

## Dimensioning the Rectangular Problem

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

- **Creating new layers in an AutoCAD drawing**
- **Placing Centerlines on the drawing**
- **Changing entities from one layer to another**
- **Placing center marks on the drawing**
- **Using of the Dimensioning toolbar**
- **Using the Linear Dimension command to dimension a part**
- **Setting system variables using the Dimension Style window**
- **Using the Aligned Dimension command to dimension a part**
- **Using the Radius Dimension command to dimension a part**
- **Using the Diameter Dimension command to dimension a part**
- **Using the Break Command to shorten a line**
- **Using the Angular Dimension command to dimension a part**
- **Using the Properties Tool to Modify a Dimension Variable**
- **Using the Edit Text Tool**

## Creating New Layers in an AutoCAD 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, think of layers as you would a sandwich: each different item has its own flavor and color, but when you stack them all together you have a complete, delicious meal. 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 layer 0, the AutoCAD default layer. You can see the current layer listed in the Layer Control list box in the Layers panel under the Home tab. (Figure 3.1). To create new layers using the Layers Properties Manager, select the Layers command button from the Layers panel.

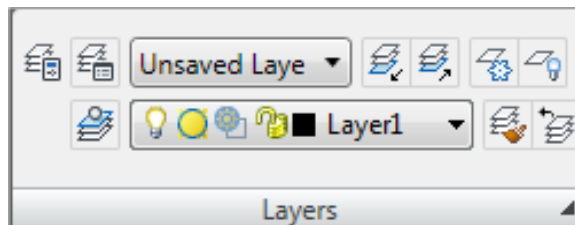


Figure 3.1 – The Layers panel

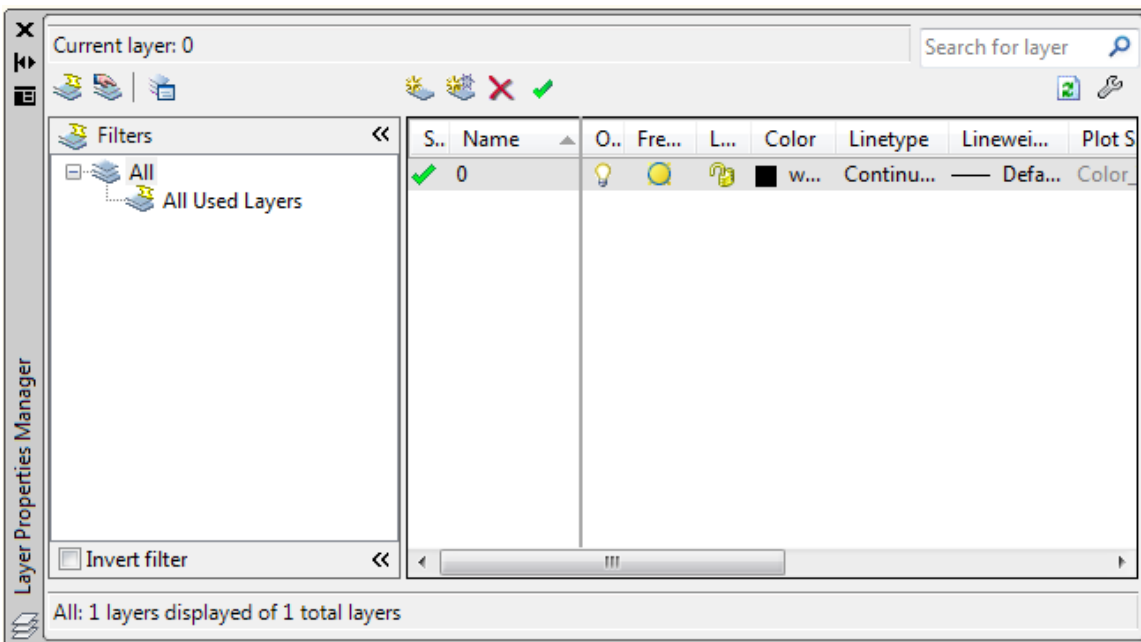
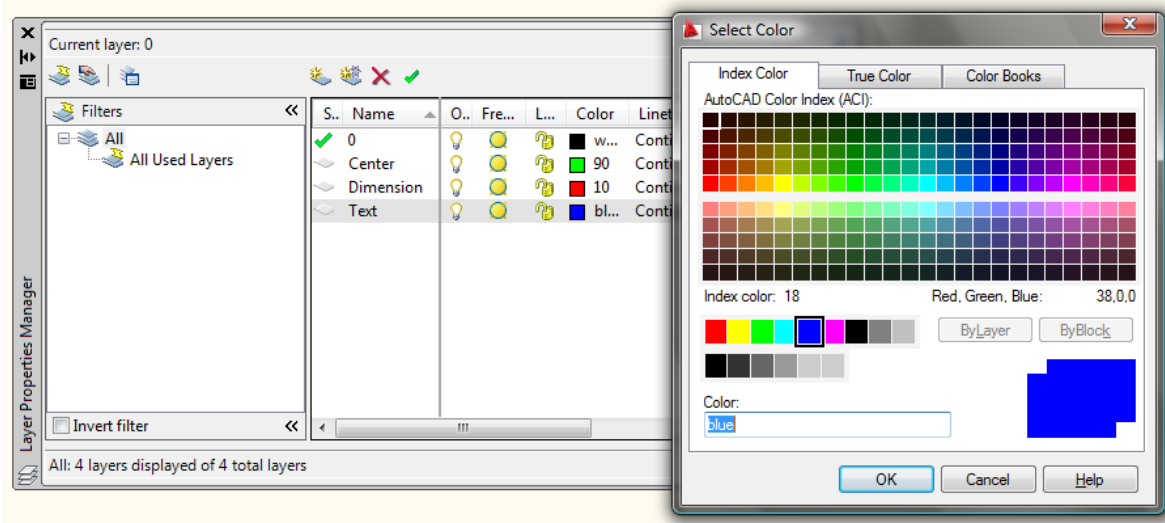


Figure 3.2 – The Layer Properties Manager window

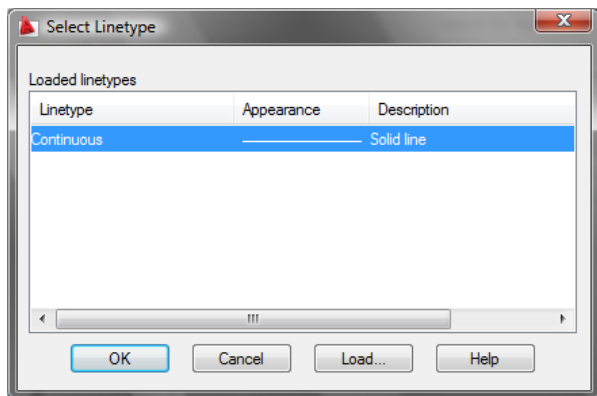
To dimension your part, you need make three new layers: the Dimension layer for dimensions, the Center layer for centerlines, and the Text layer for text and notes. To create the layers, select the New button to add a new layer under the 0 layer. Immediately, the layer name is ready for editing, so type “Dimension” and hit ENTER twice to add your next new layer. Type “Center”

and hit ENTER twice to add another new layer. Type “Text” and ENTER once to finish adding new layers. Next, you need to assign the layers different colors to help distinguish between them. To change the Dimension layer color, place your mouse arrow over the color box in the Dimension layer row and single left click with your mouse. The Select Color window will appear in front of the Layers Properties Manager (Figure 3.3). Select red from the Standard Colors row and select the OK button. You will now see that Dimension layer is red. Repeat the process to make the Center layer green and the Text layer blue.

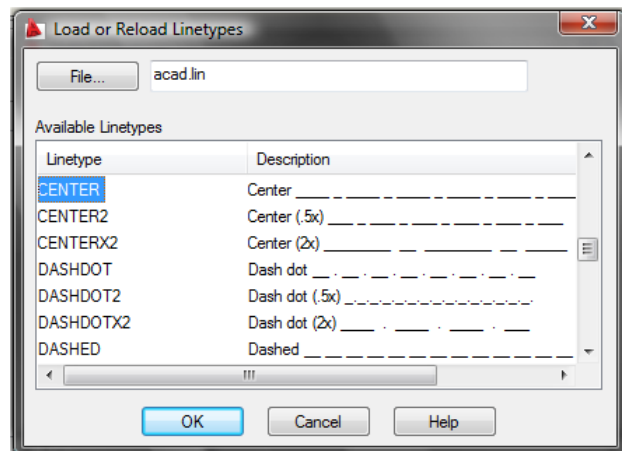


**Figure 3.3 – The Select Color window**

To finish creating and modifying layers, you need to change the Continuous linetype to a Center linetype for your Center layer. Place your mouse arrow over the text “Continuous” in the Center layer row and single left click with your mouse. The Select Linetype window will appear in front of the Layers Properties Manager (Figure 3.4). Note that only the Continuous Linetype is loaded, so select the Load button and the Load or Reload Linetypes window will appear. Scroll down, highlight the CENTER linetype as shown in Figure 3.5, and select the OK button. Highlight the CENTER linetype in the Select Linetype window and select OK. You will now see that the Center layer has a CENTER linetype.



