Chapter

3

Dimensioning the Rectangular Problem

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

1. **Creating new layers in an AutoCAD drawing**
2. **Placing Centerlines on the drawing**
3. **Changing entities from one layer to another**
4. **Placing center marks on the drawing**
5. **Basic use of the Dimensioning toolbar**
6. **Using the Linear Dimension command to dimension a part**
7. **Setting system variables using the Dimension Style window**
8. **Using the Aligned Dimension command to dimension a part**
9. **Using the Radius Dimension command to dimension a part**
10. **Using the Diameter Dimension command to dimension a part**
11. **Using the Break Command to shorten a line**
12. **Using the Angular Dimension command to dimension a part**
13. **Using the Properties Tool to Modify a Dimension Variable**
14. **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 in a drawings like looking at an animation of the human body in an encyclopedia where each system of the body is demonstrated on a separate film so that the muscle are on one layer and the circulation system is on another. The layer system in a CAD program allows you to view, revise, or control information that is grouped together. In chapter 2, The Rectangular Problem was drawn on layer 0, the AutoCAD default layer when a new file is started. You can see the current layer listed in the Layer Control list box in the Object Properties toolbar (Figure 3.1). To create new layers using the Layers Properties Manager, select the Layers command button to the left of the Layers Control list box.

Figure 3.1 - The Object Properties toolbar

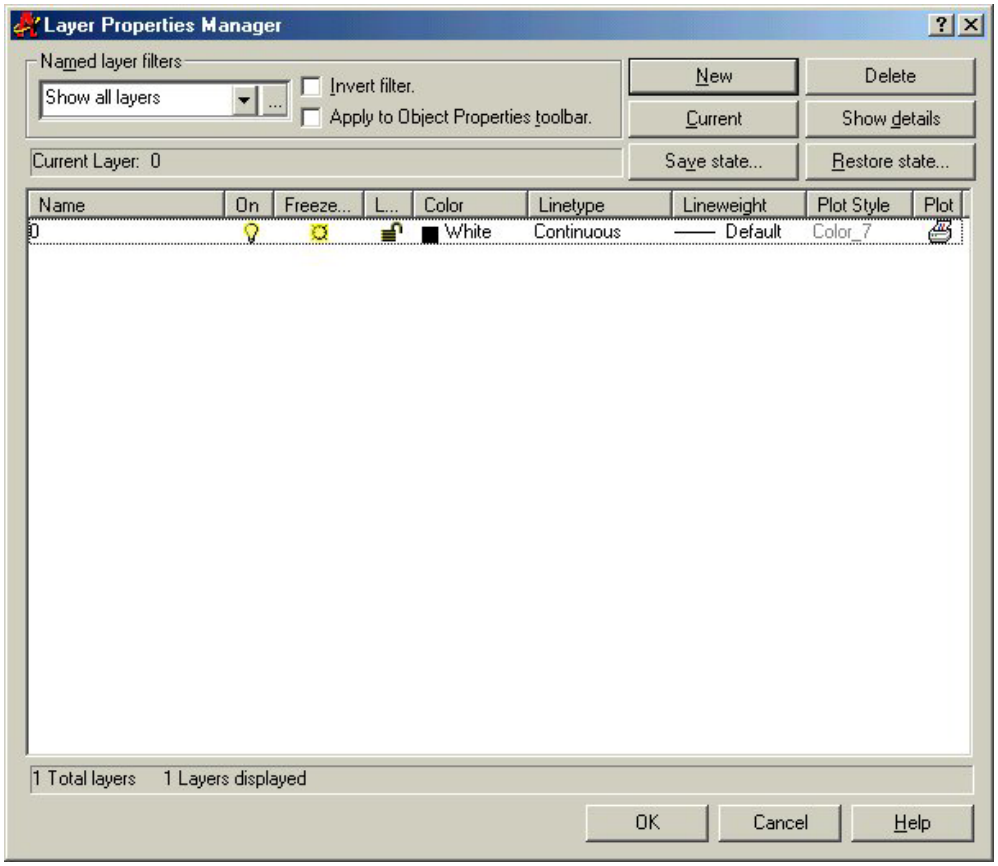


Figure 3.2 - The Layer Properties Manager window

To dimension our 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 line ready for editing under the 0 layer. Immediately, the layer name is ready for editing, so type, "Dimension", and ENTER twice to add another new layer. Type, "Center", and ENTER twice to add another new layer. Type, "Text", and ENTER once to finish adding new layers. Next, you need to add different colors to help distinguish between your layers. To change colors, 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 Standards 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.

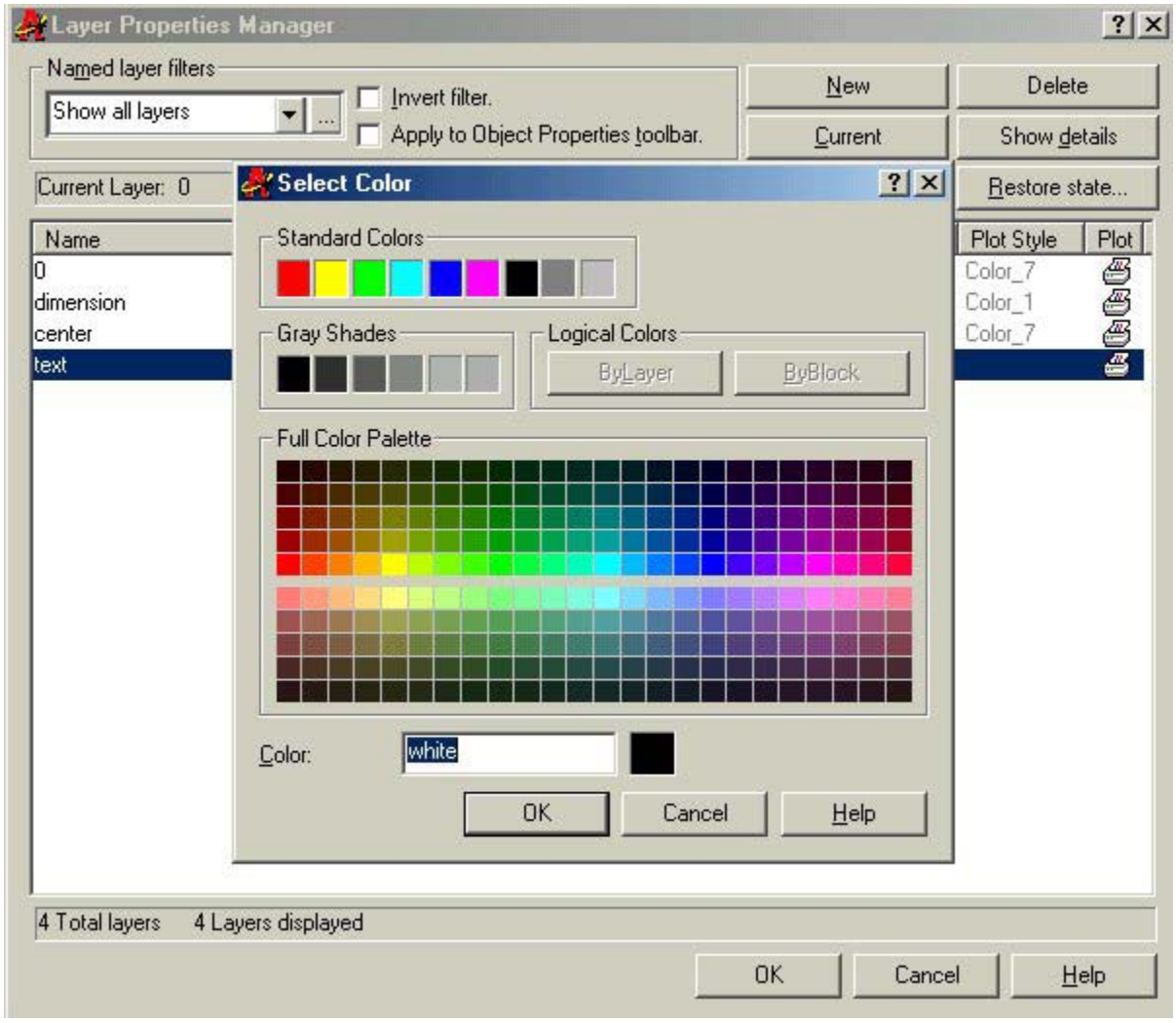
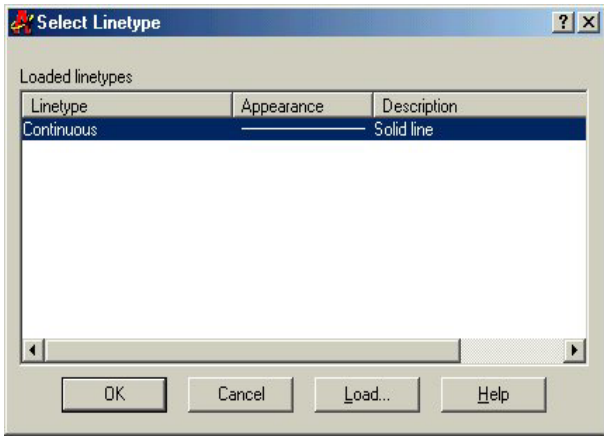


