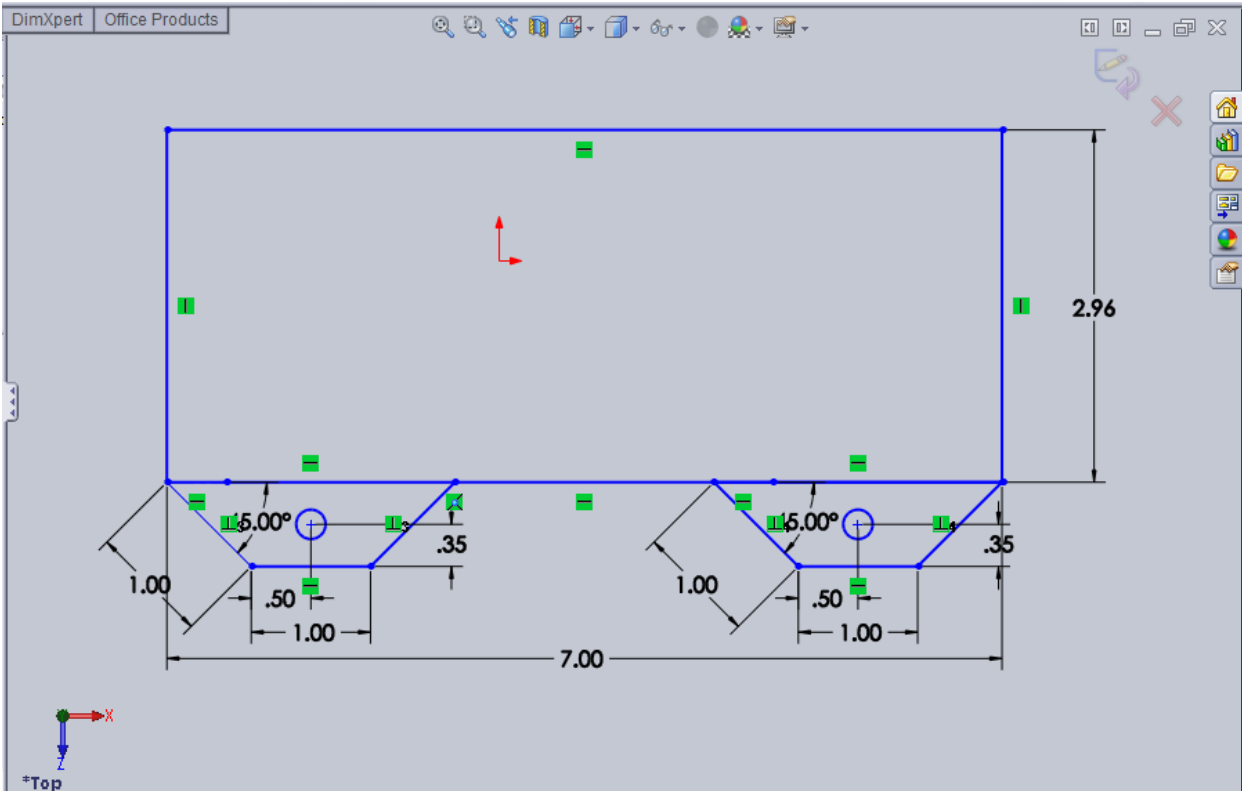


Figure 2.22 – Finished Copying

Drawing another Circle

Next, we will repeat drawing a circle on the drawing. We pick the Circle on the Sketch ribbon.



When we pick the Circle, we then point and click anywhere inside the lower left hand corner of the rectangle. We draw a 0.075 radius circle. We press the green checkbox icon on the top of the pane and the circle is finished.

We can add a smart dimension by choosing that icon on the Sketch ribbon and pick the perimeter of the circle. The 0.15 diameter dimension will appear. Click on the green check on the Modify window to make it permanent.

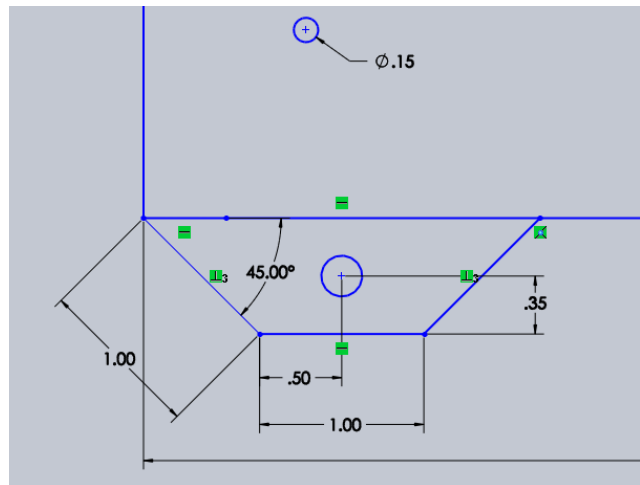


Figure 2.23 – Draw a 0.15 Circle

Dimension and Move the Circle

After we have drawn a single or series of entities, we want to move them to a precise position on the drawing. We can do that by applying the dimensions and changing the measurements and the entity will move its position.