**Figure 3.4 – The Select Linetype window**



**Figure 3.5 – Load or Reload Linetypes**

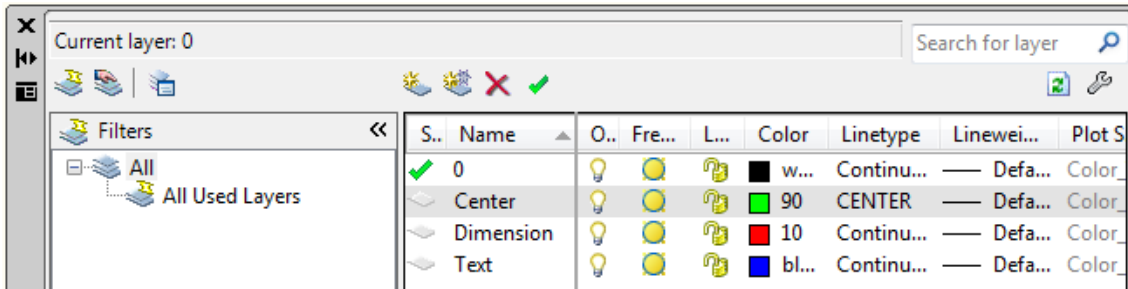


Figure 3.6 – Four completed layers

## Placing Centerlines on the Drawing

Now you can set up your centerlines. You placed the centerlines on the drawing right after drawing the 6 by 4 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 to begin with. Pick the vertical and horizontal centerline, noticing that they have become dotted. Select the dropdown arrow to the right of the Layers Control list box to open the list. Highlight the Center Layer.

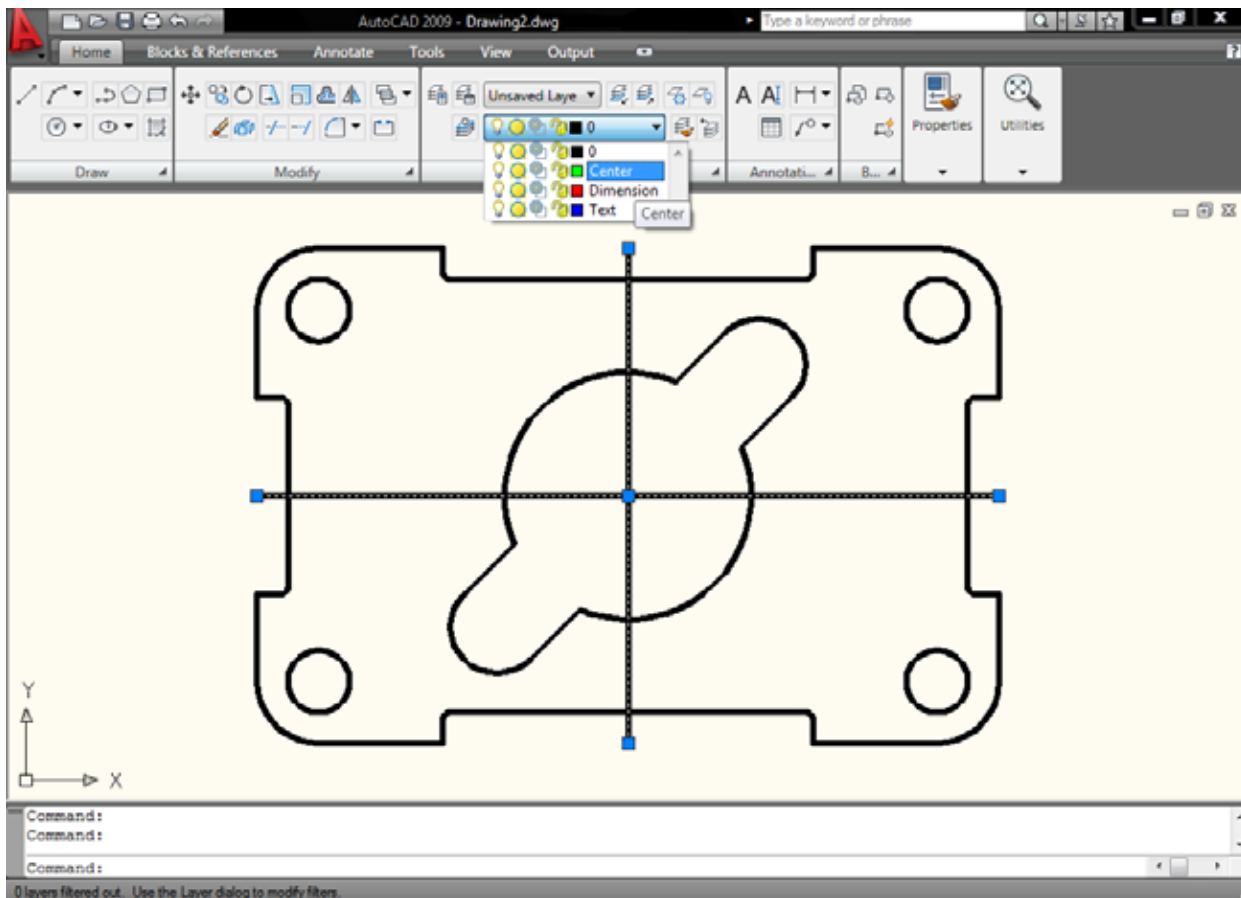
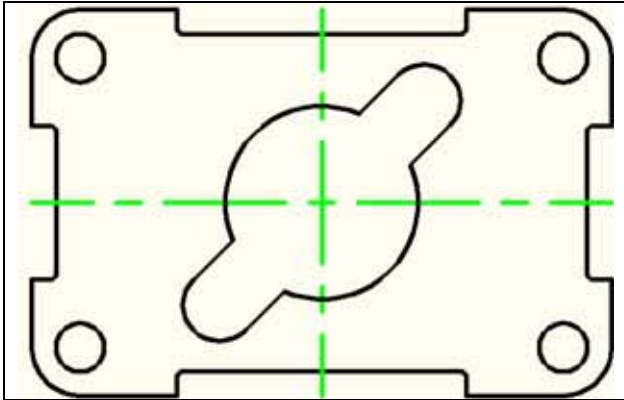


Figure 3.7 – Selecting the Center layer in the Layer Control list box



The Center layer replaces the 0 layer to show that these two lines now reside on the Center layer. Press the ESC key on your keyboard and the lines are no longer dotted and are removed from the selection set. Centerlines are critical in the dimensioning process since they show symmetry in a part and reduce the amount of dimensions on the drawing.

