

# Chapter 3

## Dimensioning the Rectangle Problem

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In this chapter, you will learn the following to World Class standards:

- **Creating New Layers in a Solid Edge Drawing**
- **Placing Centerlines on the Drawing**
- **Changing Entities from One Layer to Another**
- **Placing Center Marks on the Drawing**
- **Basic Use of the Dimension Panel**
- **Using the Distance Between Command**
- **Using the Distance Between Command for Aligned Dimensions Using Two Points**
- **Using the Smart Dimension Tool to Dimensions Radii and Diameters**
- **Using the Line Tool to Draw a Reference Line when Measuring Angles**
- **Using the Angle Between Tool**
- **Using the Edit Definition-Prefix Tool to Add Text Below a Dimension Using a Subfix**

## Creating New Layers in a Solid Edge Drawing

If you have used any graphical software in your career, you will already be familiar with the concepts of layers. If you are not familiar with the idea, think of layers as you would a sandwich: each item has its own flavor and color, but when you stack them all together you have a whole sandwich. The layer system in a CAD program allows you to view, revise or control information that is grouped together, like taking the mustard off a sandwich without removing the bread. In chapter 2, the Rectangular Problem was drawn on the default layer which is the layer chosen automatically whenever you start a new drawing file in Solid Edge. You can see the current layer listed in the Layer Window on the left of the screen below the ribbon (Figure 3.1). To create new layers, you only need to select the **New Layer** tool on the Layers Window (Figure 3.2).

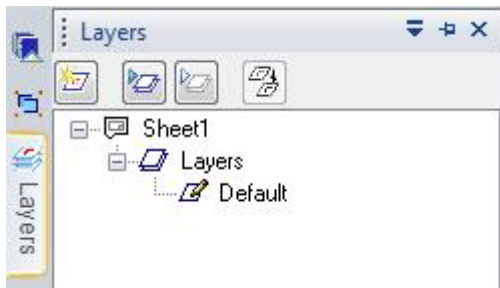


Figure 3.1 – Layers Window



Figure 3.2 – Create New Layer Tool

To dimension your part, you need to make three new layers: the **Dimension** layer for dimensions, the **Center** layer for centerlines and the **Text** layer for text and notes. To create the layers, select the **New Layer** tool and it will add a layer below the default layer. Immediately, the layer name is ready for editing, so type “**Dimension**” and press **Enter**. Repeat the process to add the **Center** and **Text** layers.

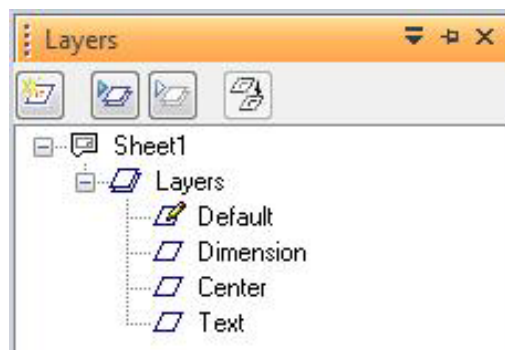


Figure 3.3 – Three Added Layers

## Placing Center Lines on the Drawing

Now you need to set up your centerlines. You placed the centerlines on the drawing right after drawing the 6x4 perimeter, so you only need to change layers for both of these lines. Most CAD technicians would only select these entities after selecting the command, but now you will learn to create your selection set at first. Pick the vertical and horizontal line, noticing that they become pink (Figure 3.4). In the Select Window, the drop-down arrow on the top will read “Visible”. You need to change the two centerlines into dashed lines by picking the “Center” setting (Figure 3.5). After that, select the **Line Color** tool and change the color to brown (Figure 3.6).

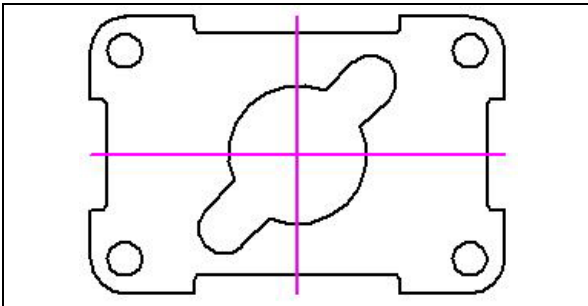


Figure 3.4 – Selecting the Centerlines

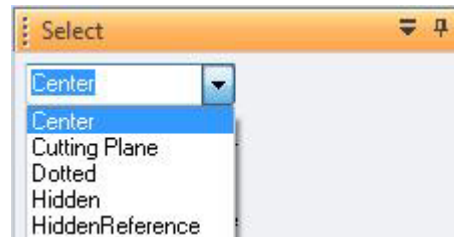


Figure 3.5 – Changing to “Center”