Figure 3.3 - The Select Color window

To finish creating and modifying layers, you need to change the Continuous linetype to a Center linetype. 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 and 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. Now you can set up your centerlines.



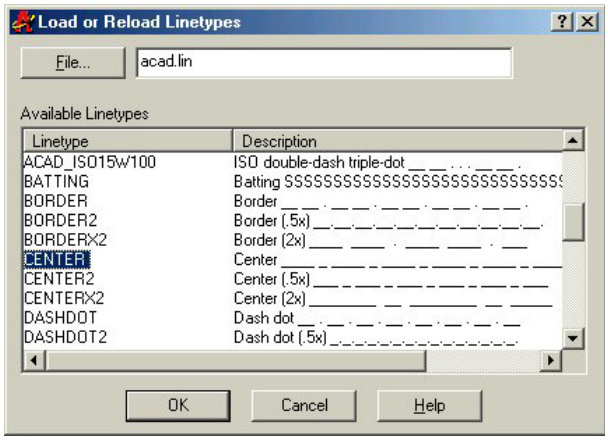


Figure 3.4 - The Select Linetype window

Figure 3.5 - Load or Reload Linetypes

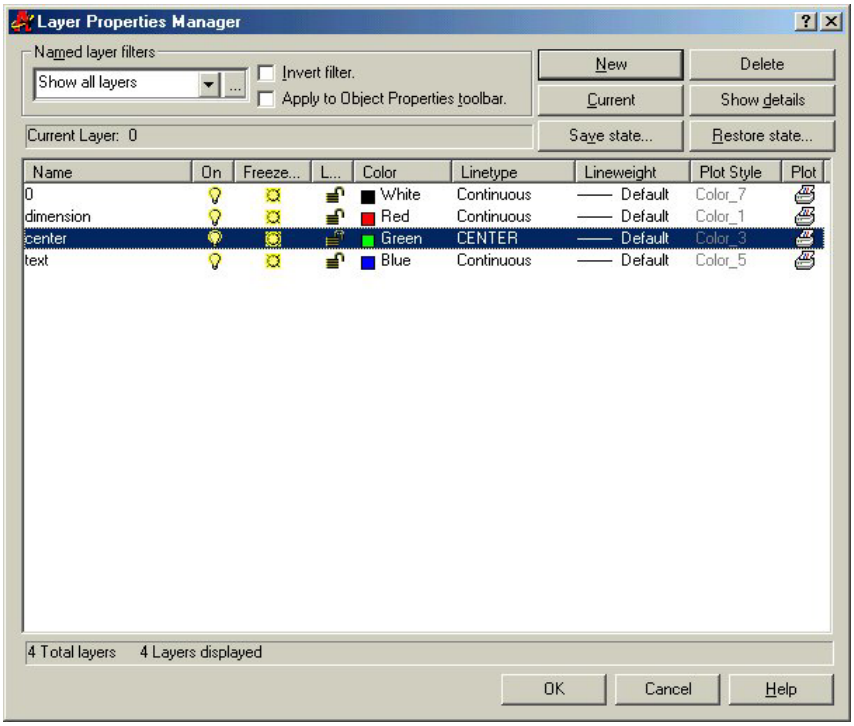


Figure 3.6 - Load or Reload Linetypes

Placing Centerlines on the Drawing

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 entities after selecting a command, but now you will learn to create your selection set initially. Pick the vertical and horizontal centerline, noticing that they have become dotted. Select the black arrow to the right of the Layers Control list box to open the list. Highlight the Center Layer.