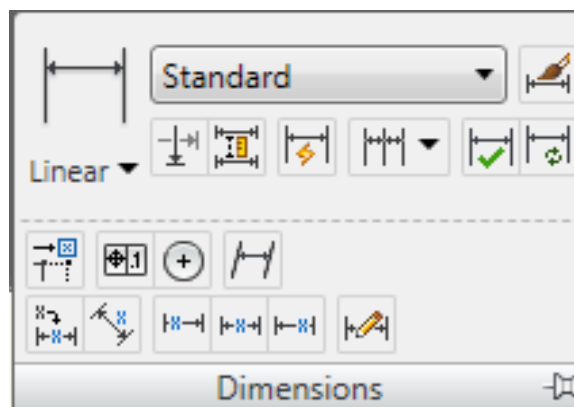
**Figure 3.8 – The two centerlines**

## Placing Center Marks on the Drawing

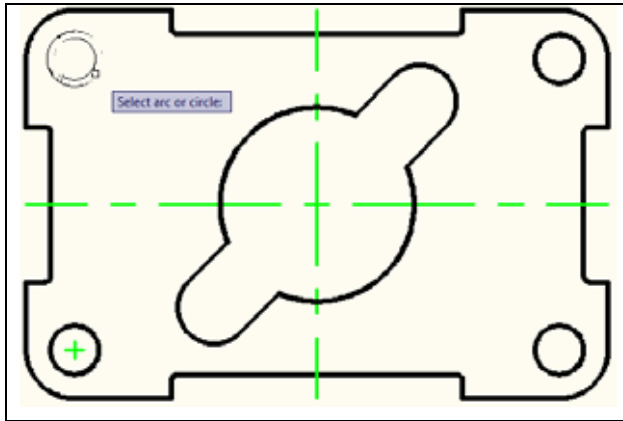
Dimensioning is the easiest task to be performed in CAD. By completing the difficult work in the last chapter, you are now rewarded with picking object snap points and laying out a visually pleasant set of dimensions. Technicians with any artistic talents will enjoy the ability to have freedom of layout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



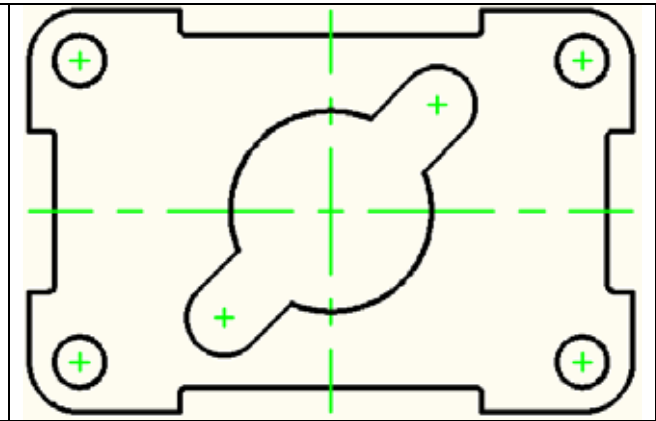
In the Layer Control list box, select the Center Layer. When you add the center marks to the drawing, they will be green. Next, you need to place center marks inside the four 0.5 circles and inside the 0.75 arcs on the inner detail. Select the Center Mark tool on the Dimension panel under the Annotate tab. (Once again, you may have to expand the Dimension panel by clicking the small black triangle in the bottom right corner of the panel to find the Center Mark tool.) The mouse arrow will turn into a pick box. Select the 0.5 circle on the lower left corner of the part (Figure 3.10). A green center mark appears inside the circle. Press ENTER to repeat the command. 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 3.11).



**Figure 3.9 – The Dimension panel, expanded**



**Figure 3.10 – Placing center marks**

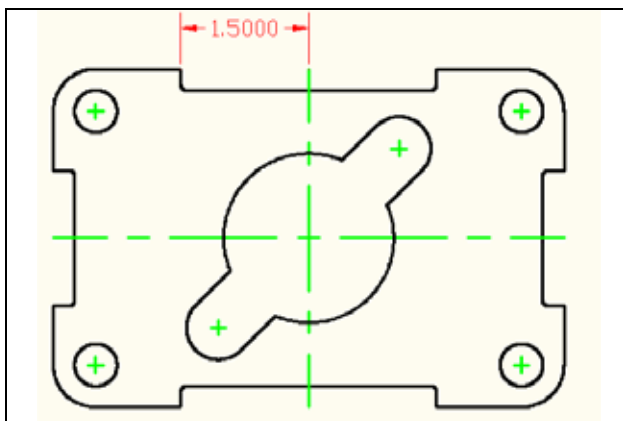


**Figure 3.11 – All 6 center marks placed**

Did you see how easy the dimensioning tools work? As long as you draw your part accurately, the associated dimensions in the CAD software will do the rest of the work. Use the reported dimensions to check your work.

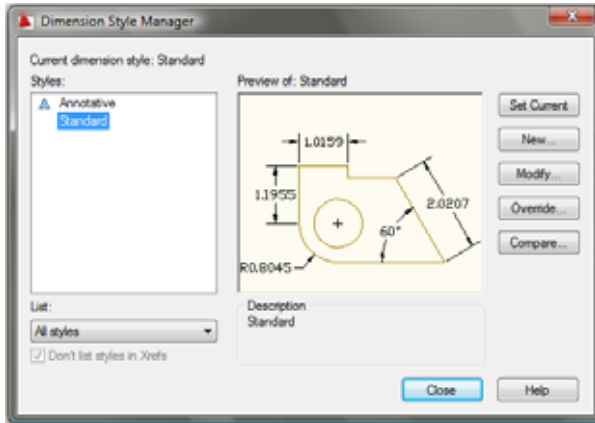
## The Linear Dimension Command

The first dimension you will place is the 1.5000 from the left corner of the indent to the centerline. Before placing any dimensions, make sure to select the Dimension layer from the Control list box. Next, select the Linear Dimension tool on the Dimension panel under the Annotate tab. The command line will ask you to “Specify first extension line origin or <select object>”, so select the endpoint at the top of the indent. The command line will immediately prompt you to “Specify second extension line origin”, so select the endpoint at the top of the vertical centerline. The next prompt on the command line will be to “Specify Dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]”, so abstractly select a point above the part so the dimension is neatly placed close to the part but none of the dimension, dimension lines, and dimension extension lines are on the object lines of the part (Figure 3.12)



**Figure 3.12 – Your first dimension**

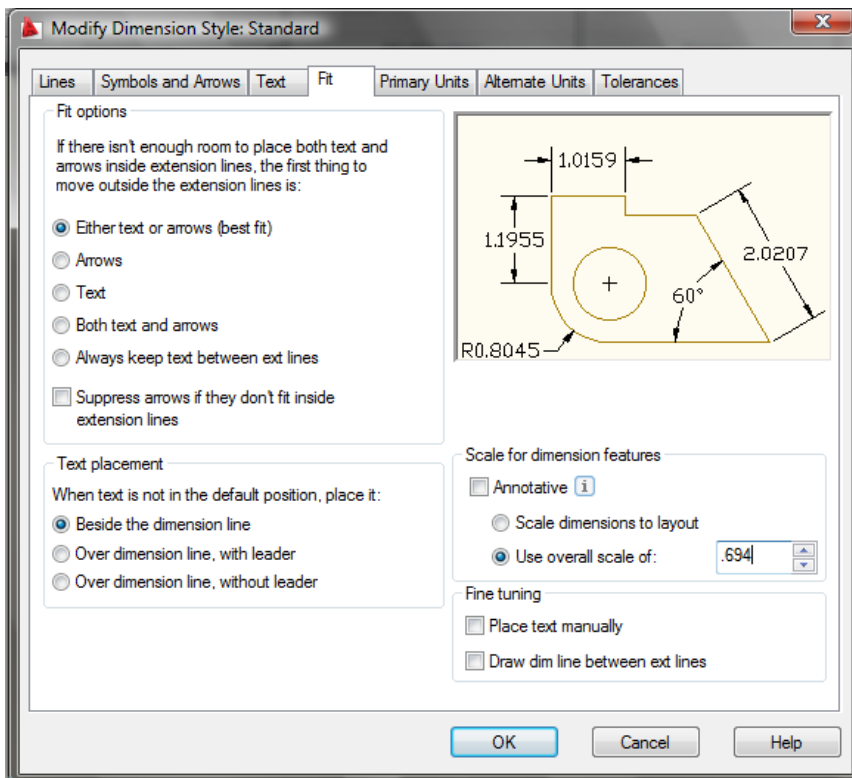
The 1.5000 dimension is a four placed decimal number. On a sheet metal part that is going to be held to (plus or minus) 0.010-inch tolerance, a four place decimal is unnecessarily accurate and may cause confusion, as some companies use decimal places to convey tight tolerances. On our drawing, you need to change the dimension to a 2-place decimal. Also, the dimension text is 0.18 high. Our drawing only needs dimension text that is 0.125 high.



To change the system variables controlling dimensions, select the Dimension Style tool on the Dimension panel under the Annotate tab to open the Dimension Style Manager window. Select the Modify button to access the dimension variables (Figure 3.13).