Figure 3.6 – Selecting Line Color

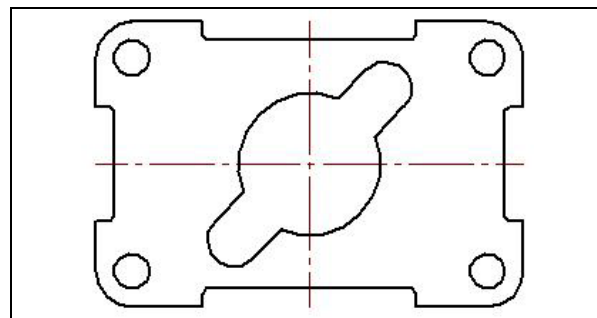


Figure 3.7 – Finished Centerlines

The next thing you need to do is to move the two centerlines to the **Center** layer. Do this by selecting the lines once more, and then select the **Move Elements** tool on the Layers Window (Figure 3.8). When the Move Elements Window appears, select the **Center** layer and click the “Okay” button at the bottom of the window (Figure 3.9). This will move your centerlines to the **Center** layer.



Figure 3.8 – Selecting to Move Elements

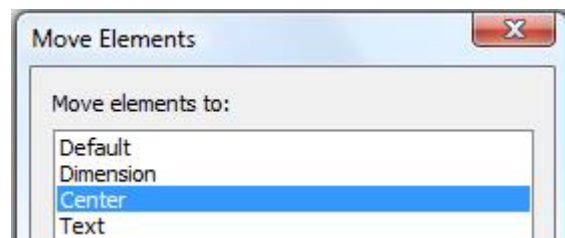


Figure 3.9 – Selecting to Move to Center

## Placing Center Marks on the Drawing

Dimensioning is the easiest task to be performed in CAD. By completing the difficult work in the last chapter, you are now rewarded with picking IntelliSketch points and laying out a visually pleasant set of dimensions. Technicians with any artistic talents will enjoy the ability to have freedom of layout in this section.



In the Layers Window, select the Center layer by double clicking on it with your left mouse button. A pencil should appear in the box next to the layer to signify that it is active (Figure 3.10). Next, you need to place center marks inside the four 0.5 circles and inside the 0.75 arcs on the inner detail. Select the **Center Mark** tool on the **Annotation** panel under the **Home** tab. The mouse arrow will turn into a pick box. Select the 0.5 circle on the lower left corner of the part (Figure 3.11). A center mark will appear inside of the circle. Follow a clockwise or counterclockwise pattern that will help you to get every circle on your drawing and improve your drawing efficiency. Lastly, select the two 0.75 arcs on the inner detail. You will have six center marks on the drawing when finished (Figure 3.12).

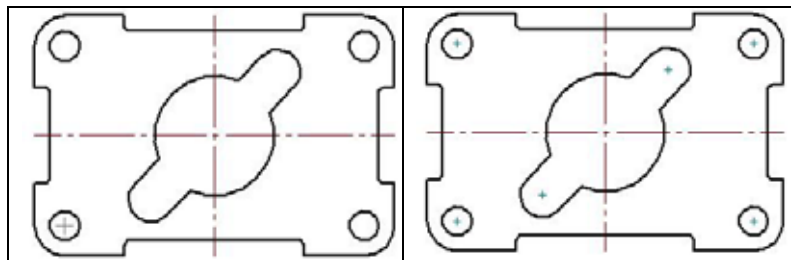
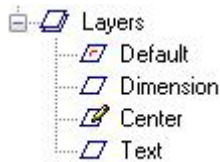


Figure 3.10

Figure 3.11

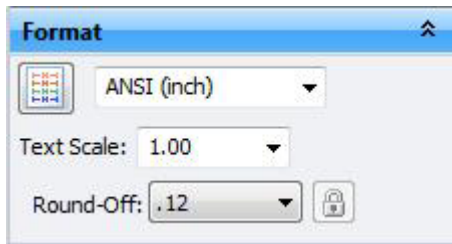
Figure 3.12

Did you see how easy the dimensioning tools work for you? As long as you draw your part accurately without fudging the numbers, the associated dimensions in the CAD software will do the rest of the work.

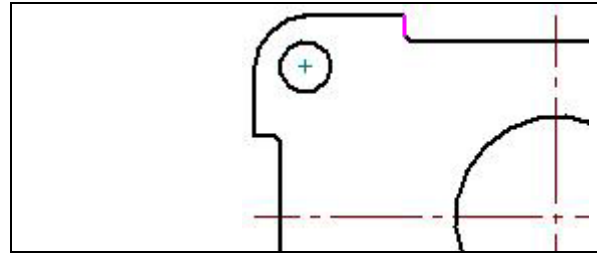
## The Distance Between Command

The first dimension you will place is the 1.50 from the left corner of the indent to the center line. Also, before you start adding dimensions, make sure you activate the **Dimension** layer by double clicking on it the same way you selected the **Center** layer. Select the **Distance Between** command on the **Dimension** panel. To the left of the screen, the Distance Between Window will appear. In the box indicating **Round-Off**, select the two decimal round-off, or **.12** setting (Figure 3.13). Next, pick the vertical line on the left side of the indent (Figure 3.14). Then pick the center line. Now place the 1.50 dimension as in Figure 3.15. If the Round-Off setting changes to the default three decimal places, be sure to change it back before you continue to dimension your drawing. Also, make sure you right click after drawing every dimension or else you may receive an error message.