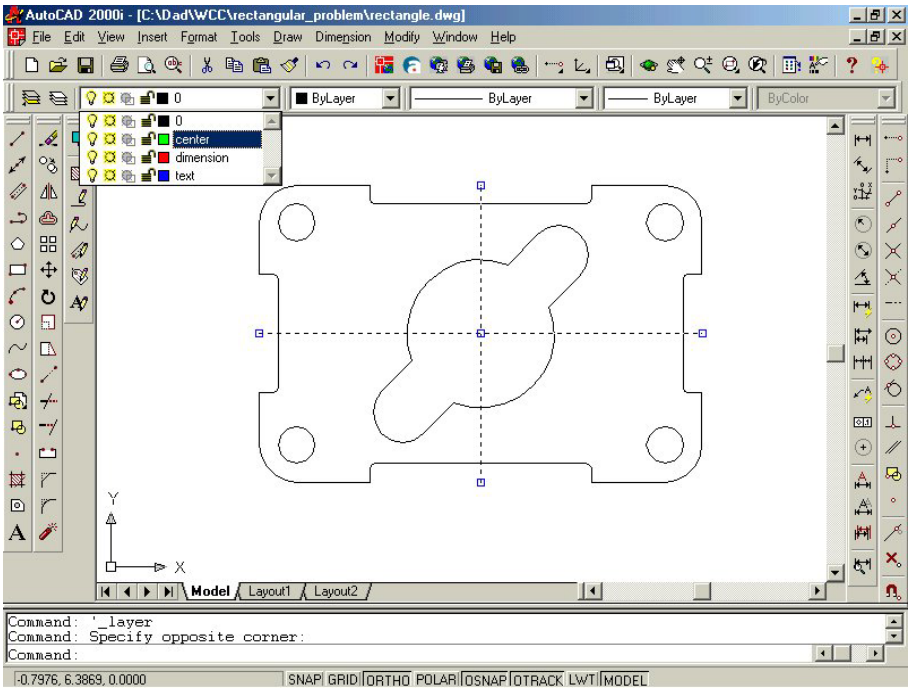


Figure 3.7 – Selecting the Center layer in the Layer Control list box

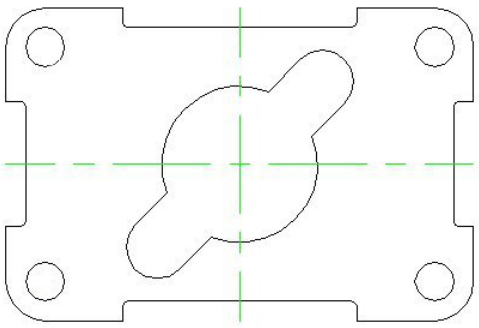


Figure 3.8 – The two centerlines

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. When an individual reads your print, clarity improves the communication process.

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 toolbar. 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 toolbar

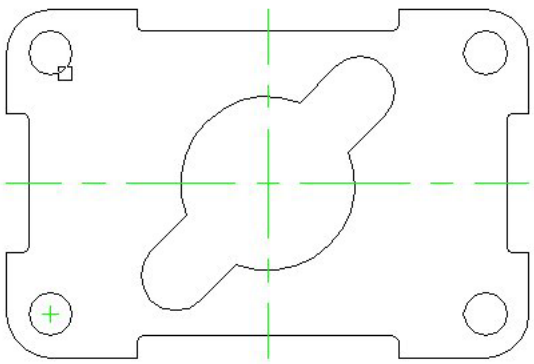


Figure 3.10 – Placing center marks

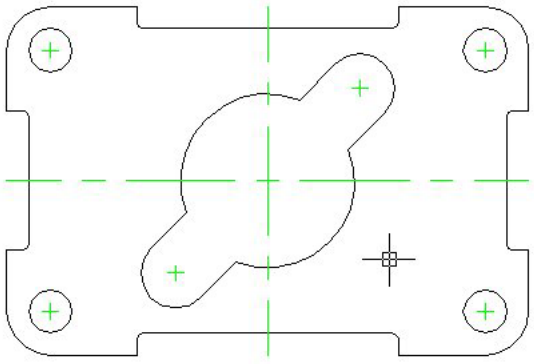


Figure 3.11 – All 6 center marks located

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. Use the reported dimensions to check your work.

The Linear Dimension Command

The first dimension you will put on the part is the 1.5000 from the left corner of the indent to the centerline. Select the Linear Dimension tool on the Dimension toolbar. 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 closed to part but no part of the dimension, dimension lines, and dimension extension lines are on the object lines of the part (Figure 3.12)

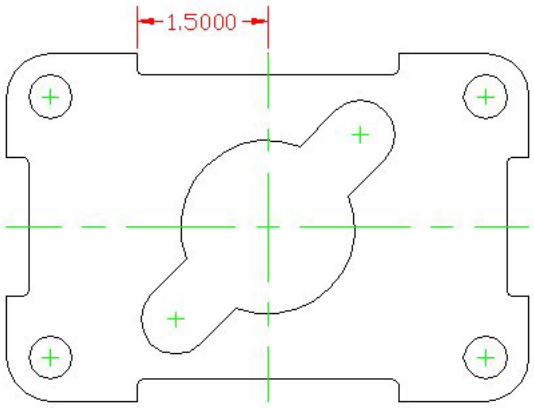
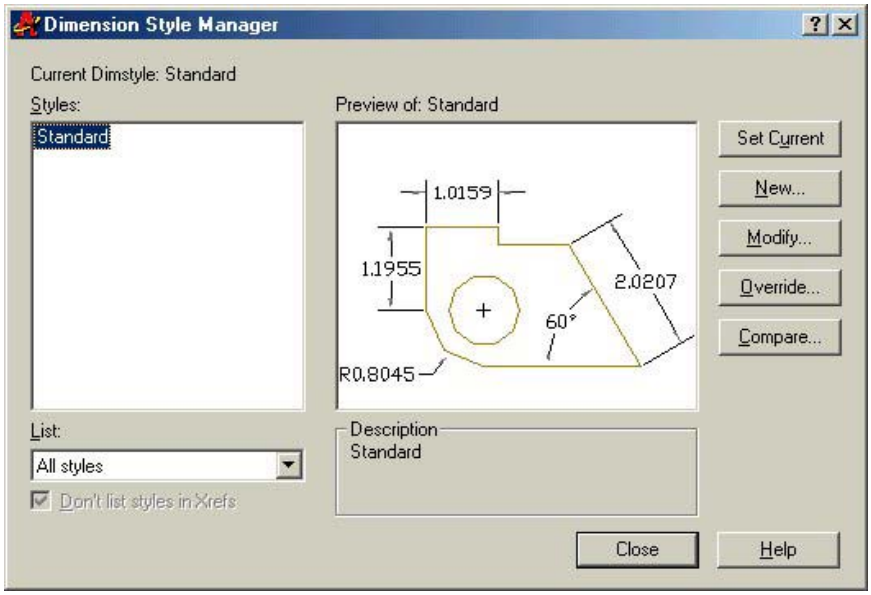


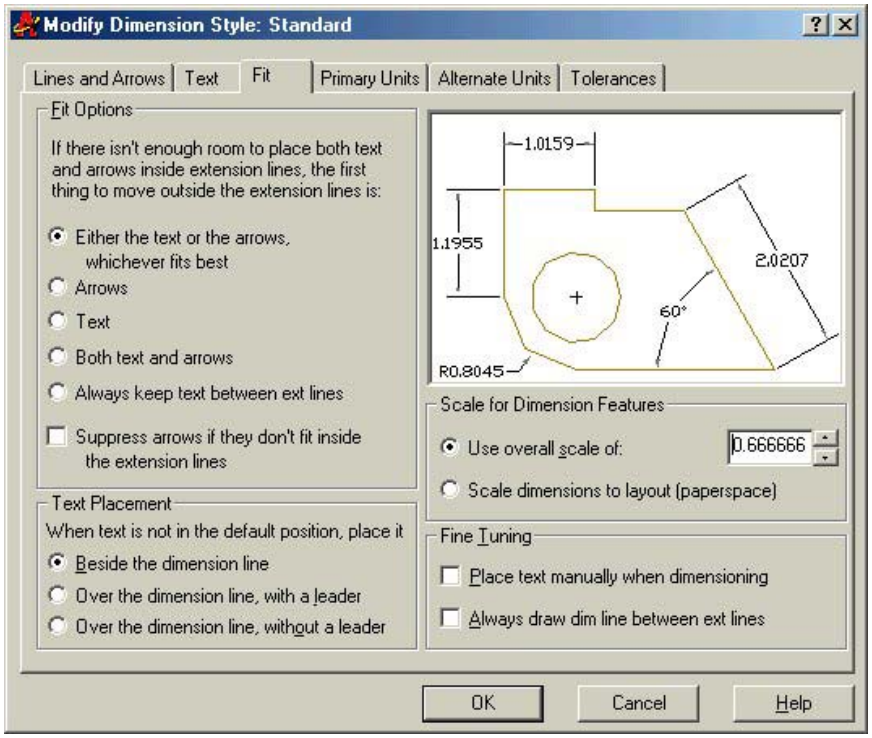
Figure 3.12 – Your first dimension (1.5000)

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 overkill and may cause confusion when 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.