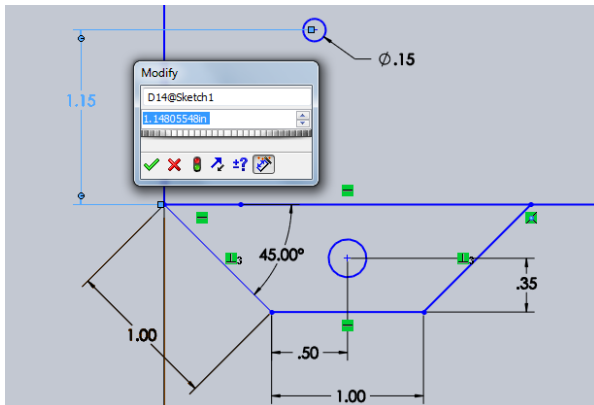


Figure 2.24 – Add the Vertical Dimension

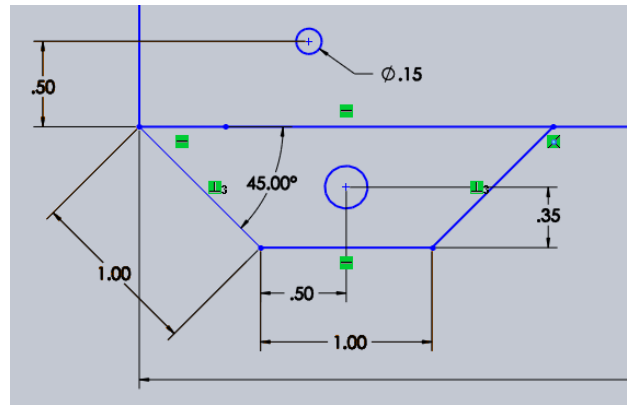


Figure 2.25 – Change the Location

We place the vertical dimension from the lower left hand corner of the rectangle and to the center of the 0.15 diameter circle. Our dimension shows 1.15 but your drawing can be entirely different. We type 0.5 in the Modify window and we press the green check mark and both the dimension and the circle's position changes.

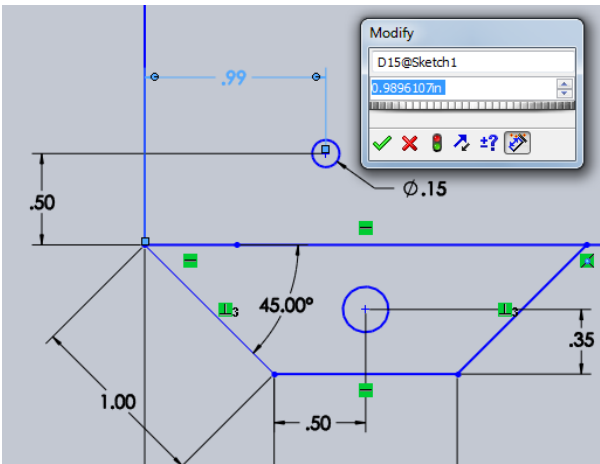


Figure 2.26 – Add the Horizontal Dimension

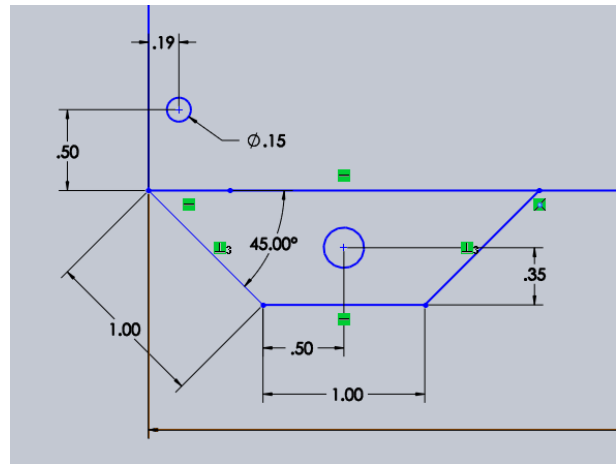
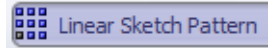


Figure 2.27 – Change the Location

We place the horizontal dimension from the lower left hand corner of the rectangle and to the center of the 0.15 diameter circle. Our dimension shows 0.99 and again your drawing can be dissimilar. We type 0.1875 in the Modify window and we press the green check mark and both the dimension and the circle's position changes.