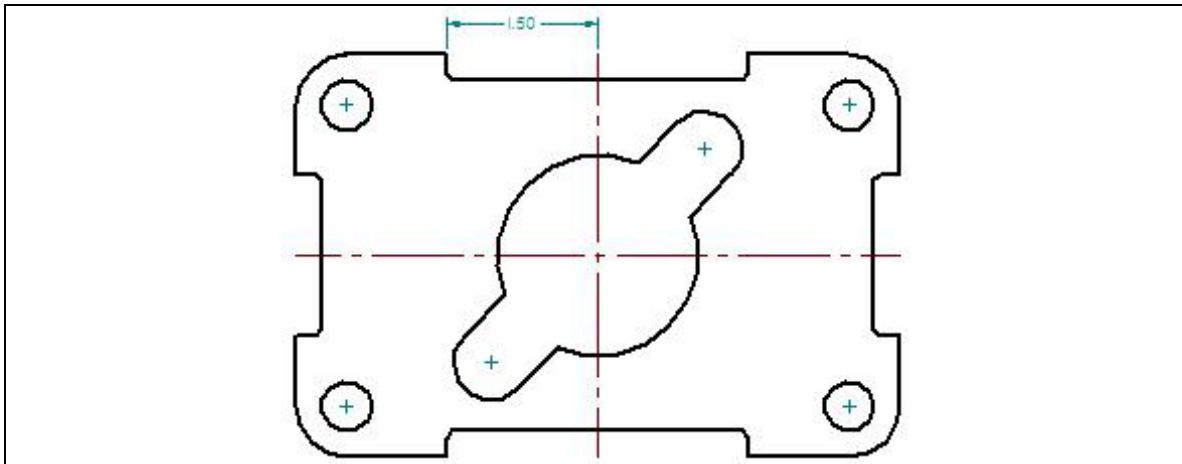




**Figure 3.13 – Select .12 Round-Off**

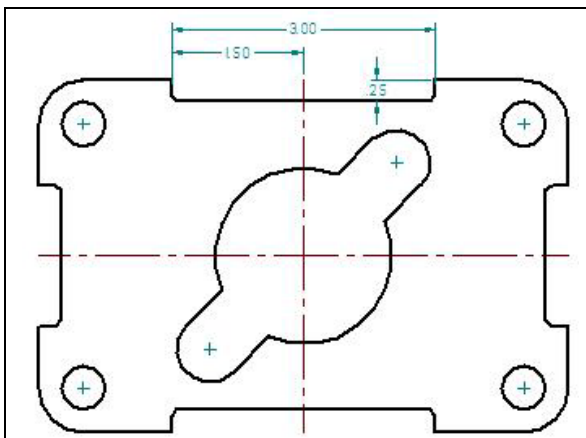


**Figure 3.14 – Selecting the Vertical Line**



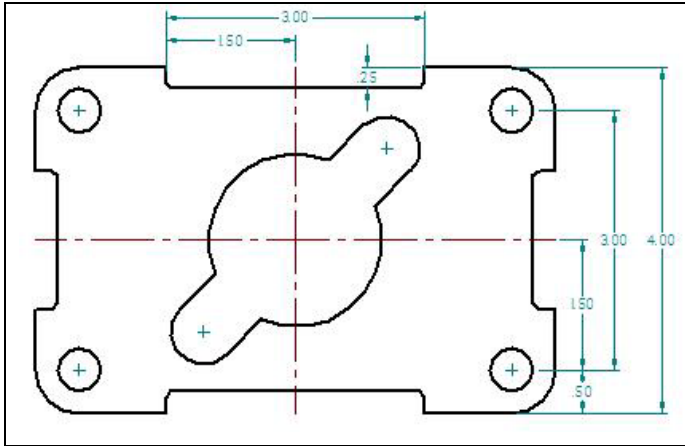
**Figure 3.15 – Your First Dimension**

When placing more dimensions, it is important to specify extension line locations off of the part and not off of any existing dimension extension lines. The problem with dimensioning off of dimension extension lines is that if your customer or supervisor removes a detail and the dimension that goes with it, the other dimensions referencing from the dimension extension line will be hanging out in space. All dimension extension lines must be referenced from the original part or centerlines.