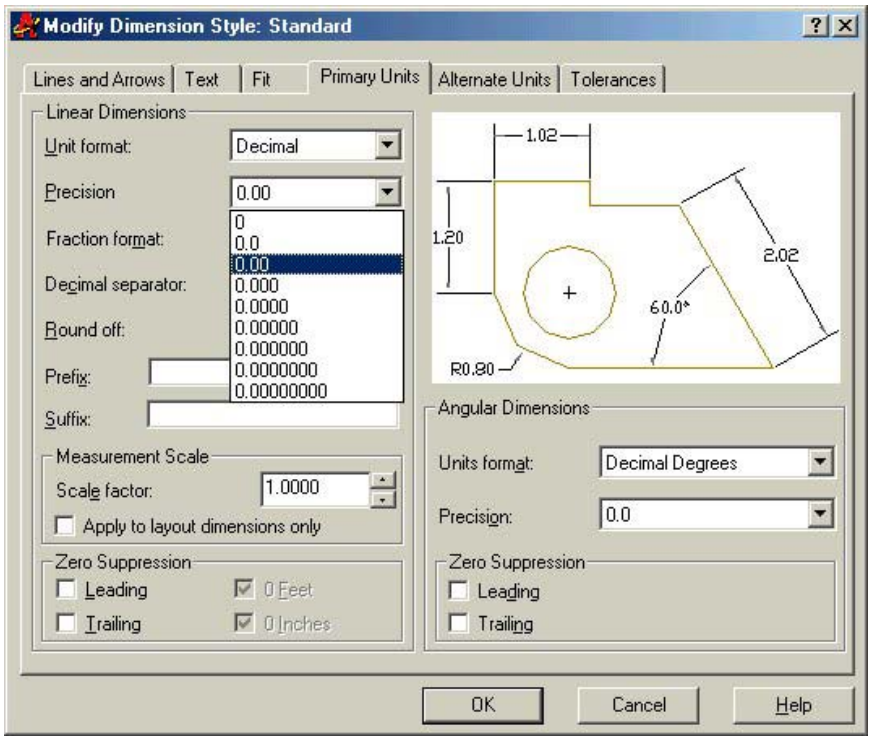
To change the system variables controlling dimensions, select the Dimension Style tool on the Dimension toolbar to open the Dimension Style Manager window. Select the Modify button to access the dimension variables (Figure 3.13).

Figure 3.13 – The Dimension Style Manager window



There are 6 tabs in the Modify Dimension Style window; Lines 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). Scaling the dimensions to 0.694 will change the dimension text height to 0.125. There is only one textbox in this window, so it is easy to locate and make the change.

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 half, select the black triangle to open the list, 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

Now you can see that the 1.50 dimension is to our standard (Figure 3.16), so now you want to place additional dimensions on the part. You do not want to be haphazard concerning placing addition dimensions, but you need to follow a pattern to again improve efficiency and avoid errors of omission like forgetting a dimension.

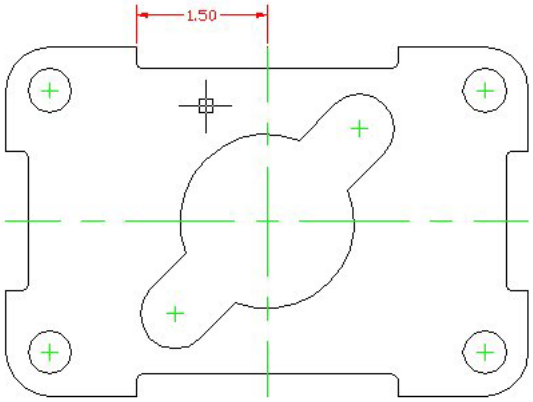


Figure 3.16 – A 2 place dimension

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 is your customer or supervisor removes a detail and 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.

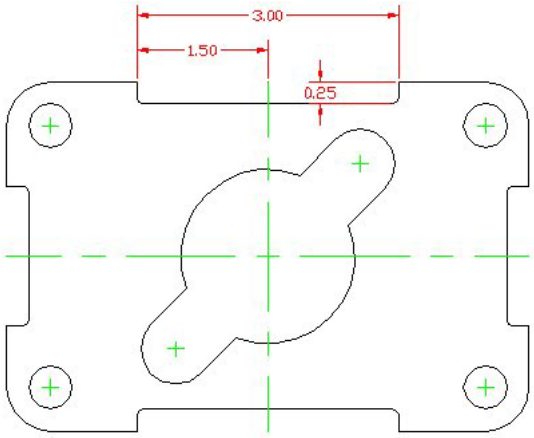


Figure 3.17 – 3 linear dimensions

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 using the Properties tool on the Standard toolbar.

Next, you need to continue to us the Linear

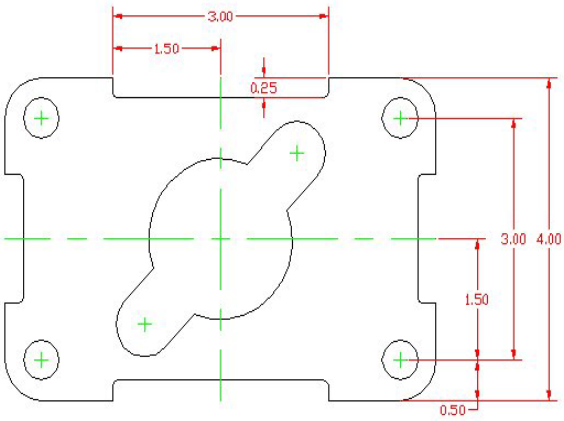


Figure 3.18 – Dimensioning the right

You are following a clockwise pattern placing the dimensions on the drawing, when you continue to the bottom of the part. Dimension extension lines may cross each other, but dimension lines (red lines with arrows) may never cross each other according to ANSI and ISO worldwide drafting standards. 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.

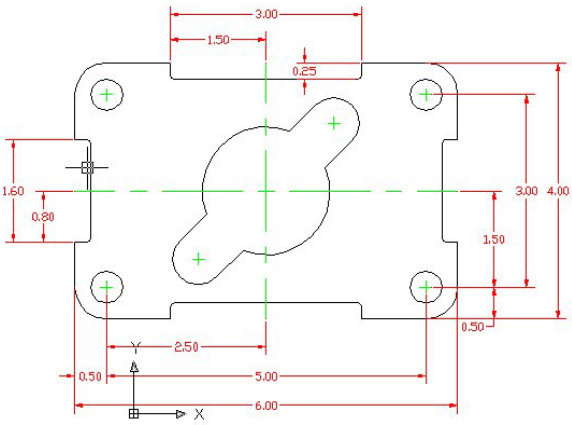


Figure 3.20 – Dimensioning the left

Dimension command by pressing ENTER each time to repeat the command. Always dimension from the inside to the outside of the part, so that you do not get into a position where there is no longer any room for a dimension. Do the 1.50, the 3.00 and 4.00 dimensions trying to get the spacing to look even (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. See how the dimensions line up.