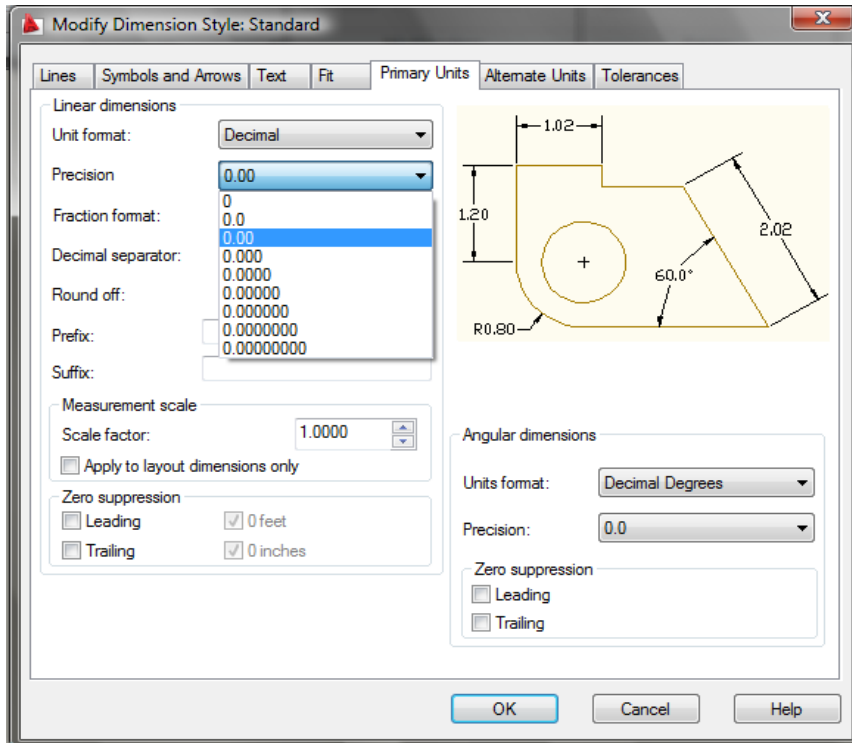


**Figure 3.13 – The Dimension Style Manager window**



There are 7 tabs in the Modify Dimension Style window: Lines, Symbols and Arrows, Text, Fit, Primary Units, Alternate Units, and Tolerances. You need to select the Fit tab and change the overall scale of dimension from 1.0 to 0.694 (Figure 3.14). There is only one textbox in this window, so it is easy to locate and make the change. Next, go to the Text tab and change the text size to 0.125.

**Figure 3.14 – The Dimension Style Manager window, Fit tab**



Select the Primary Units tab to change decimal places. The form is broken into Linear Dimensions and Angular Dimensions halves. In the Precision list box in the Linear Dimensions section, select the dropdown menu to open the list and highlight the 0.00 in the list to change from 4-place to 2-place decimal. In addition, you can change the Precision in the Angular Dimensions list box to 0.0 (Figure 3.15).

Figure 3.15 – The Dimension Style Manager window, Primary Units tab

## More Linear Dimensions

You can see that the 1.50 dimension is to our standard (Figure 3.16), so now you can place additional dimensions on the part. You do not want to haphazardly add dimensions; rather, you should follow a pattern to again improve efficiency and avoid errors of omission, like forgetting a dimension.

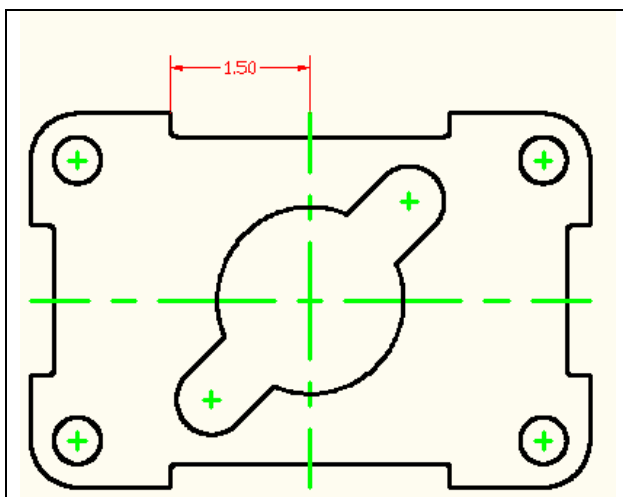
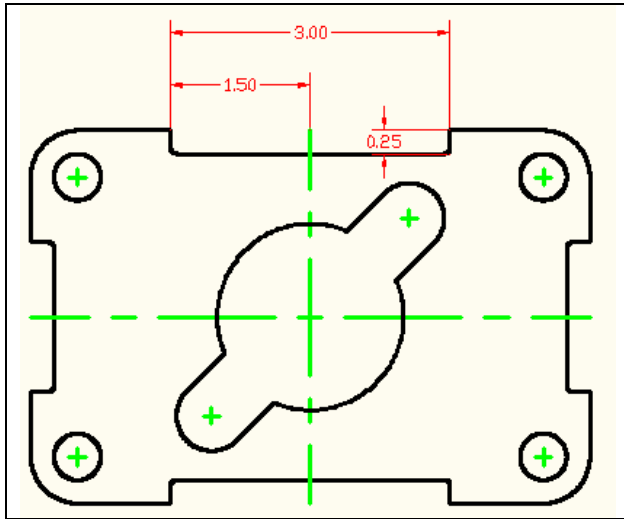


Figure 3.16 – The reformatted dimension

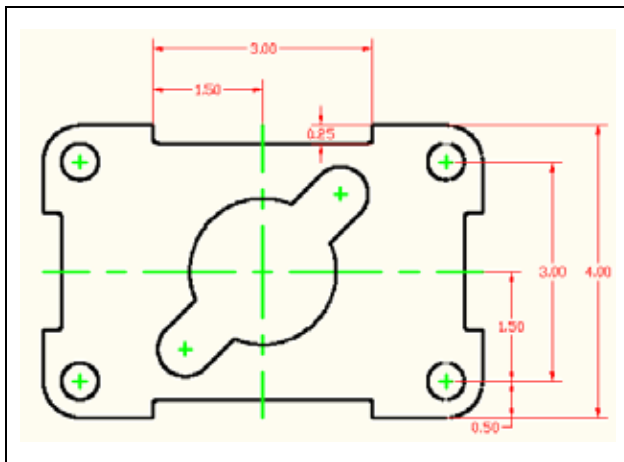
When placing new dimensions on a drawing, continue to specify extension line locations off the part (black line) and not off any existing dimension extension lines (red lines). The problem with dimensioning off dimension extension lines is that if your customer or supervisor removes a detail the dimension that goes with it; the other dimensions referencing from the dimension extension lines will be hanging out in Model Space. All dimensions extensions lines must reference from the original part or centerlines.





**Figure 3.17 – 3 linear dimensions**

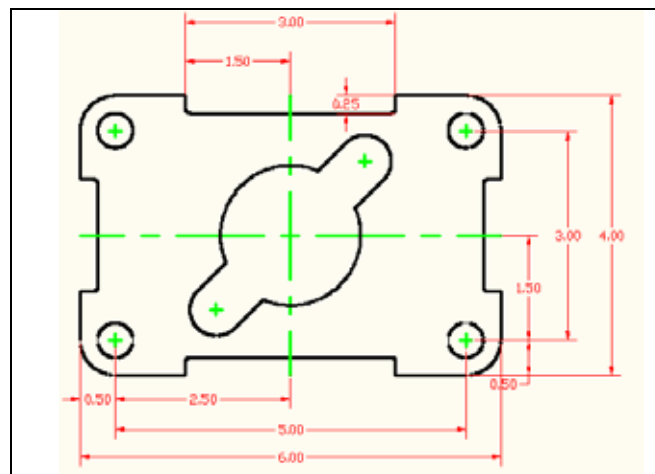
To improve your efficiency, finish dimensioning the top of the part, placing the 3.00 and 0.25 dimensions as shown in Figure 3.17. The Linear Dimension tool allows you to place horizontal or vertical dimension by pulling the dimension in the third step towards direction of placement. Do not worry that the lower dimension extension line of the 0.25 dimension is on the part. You will turn off that dimension extension line later using the Properties tool on the Standard toolbar.