**Figure 3.16 – 3 Top Dimensions**

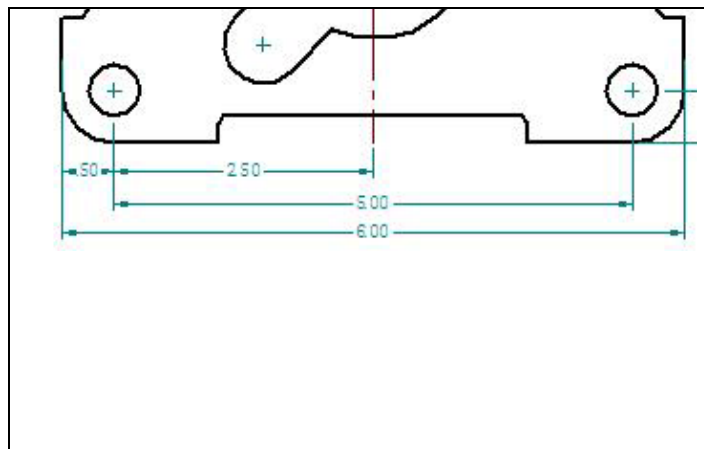
To improve your efficiency, finish dimensioning the top of the part, placing the 3.00 and 0.25 dimensions as shown in Figure 3.16. The Distance Between tool allows you to place horizontal or vertical dimensions by pulling the dimension towards the direction of placement. Do not worry that the lower dimension extension line of the 0.25 dimension is on the part. You will turn off that dimension extension line later.



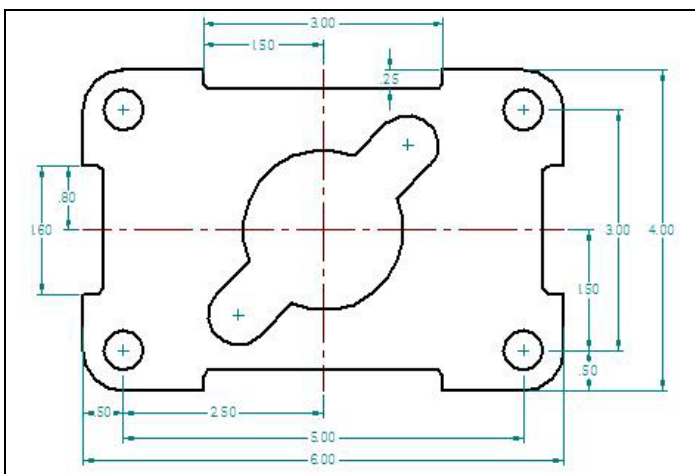
Continue to use the **Distance Between** tool to dimension the right side of the part. Always dimension from the inside out so you always have room for more dimensions. Do the 1.50, 3.00 and 4.00 dimensions trying to space them evenly. To place the 0.50 dimension, select the right quadrant of the 0.5 circle and then the bottom of the part. Try to line it up with the 1.50 dimension.

**Figure 3.17 – Dimensioning the Right**

Following a clockwise pattern, now place the dimensions on the drawing. Dimension extension lines may cross each other, but according to ANSI and ISO worldwide drafting standards, dimension lines may never cross each other. Do the 2.50, 5.00 and 6.00 dimensions in order. Lastly, place the 0.50 dimension, once again aligning it with the 2.50 dimension.



**Figure 3.18 – Dimensioning the Bottom**

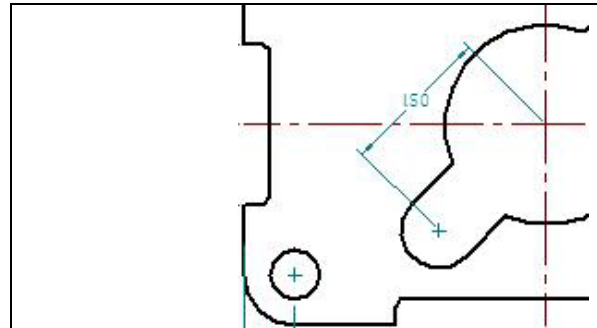
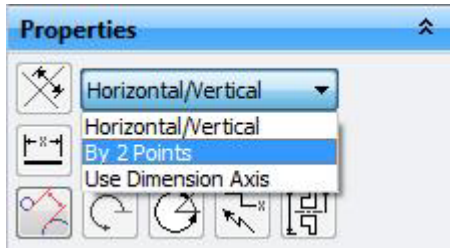


**Figure 3.19 – Dimensioning the Left**

Place the last two dimensions on the left side of the drawing, doing the 0.08 dimension first and the 1.60 dimension second. There are 13 linear dimensions on this part.