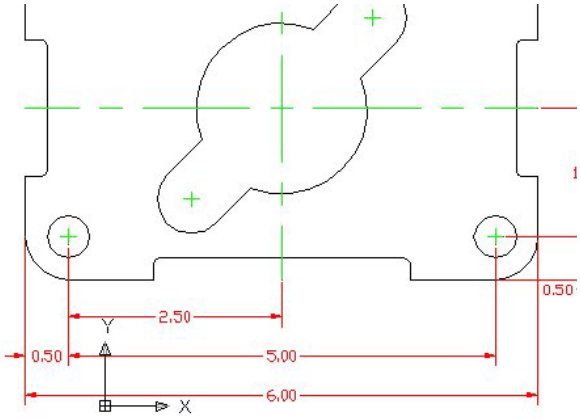


Figure 3.19 – Dimensioning the bottom

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. Now, we are ready to place the two Aligned Dimensions.

The Aligned Dimension Command

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.

Select the Object Snap Settings tool on the Object Snap toolbar to open the Drafting 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.

Place the 1.50 aligned-dimension on the drawing 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.

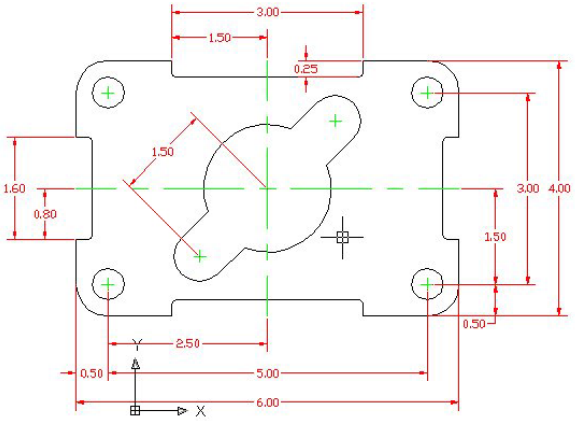


Figure 3.21 – 1st aligned dimension

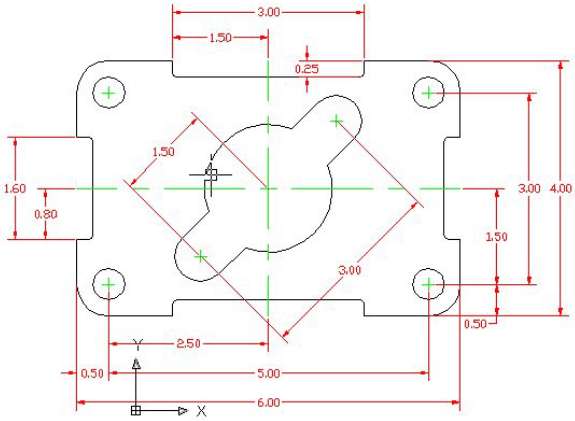
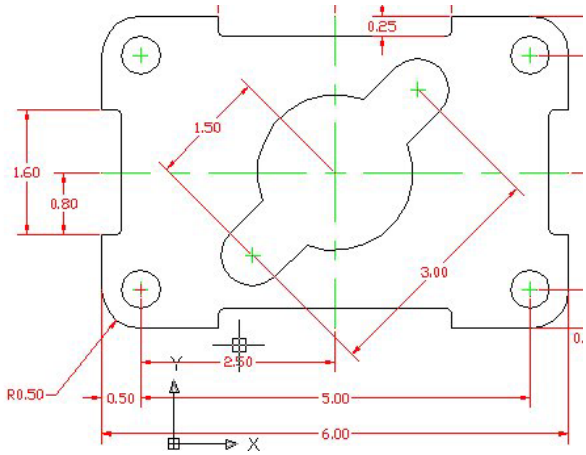


Figure 3.22 – 2nd Aligned dimension

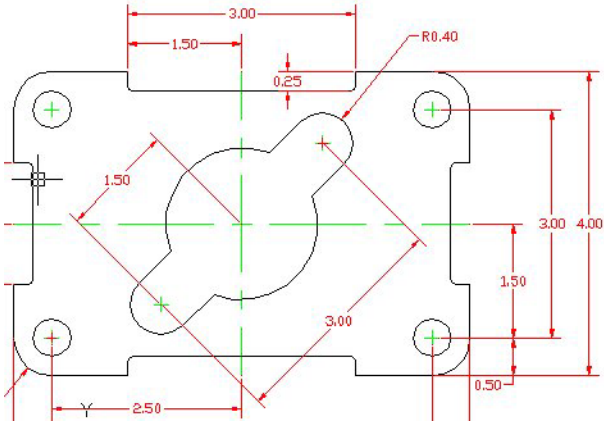
Using the Radius Dimension Tool

The Radius Dimension tool is as easy to use as the Center Mark tool. Once selecting the Radius Dimension tool on the Dimension toolbar, 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.



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).

Figure 3.23 – The first radius dimension



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

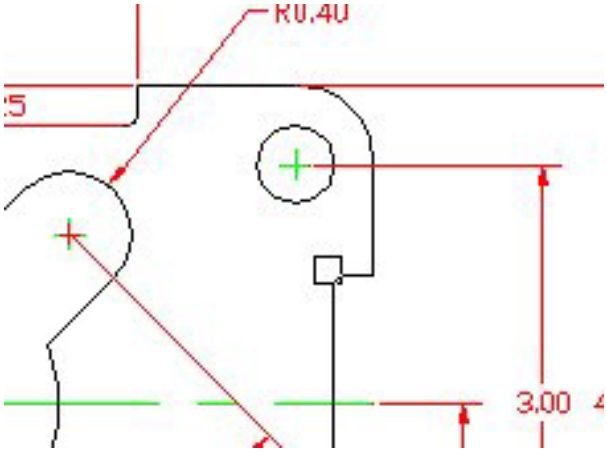


Figure 3.25 – Selecting the last radius

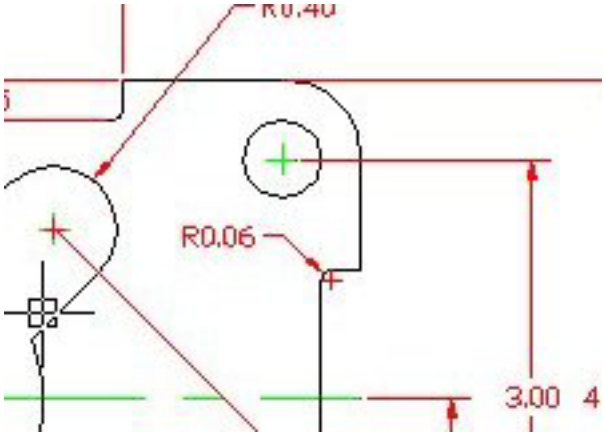
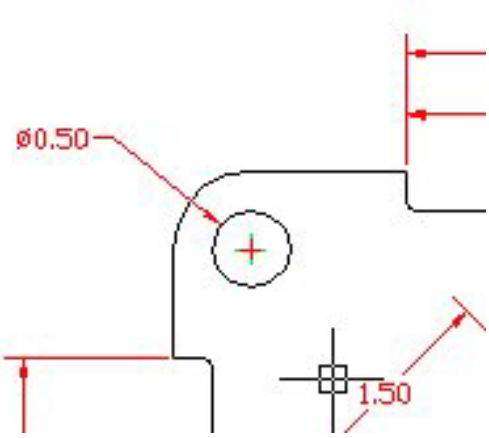