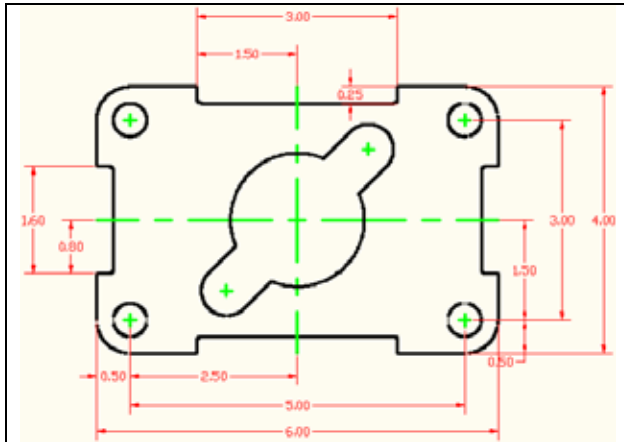
**Figure 3.18 – Dimensioning the right side**

Continue to use the Linear Dimension command by pressing ENTER each time to repeat the command. Always dimension from the inside to the outside of the part, so you always have room for more dimensions. Do the 1.50, the 3.00 and 4.00 dimensions, trying to space them evenly (Figure 3.18). To place the 0.50 dimension, select the right of the 0.50 centerline and the part. For the third selection to show position, select the 1.50 dimension line. The dimension lines will align.

Following a clockwise pattern, now place the dimensions on the bottom of the drawing. Dimension extension lines may cross each other, but according to ANSI and ISO worldwide drafting standards, dimension lines (red lines with arrows) may never cross each other. Do the 2.50, 5.00 and 6.00 dimensions in order. Lastly, place the 0.50 dimension, reinforcing the alignment technique you have just learned. Make the third selection when placing the 0.50 dimension on the 5.00 dimension line.



**Figure 3.19 – Dimensioning the bottom side**



Place the last two dimensions on the left side of the drawing, doing the 0.80 dimension first and the 1.60 dimension second. There are 13 linear dimensions on this part.

**Figure 3.20 – Dimensioning the left side**

## The Aligned Dimension Command

Now, we are ready to place the two Aligned Dimensions. The Aligned Dimension tool is used to easily place a dimension in alignment with two picked reference points. You are going to use the Aligned Dimension command to place two dimensions on the drawing: to define the distance from the center of the part to the 0.75 radius, and the distance from both 0.75 radii. In order to easily select the intersecting point of the centerlines, you need to change your Osnap settings.



Right click the Object Snaps tool on the bottom of the screen and select Settings from the menu to bring up the Object Snaps tab in the Draft Settings window. Select the Clear All button to clear all previous Osnap settings. Check the Intersection checkbox and press OK to close the window. Now the other Osnaps will not interfere with your ability to select a reference point. Feel free to change Osnap settings at key times in the drawing process, such as between major command functions like drawing lines and circles or when moving from drawing to dimensioning.



Select the Aligned Dimension tool by clicking the word Linear on the Dimension panel to open a drop-down menu. You can then select the Aligned Dimension tool. Place the 1.50 aligned-dimension on the drawing by selecting the intersection of two centerlines and positioning the dimension as shown in Figure 3.21. Next, place the 3.00 aligned-dimension on both 0.75 radius centerlines and position the 3.00 dimension text as shown in Figure 3.22.

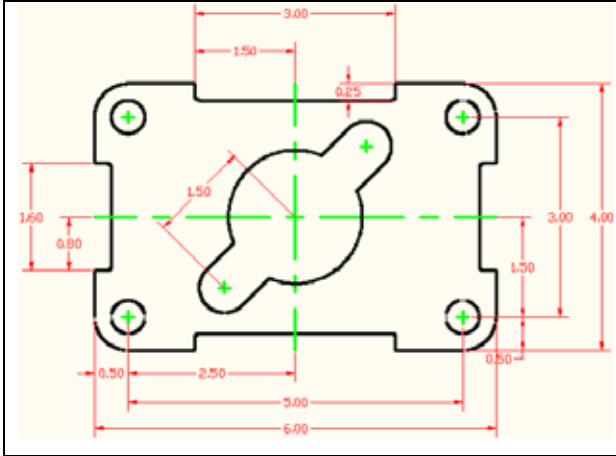


Figure 3.21 – 1<sup>st</sup> aligned dimension

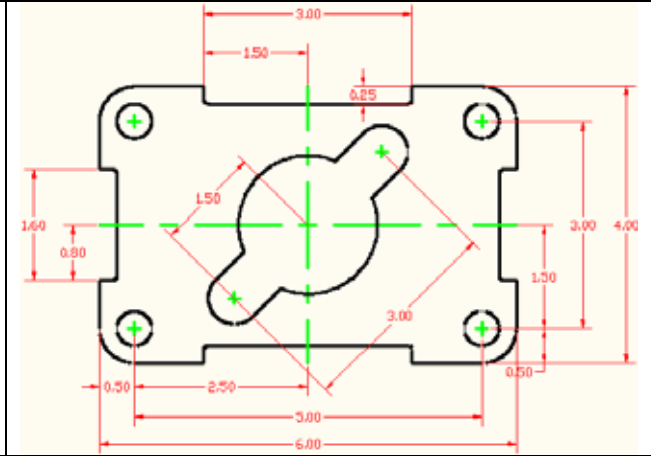



Figure 3.22 – 2<sup>nd</sup> aligned dimension

## Using the Radius Dimension Tool

The Radius Dimension tool is as easy to use as the Center Mark tool. Select the Radius Dimension tool in the same way you selected the Aligned Dimension tool. The mouse pointer will change from an arrow to a pick box to allow you to choose an arc or circle in which to place a radius. 

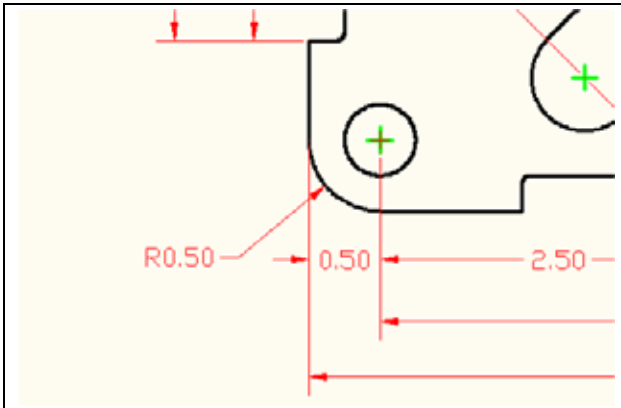
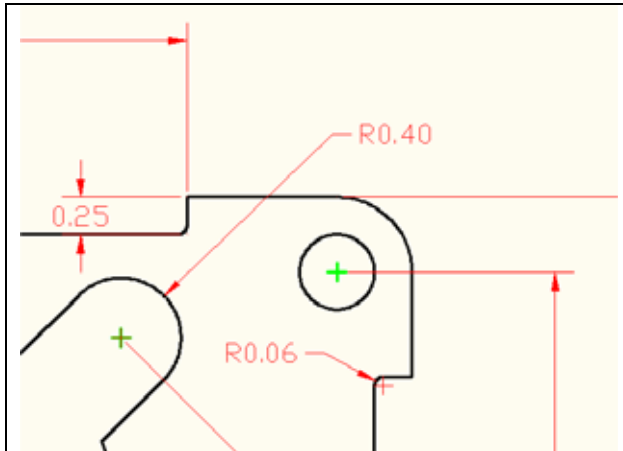


Figure 3.23 – First radius dimension

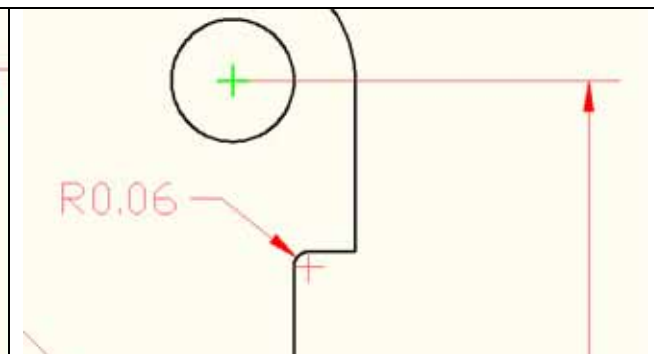
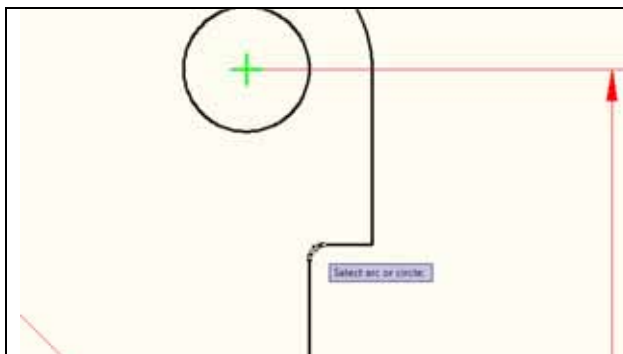
To place a 0.50 radius on the lower left corner of the part, select the Radius Dimension tool on the Dimension toolbar. Place the pick box on the 0.50 radius arc (black arc). The second pick of the mouse will place the R0.50 dimension text on the drawing. Notice that moving your mouse around the circumference of the center point will change the location of where the pointing arrow intersects with the arc. A red center mark is placed on the drawing with the radius dimension (Figure 3.23).