Continuing with the **Distance Between** command, you will now dimension the distance from one of the 0.75 radii and the center of the part and the distance between the two radii. However, before you select the points, you need to change the means by which you

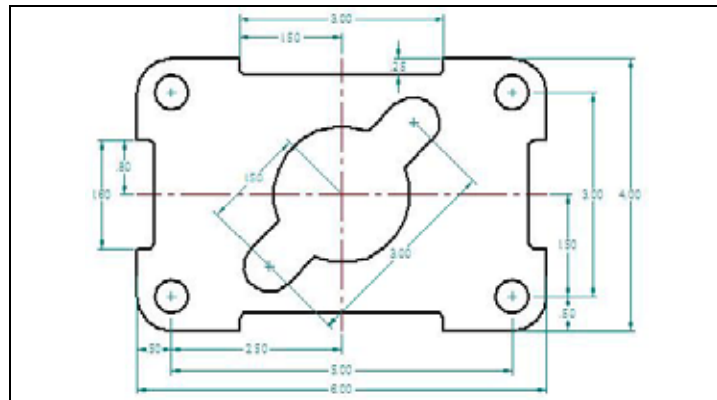
draw the dimension. In the Distance Between Window, in the Properties box, you need to change the drop-down box to the “By 2 Points” setting (Figure 3.20). This will allow you to properly draw the dimensions in the drawing. Now, pick the center of the left 0.75 radius and then pick the midpoint of one of the centerlines. Place the 1.50 dimension above and to the left of the inner detail like in Figure 3.21.



**Figure 3.20 – Selecting “By 2 Points”**

**Figure 3.21 – Finished 1.50 Dimension**

Next you will make a 3.00 dimension by selecting the centers of the two 0.75 radii. Select the first center point and, after selecting the second, place the 3.00 dimension to the lower right (Figure 3.22).



**Figure 3.22 – Finished 2 Point Dimensions**

## Using the Smart Dimension Tool

The Smart Dimension tool in the Solid Edge software is a very versatile tool that allows you to dimension many different elements of a part without using a specific dimensioning tool. In particular, you are going to be using it to add the dimensions to the filleted radii as well as the diameters of the circles. To begin, select the **Smart Dimension** tool from the **Dimension** panel. Next, you want to dimension the filleted corners of the drawing so select the lower left radius (Figure 3.23). Since we are using the Smart Dimension tool, the software automatically recognizes that it is a 0.50 radius and dimensions the part accordingly (Figure 3.24). You only need to dimension one of the corner radii because they are all the same. Next, you are going to dimension the 0.38 radius of the inner detail. Begin by selecting the upper right radius of the inner part and it will once again automatically give you the 0.38 radius measurement (Figure 3.25).



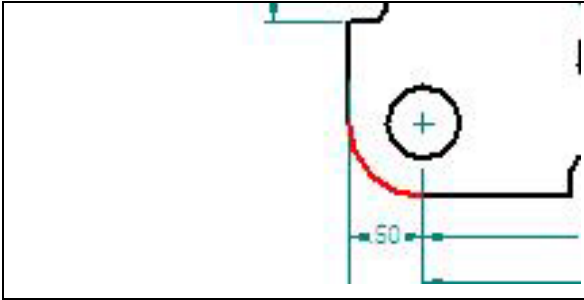


Figure 3.23 – Selecting the Corner

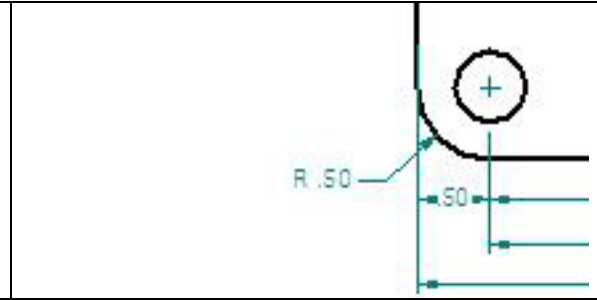


Figure 3.24 – Dimensioned 0.50 Radius

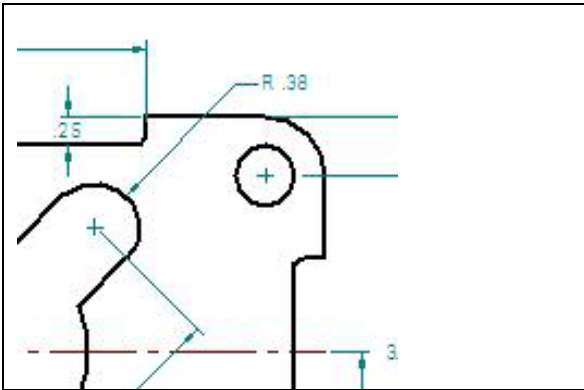


Figure 3.25 – Dimensioned 0.38 Radius

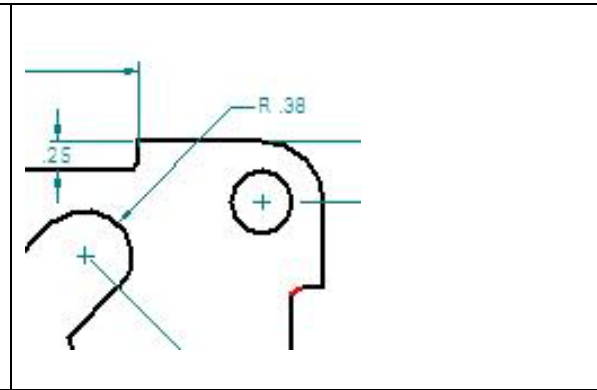


Figure 3.26 – Selecting 0.06 Radius

The last radius to dimension is the 0.06 radius that rounds the inner corner of the indent. Select the top radius of the right indent as in Figure 3.26. Place the measurement to the left and inside the part (Figure 3.27).