Figure 3.26 – The 0.06 radius

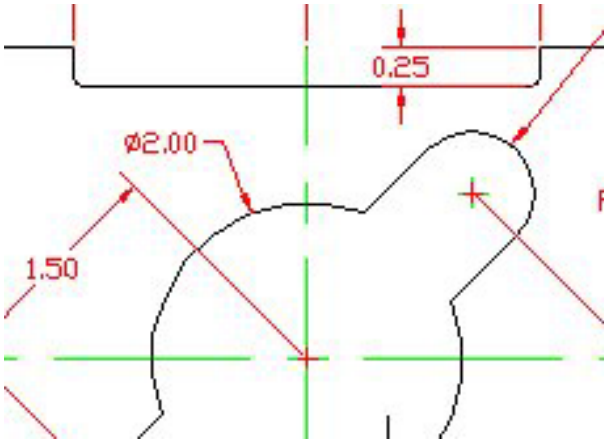
Using the Diameter Dimension Tool

You need to place two dimensions showing diameters on the drawing. The Diameter Dimension tool is located next to the Radius Dimension tool on the Dimension toolbar.



Select the Diameter Dimension tool on the Dimension toolbar and the aperture cursor will change to a pick box to select an arc or circle. 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 to 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, since 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. A dimension extension line cannot touch an object or centerline per ANSI or ISO

standards, 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 toolbar. 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).

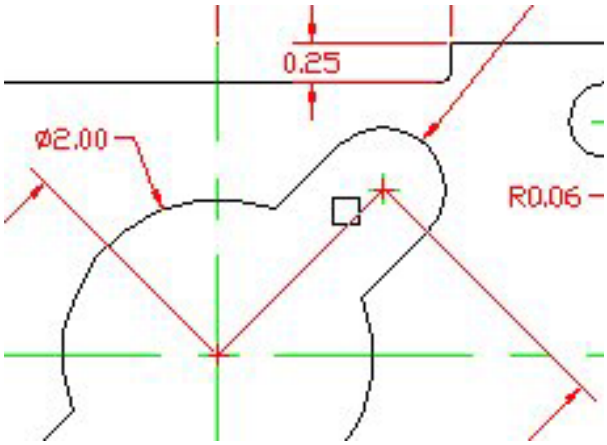


Figure 3.29 – Adding a line segment

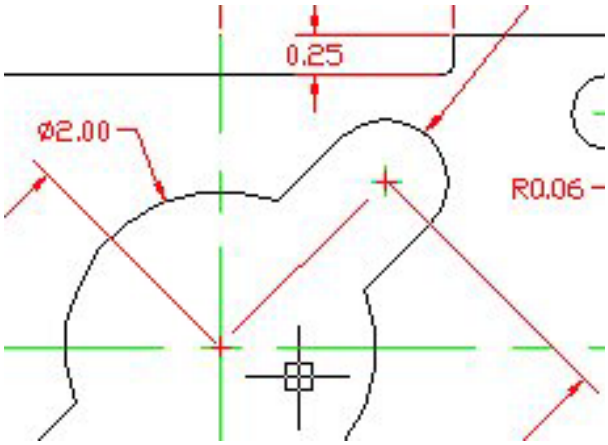


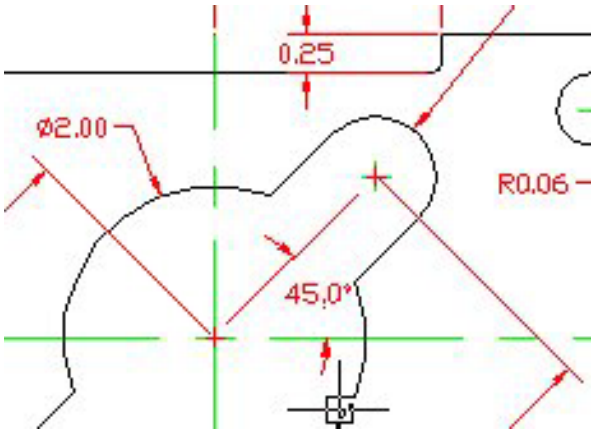
Figure 3.30 – Breaking the line segment

Select the Break command on the Modify toolbar. Now, the command line will prompt you to "select object", but it is more complicated than just picking the line, since the point that you select will be the first cutting point in the break sequence. Pick close to the end of the line segment as shown in Figure 3.29. Next the command line will prompt you to "specify second break point or [First point]", where you need to pick above and to the right of the red extension line. If you do not pick clearly to the right and above the first selection point, you may end up with a small line segment that will need to be erased using the Erase command on the Modify toolbar. Repeat the Break command by pressing ENTER and remove part of the line segment as shown in Figure 3.30.

Using the Angular Dimension Tool

Another easy command to use and the last dimension tool in this problem is the Angular Dimension tool that is located on the Dimension toolbar. Select the Angular Dimension tool and you need to select two converging lines (non-parallel) to place the one place decimal on the drawing. For the first pick, select the red dimension extension line just drawn and for the second pick, 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 and will add to the clarity of the information.



Three red centerlines need to be removed from the drawing. A red dimension extension line needs to be removed from covering the object line on the 0.25 dimension. You need to add "4 places" under the 0.50 diameter dimension text.

Figure 3.31 – An angular dimension

Using the Properties Tool to Modify a Dimension Variable

Select the Properties tool on the Standard toolbar and select the 0.25 dimension. 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.