The second radius reinforces the lessons of placing the first radius dimension on the drawing. Again, place the pick box on the 0.40 radius arc (black arc). The second pick of the mouse will place the R0.40 dimension text on the drawing (Figure 3.24).

The third and last radius dimension on this drawing is the smallest arc and can be very hard to select. Use the corner of the pick box to grab the 0.06 radius arc (Figure 3.25) and place the R0.06 dimension text as shown in Figure 3.26.

**Figure 3.24 – Second radius dimension**

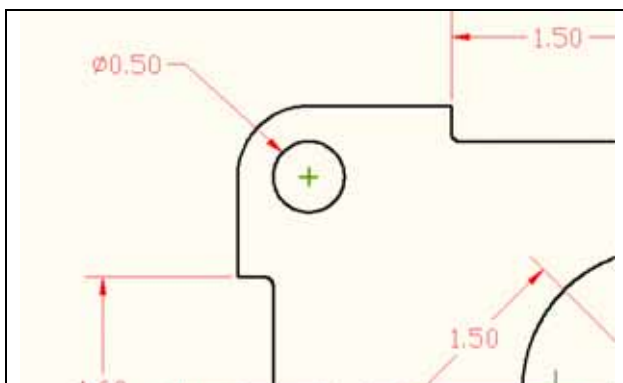


**Figure 3.25 – Selecting the last radius**

**Figure 3.26 – The 0.06 radius**

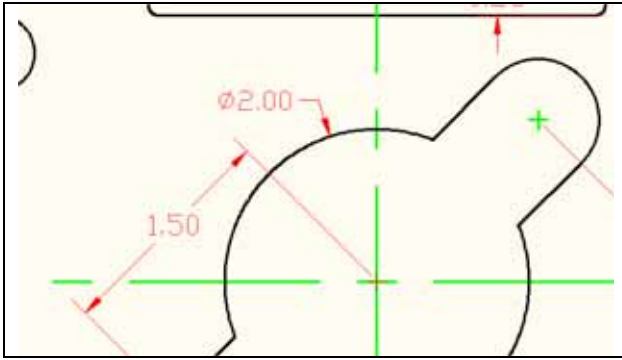
## Using the Diameter Dimension Tool

You need to place two dimensions showing diameters on the drawing. Select the Diameter Dimension tool in the same manner that you selected the Radius and Aligned Dimension Tools.



Once again, the aperture cursor will change to a pick box. Select the 0.50 circle located on the upper right hand corner of the part. Similar to the Radius Dimension command, a second selection will place the diameter text on the drawing (Figure 3.27).

**Figure 3.27 – The 0.50 diameter**



The second diameter you will place on the drawing is on the 2.00 diameter hole in the center of the part. Select the Diameter Dimension tool on the Dimension toolbar and the aperture cursor will change to a pick box. Select the 2.00 circle. Make your second selection to place the dimension text on the drawing (Figure 3.28).

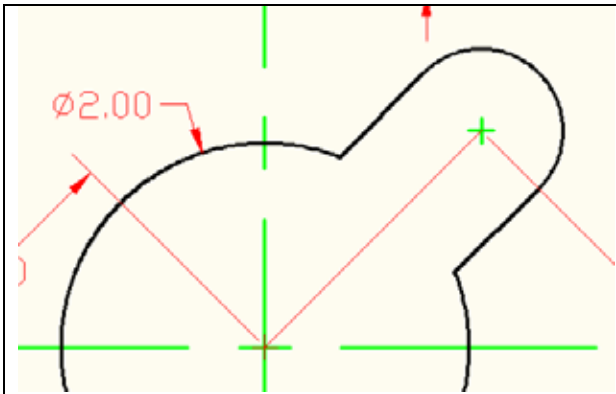
**Figure 3.28 – The 2.00 diameter**

## Using the Break Command

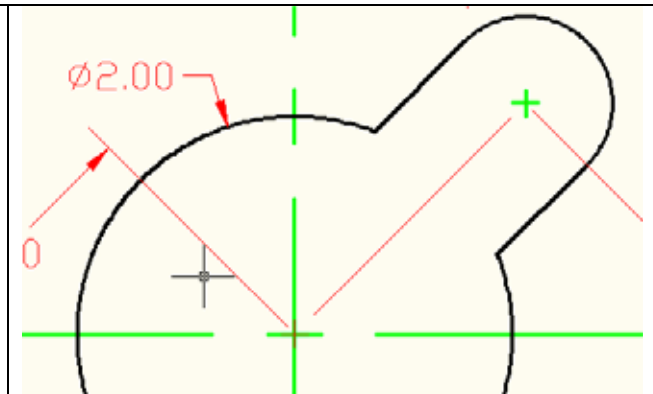
The Break command is a rarely used since the Trim command was added to the list of functions in AutoCAD. It is designed to abstractly remove portion of a line segment. Therefore, the Break command is perfect for your use when dimensioning a drawing; you are going to place your own dimension extension line in the drawing. Per ANSI or ISO standards, a dimension extension line cannot touch an object or centerline, so you will use the Break command to shorten the line segment and create a small gap between the line you added and the centerlines.



First, you need to draw a line from the intersection of the major centerline to the centerline of the upper right 0.40-radius arc. The Intersection Osnap is still on, so select the Line command on the Draw panel under the Home tab. Select the first intersection and select the first point of the line segment. Select the second intersection to create the line segment. Press ENTER to end the Line command. You have drawn your first dimension extension line without the Dimension tools (Figure 3.29).



**Figure 3.29 – Adding a line segment**



**Figure 3.30 – The broken line segment**

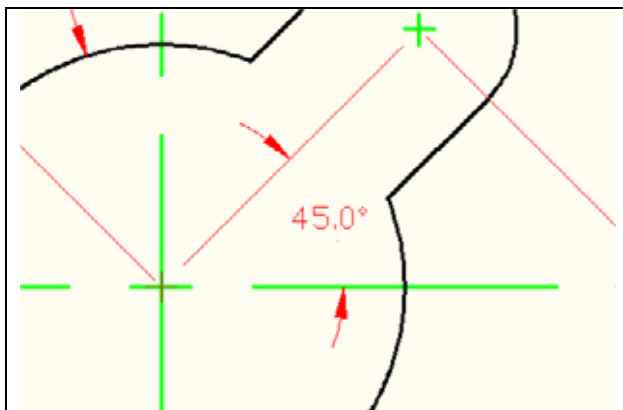
Select the Break command on the Modify panel. Now, the command line will prompt you to “select object”, but it is more complicated than just picking the line. The point that you select will be the first cutting point in the break sequence. Pick close to the bottom end of the line

segment. Next the command line will prompt you to “specify second break point or [First point]”, where you need to pick above below and to the left of the line. If you do not pick clearly to the below and to the left of the first selection point, you may end up with a small line segment. Remember, if you ever make a mistake, you can Undo it by hitting CTRL+Z or using the Undo arrow at the top left of the screen. Repeat the Break command by pressing ENTER and remove the top part of the line segment, so you end up with a shortened line as shown in Figure 3.30.

## Using the Angular Dimension Tool

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Another easy command to use and the last dimension tool in this problem is the Angular Dimension tool. It is located on the Dimension panel in the same drop-down menu as the Radius, Diameter, and Aligned Dimension tools. Select the Angular Dimension tool and select two converging lines (non-parallel) to place the degree measurement on the drawing. For the first dimension, select the red dimension extension line you just drew. Then, select the horizontal centerline directly below the dimension extension line. You will need to make a third selection to place the 45.0-degree dimension text on the drawing (Figure 3.31).



There are a few changes you need to make to finish the dimensioning of the part that will add to the clarity of the information.

Three red centerlines need to be removed from the drawing. Also, a red dimension extension line needs to be removed from covering the object line on the 0.25 dimension. Finally, you need to add “4 holes” under the 0.50 diameter dimension text.