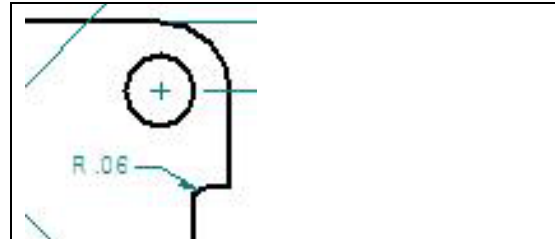


Figure 3.27 – Dimensioned 0.06 Radius

As stated before, the **Smart Dimension** tool can be used for a number of different dimensions. You will now be using it to dimension the diameters of two different circles, a 0.50 diameter circle and a 2.00 diameter circle. With the Smart Dimension tool selected, pick the top left 0.50 diameter circle (Figure 3.28). The diameter dimension will appear as a circle with a slash through it. Place the measurement outside the part and to the upper left (Figure 3.29). Next, select the 2.00 diameter circle, which is the middle part of the inner detail (Figure 3.30). If the measurement is being read as a radius, you need to go to the Smart Dimension Window on the left of the screen and select the **Diameter** tool (Figure 3.31). This will change the dimension from a 1.00 radius to a 2.00 diameter, which is what we want. Place the measurement inside the part as in Figure 3.32.



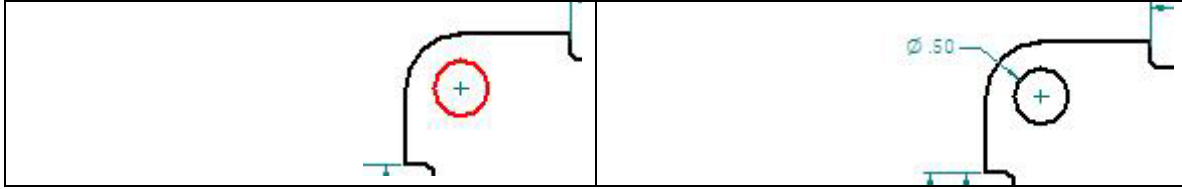


Figure 3.28 – Selecting the 0.50 Circle

Figure 3.29 – Placing the 0.50 Dimension

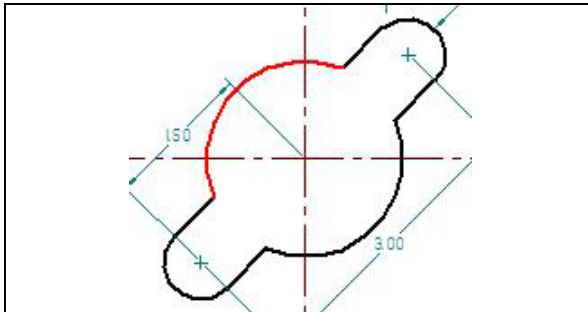


Figure 3.30 – Selecting the 2.00 Circle



Figure 3.31 – Diameter tool

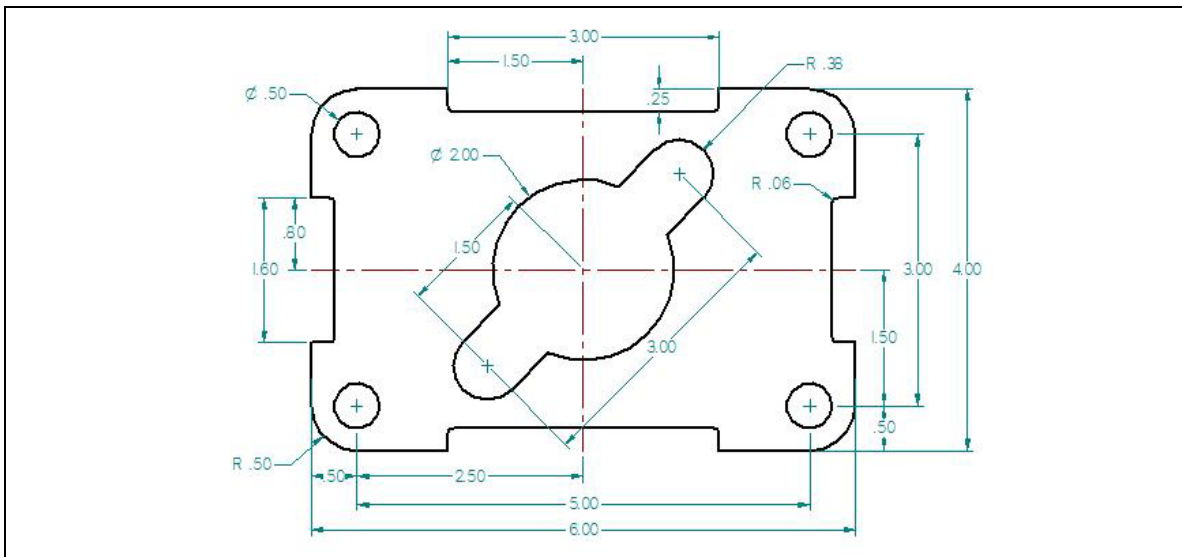
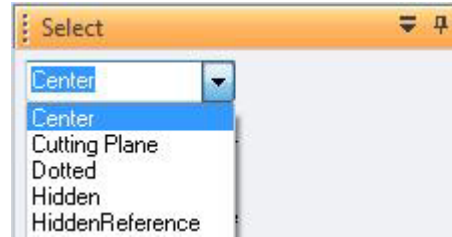
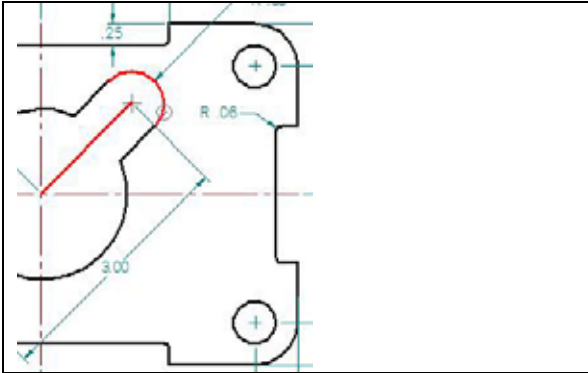


Figure 3.32 – Full Drawing with 2.00 Diameter and Other Dimensions