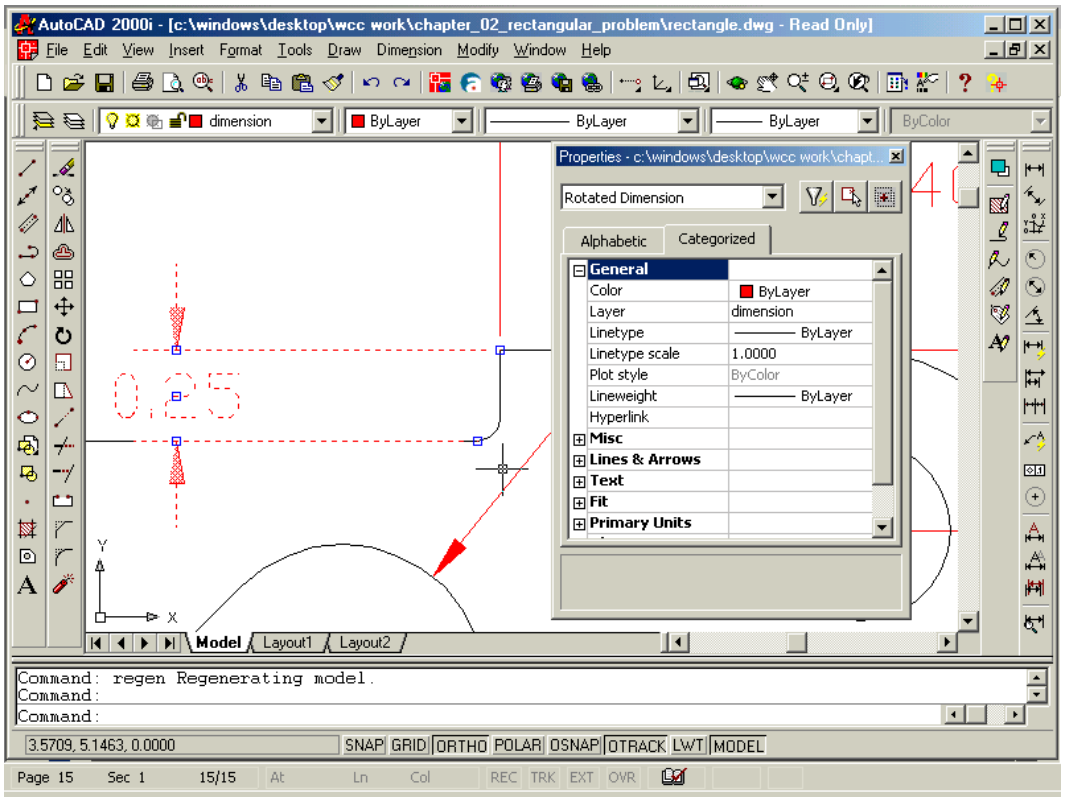
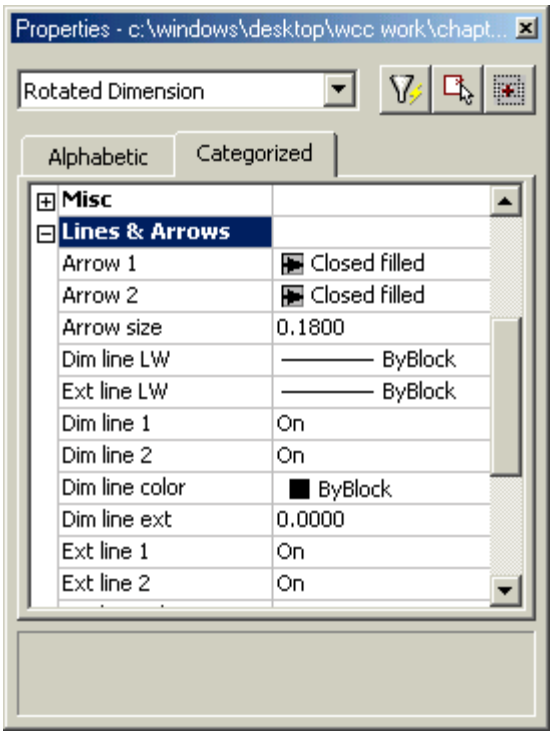


Figure 3.32 – Using the Properties window to remove a dimension extension line

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 the Ext Line 1 label to turn the extension line off.



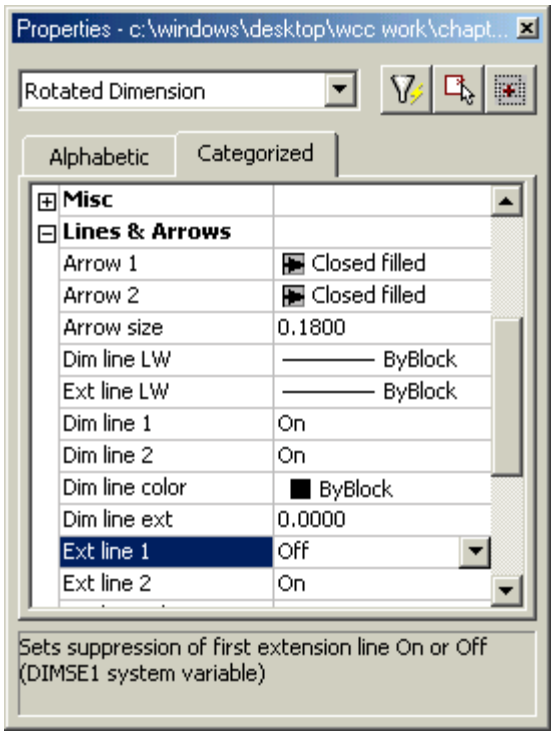


Figure 3.33 – Selecting Lines and Arrows **Figure 3.34 – Turning Ext line 1 off**

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 holding down the CTRL key adding each leader to the selection set.

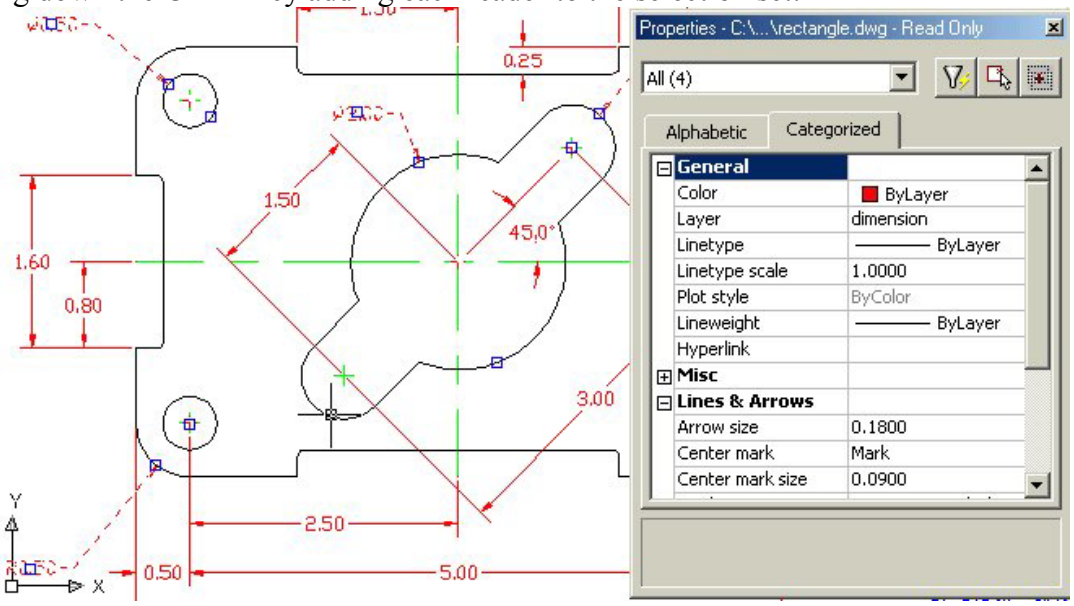


Figure 3.35 – Turning off the Center mark on four dimensions

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”.