Array the Circle

We can array an entity using a rectangular or circular pattern. The first array we will do is the rectangular configuration.



We made a window around the 0.15 diameter circle and then we select Linear Sketch Pattern on the Sketch Ribbon and we will see the Linear Pattern Pane appear on the left side of the graphical display. We pick the Direction 1 which is the X spacing. We type 6.625 for the dimension and 2 for the number of columns. We then pick the Direction 2, which is the Y spacing. We type 1.957 for the dimension and 2 for the number of rows. A rectangle with yellow circles will show the intended array.

We press the green checkbox icon on the top of the pane and three circles are added.

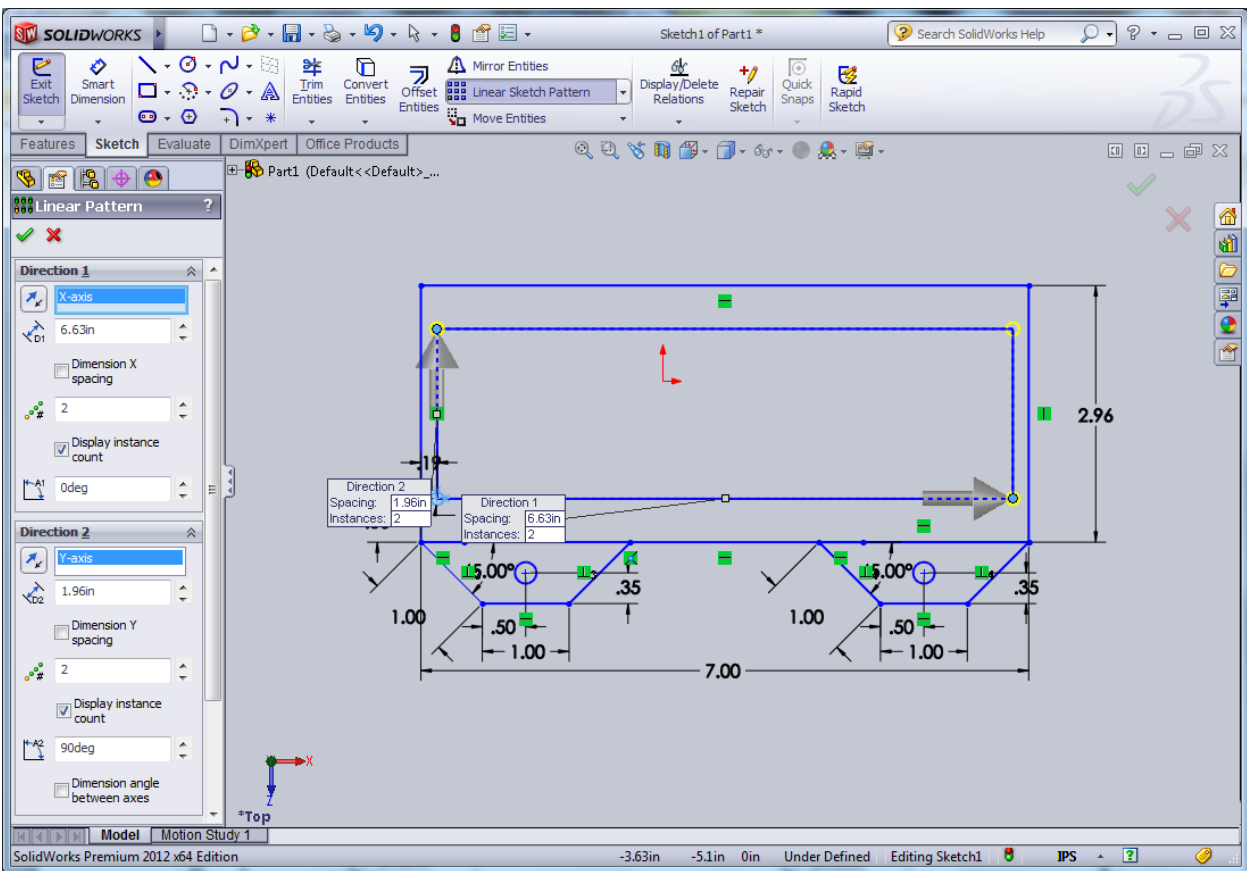


Figure 2.28 – Array the Circle

Then, we will click on the left tab and it will become highlighted in red. Click on the red area again and the 2D polygon will become a solid with a hole where the circle was located.