## Using the Line Tool to Assist Dimensioning

The next dimension you are going to put into the drawing is the 45 degree rotation in the center detail. To do so, you will need to use the **Line** tool. The **Line** tool is on the **Drawing** panel of the **Sketching** tab. Select the command and begin to draw your line at the midpoint of one of the centerlines. Then, draw the line to the right 0.38 radius center mark (Figure 3.33). When drawing the line, make sure that the layer is set to **Center** (Figure 3.34). With this line in place, you will be able to use the **Angle Between** tool to measure the angle between the horizontal centerline and the newly drawn line.

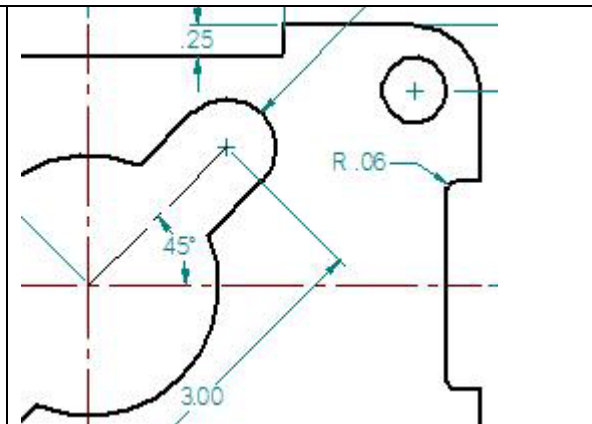
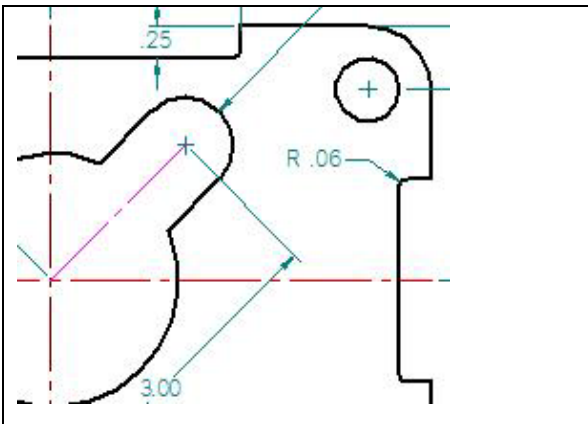




**Figure 3.33 – Choosing the Center Mark**      **Figure 3.34 – Selecting “Center”**

## Using the Angle Between Tool

Another easy command to use and the last dimensioning tool in this problem is the **Angle Between** tool. It is located on the Dimension panel. Select the Angle Between tool and then select the two lines making the angle. First, select the line you just drew from the center of the part to the right center mark of the 0.38 radius. The second line you want to select is the horizontal centerline. One thing to note is that when you are selecting the centerline, make sure you select on the outside of the center detail or else you will not get the correct dimension (Figure 3.35). If done correctly, an angle measurement of 45 degrees should appear between the two lines. Place the measurement on the inside of the center detail (Figure 3.36).