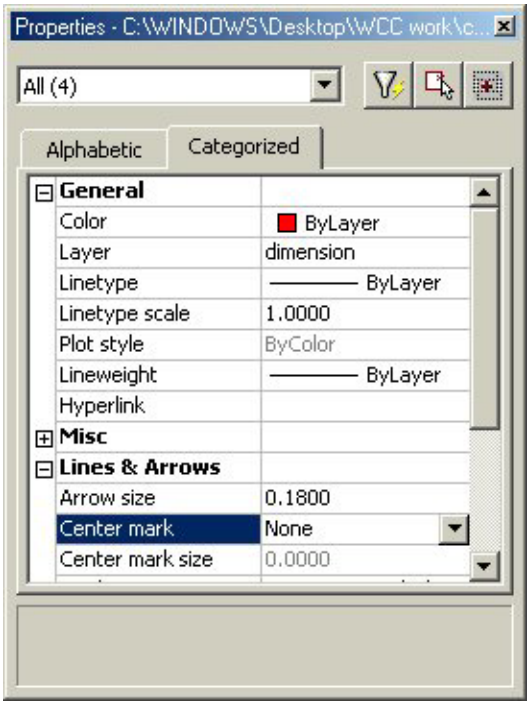


Figure 3.36 – Turn off the Center marks

Using the Edit Text Tool

To add text to the Ø 0.50 dimension stating that this description applies to “4 holes”, select the Edit Text tool on the Modify II toolbar. The Multiline Text Editor window will appear in the graphical screen. The “<>” symbol represent the auto dimension, so place the cursor behind the “<>” symbol, and ENTER. On the line below the auto dimension symbol, type “4 holes” and select “OK” to close the window.

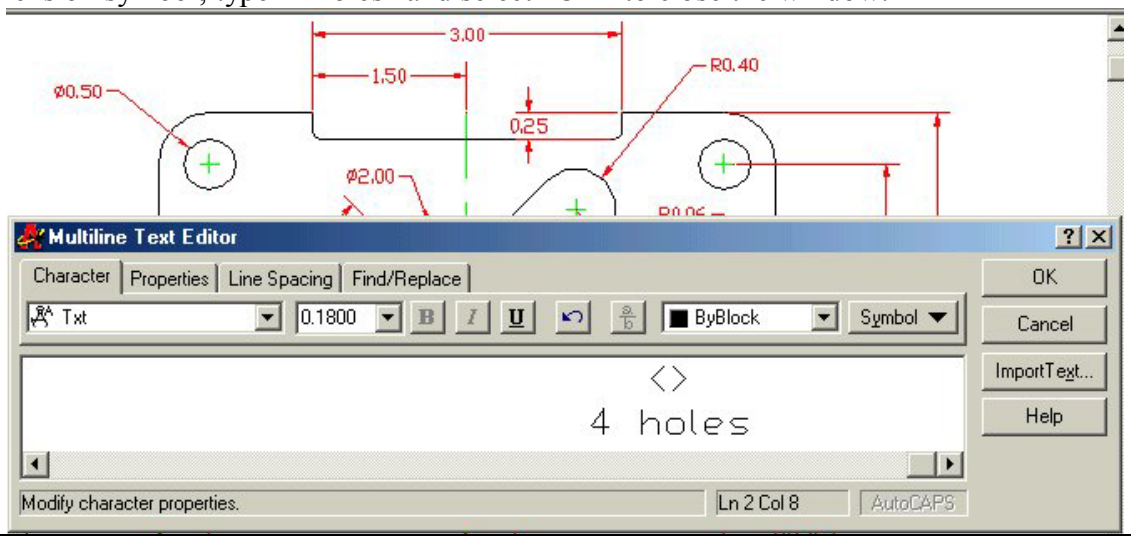


Figure 3.37 – Adding text to a dimension

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

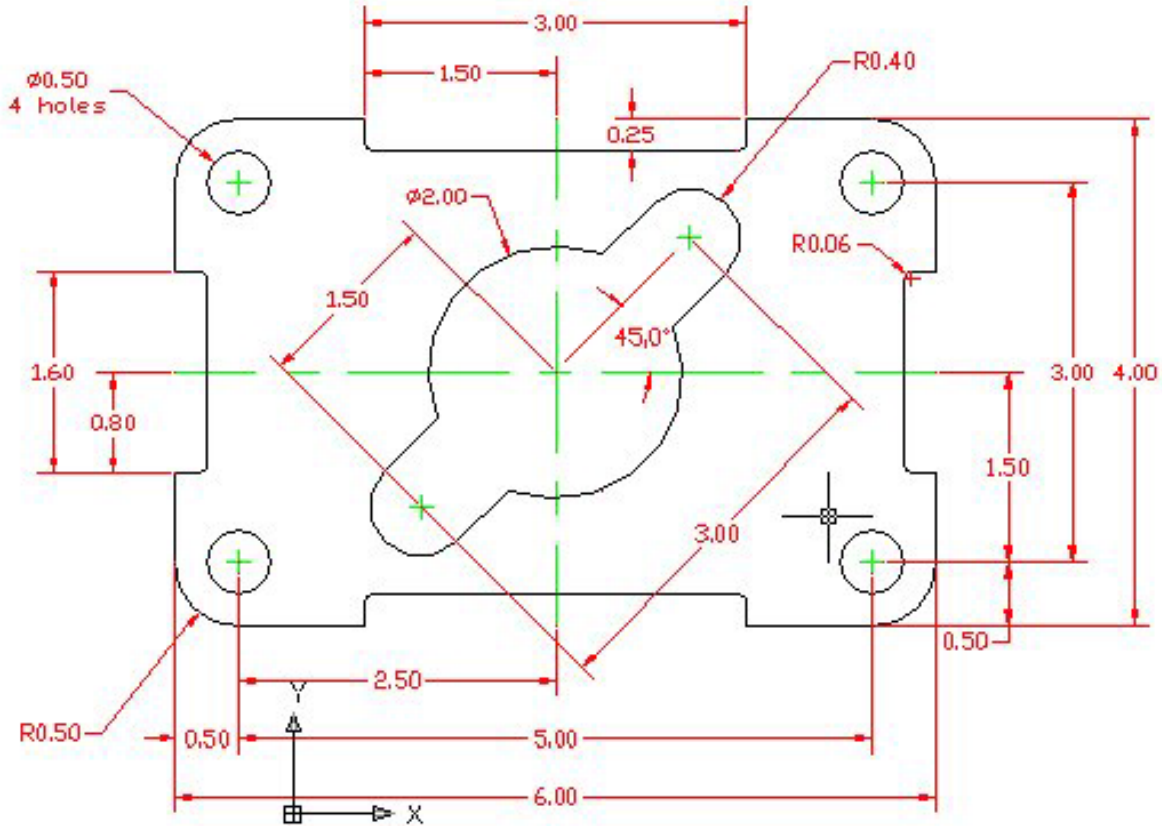


Figure 3.38 – Finished dimensioning the rectangular problem

*** World Class CAD Challenge * - 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.**