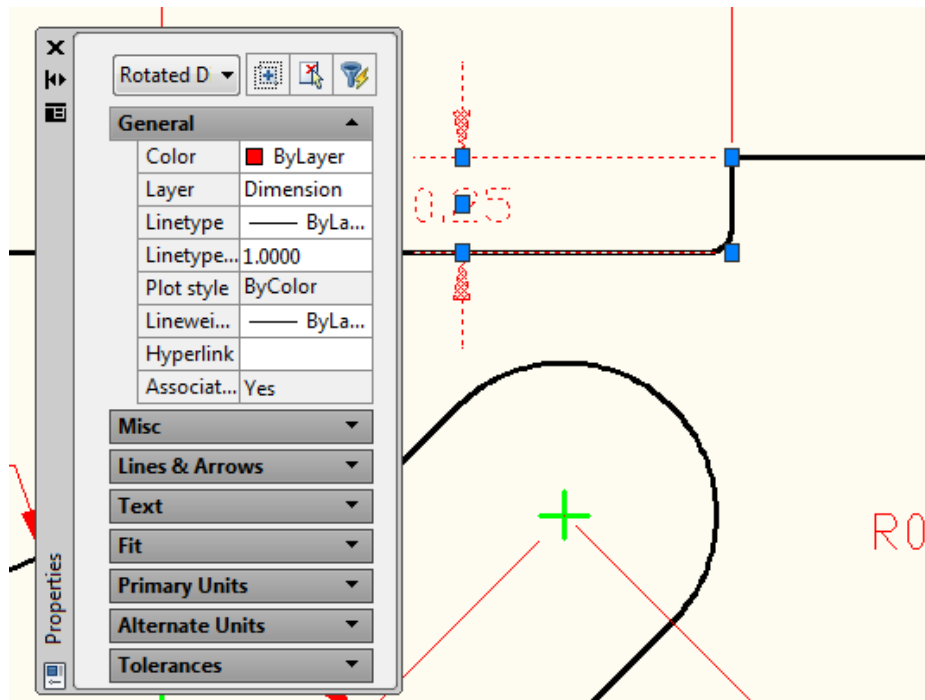
**Figure 3.31 – An angular dimension**

## Using the Properties Tool to Modify a Dimension Variable

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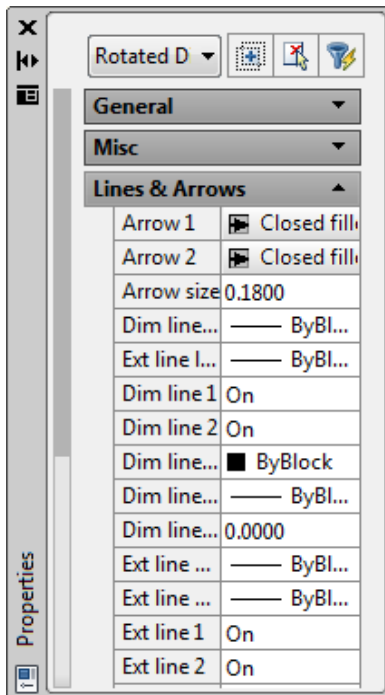
Click on the 0.25 dimension so that it becomes highlighted. A window Quick Properties window may appear; this is a default for AutoCAD 2009. Turn this feature off by deselecting the Quick Properties button at the bottom of the screen. With the 0.25 dimension highlighted, right click on it and select Properties and the Properties window will appear. The variables that represent the 0.25 dimension are listed in the General, Misc, Lines and Arrows, Text, Fit, Primary Units, Alternate Units and Tolerances categories.



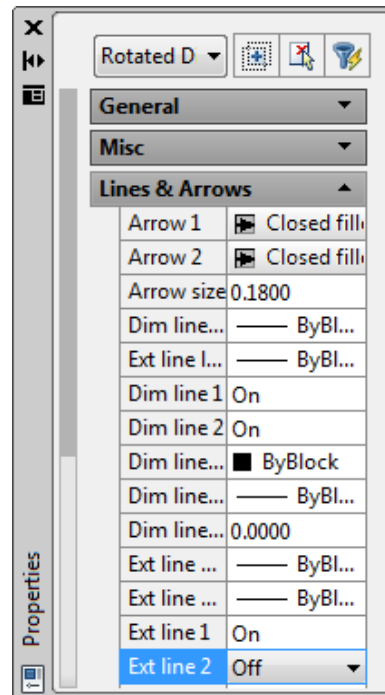


**Figure 3.32 – Using the Properties window to remove a dimension extension line**

To remove the dimension extension line that is covering the object line, select the Lines and Arrows category and double click over the “On” text to the right of either the Ext Line 1 or Ext Line 2 label to turn the extension line off. You may have to try both to see which extension line is in fact the one overlapping the object line.

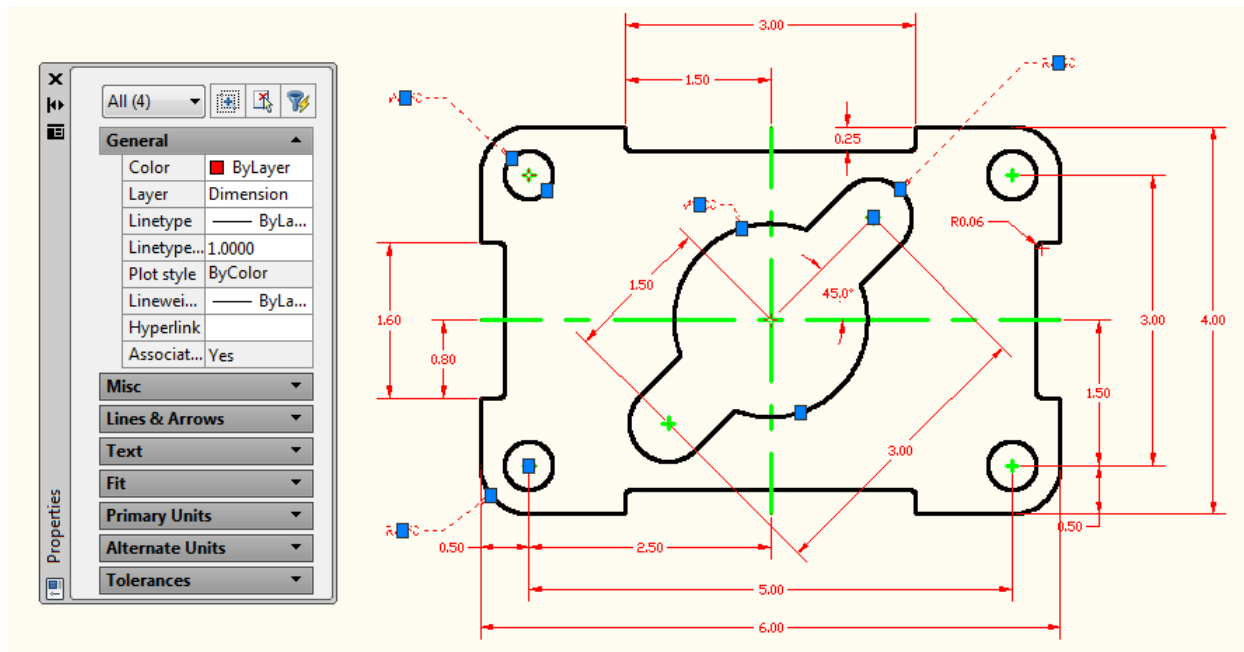


**Figure 3.33 – Selecting Lines and Arrows**



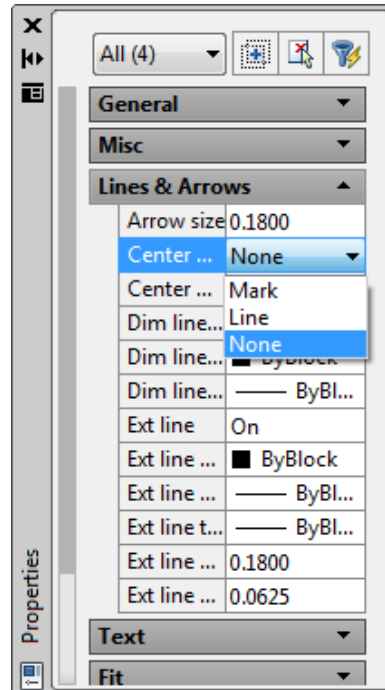
**Figure 3.34 – Turning Ext line 2 off**

Next, to turn off the center marks on the radius and diameter dimensions, zoom out to view the entire Rectangular Problem and select the R0.50, R0.40, Ø 0.50 and Ø 2.00 leaders so that each leader is added to the selection set.