**Figure 3.35 – Selecting the Two Lines**      **Figure 3.36 – Finished 45° Dimension**

There are a few changes you need to make to finish the dimensioning of the part to add to the clarity of the information. Three centerlines need to be removed from the drawing. A dimension extension line needs to be removed from covering the object line on the 0.25 dimension. Also, you need to add “4 holes” under the 0.50 diameter dimension text.

## Adding Text below the 0.50 Diameter Dimension

To add text to the 0.50 diameter dimension stating that this description applies to “4 holes”, select the 0.50 dimension with the **Select** tool. Next, pick the 0.50 dimension and the Edit Definition Window will appear to the left of the screen. On the Tolerance pane of the window, select the **Edit Definition-Prefix** tool (Figure 3.37). A window will appear on the screen as in Figure 3.38. In the box labeled **Subfix**, type “4 holes” (Figure 3.39). After typing in the text, click the okay button and the text will be added below the dimension (Figure 3.40).

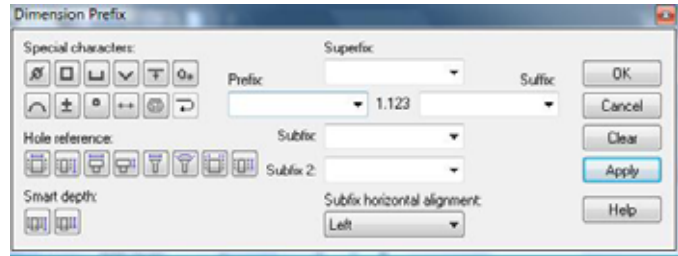
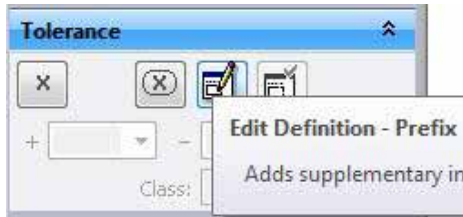


Figure 3.37 – Edit Definition tool

Figure 3.38 – Dimension Prefix Window

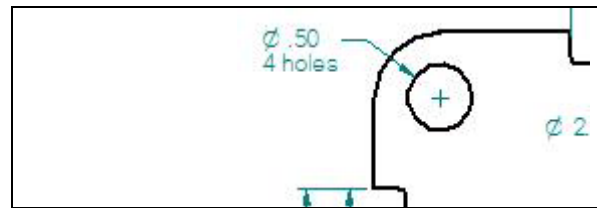
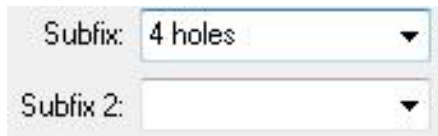


Figure 3.39 – “4 hole” Subfix

Figure 3.40 – Finished Text

Now that the Rectangle Problem has the “4 holes” added to the 0.50 diameter, this problem is close to being finished. The last set of procedures will add text to the drawing, which will be explained next chapter.

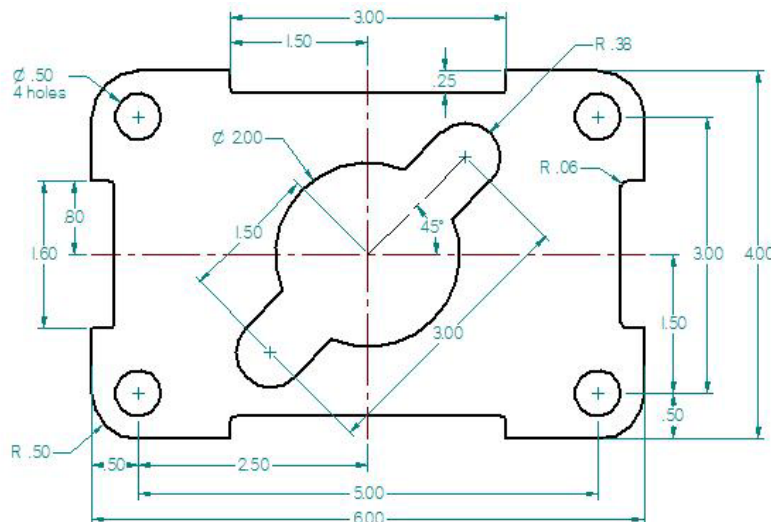


Figure 3.41 – Finished Dimensions on the Rectangle Problem

**\* World Class Challenge 22-08 \* - Close the drawing file. Create a New file and draw the entire rectangular problem on the proper layers and using proper dimensions on the drawing. Continue this drill four times, each time completing the drawing in less than 10 minutes to maintain your World Class ranking. If you are under 10 minutes, you may proceed to the next chapter, the Circular Problem.**