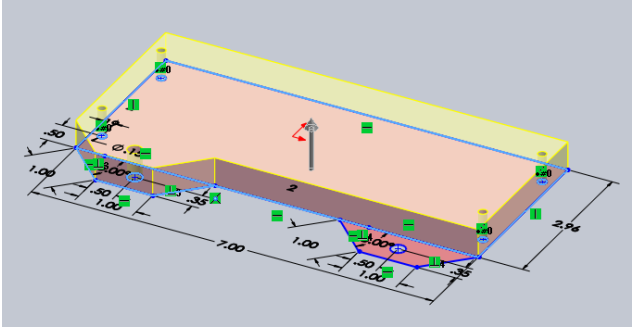


Figure 2.33 – Highlight Plate

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

The last step is to press the green check mark icon on the Extrude pane and the solid part becomes permanent.

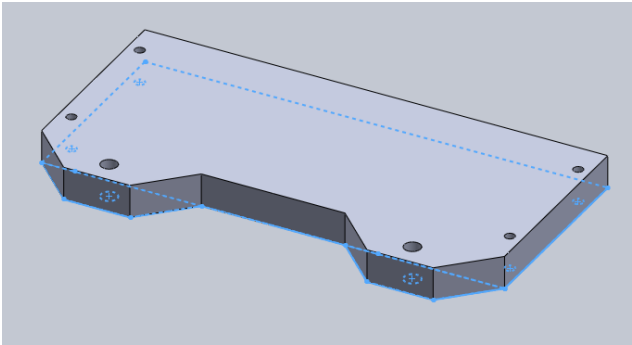


Figure 2.34 – The Solid Part

Save the drawing as Part 1 and we will now have a solid part to be used on a project.

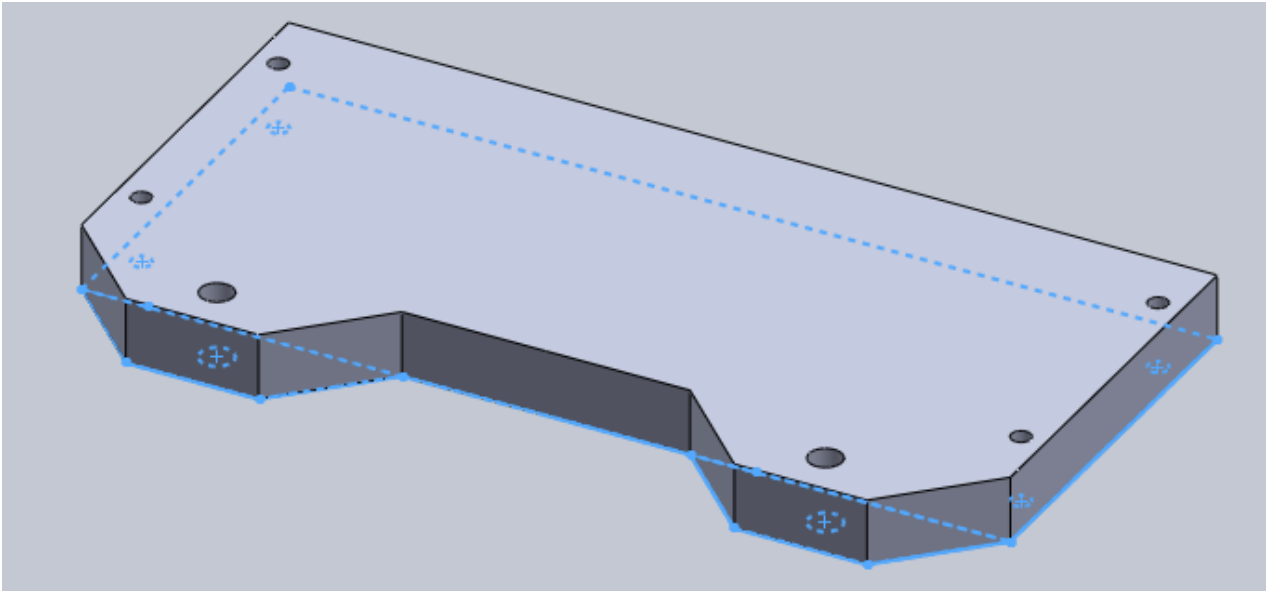


Figure 2.35 – Finished Solid

*** World Class CAD Challenge 61-03 * - Close this drawing file. Create a New file, draw the rectangle, the two tabs, and insert the circles. 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.**