**Figure 3.35 – Turning off the center mark on four dimensions**

Double click on the Center mark label to change “Mark” and the entity will change to “Line”, double click on the label and the entity will change to “None”. You can exit the Properties window by clicking the X in the top left corner of the window when you are finished.



**Figure 3.36 – Turn off the Center marks**



## Using the Edit Text Tool

To add text to the  $\varnothing 0.50$  dimension stating that this description applies to “4 holes”, you will need to use the Edit Text tool. Select the Menu Browser in the top left-hand corner of the screen. Then select Modify, then Object, then Text, and finally Edit. A pick box will appear, allowing you to select the 0.50 diameter dimension. Automatically, the Multiline Text tab will appear, enabling you to edit the dimension label. Move the cursor to the far right side of the 0.50 label and hit ENTER to begin a new line. Type “4 holes” and click anywhere outside the text box to place the new text. Hit ENTER to exit the Edit Text tool.

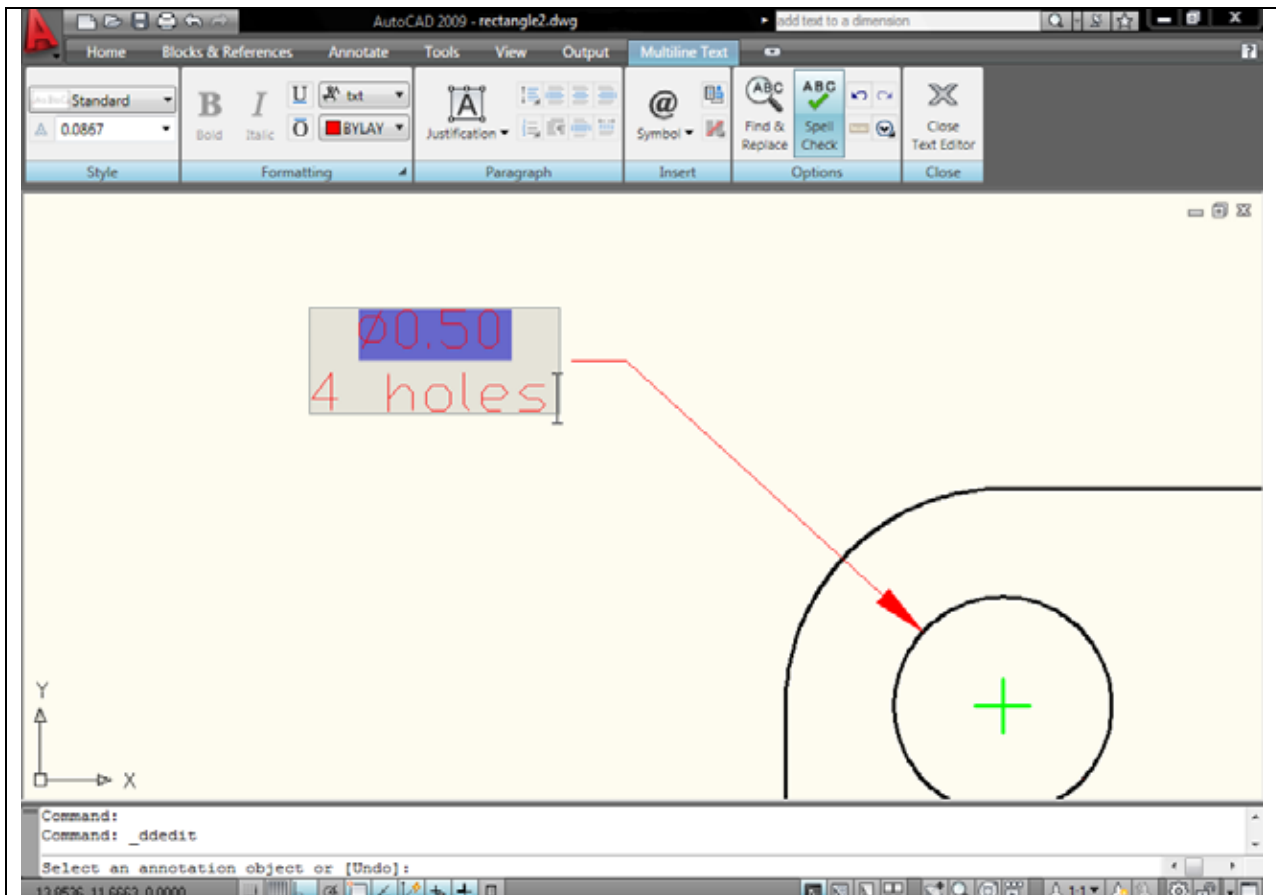
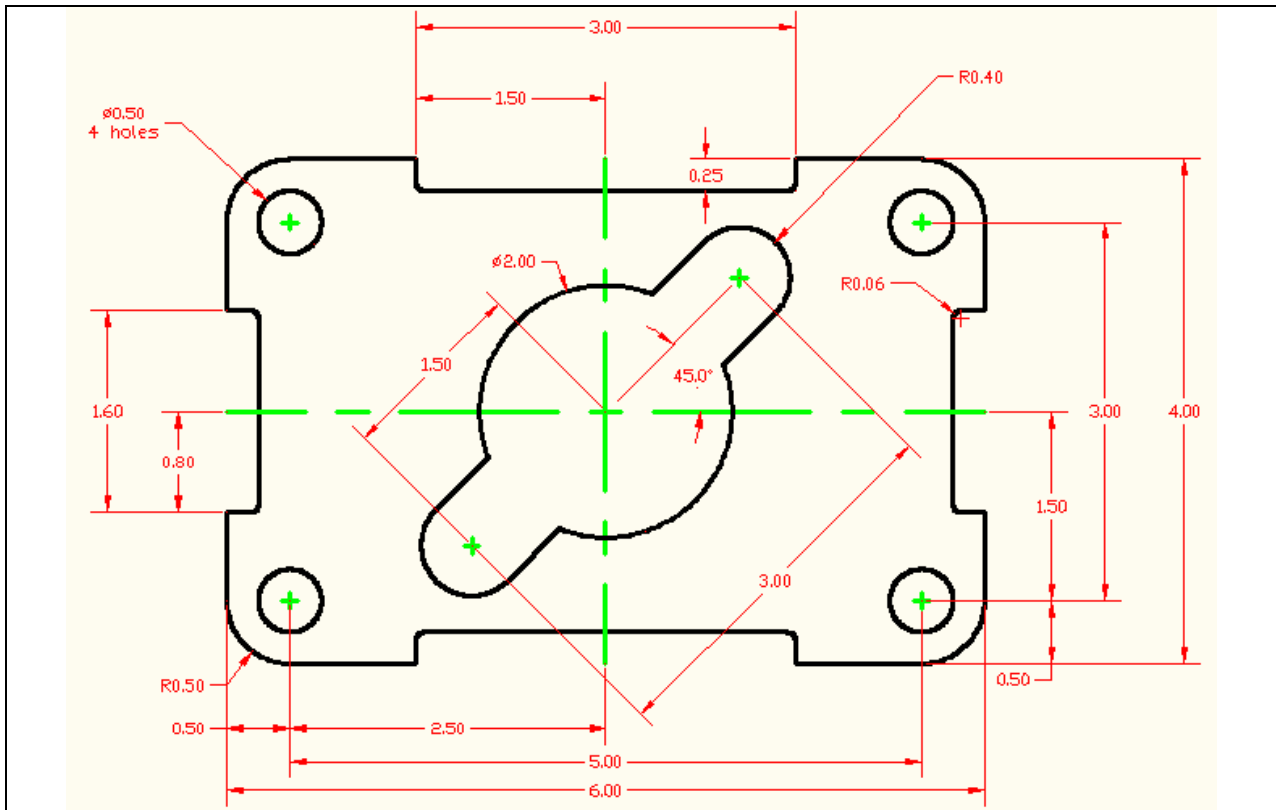


Figure 3.37 – Adding text to a dimension



**Figure 3.38 – Finished dimensioning of the rectangular problem**

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

**\* World Class CAD Challenge 02-08 \* - Close this drawing file. Create a New file and draw the entire rectangular problem on proper layers, using proper dimensions on the drawing. Continue this drill four times, each time completing the drawing under 10 minutes to maintain your World Class ranking. If you are under 10 minutes, you may proceed to the next chapter, the Circular Problem.**