

Dimensioning the Rectangular Problem

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

- 1. Creating new layers in a progeCAD 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 Settings 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 Dimension Settings to Modify Single Dimensions**
- 14. Using the Edit Text Tool**

Creating New Layers in an progeCAD 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 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 layer 0, the progeCAD default layer set when a new file is started. You can see the current layer listed in the Layer Control list box in the Layers section of the Entity 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 - Layers section of the Entity Properties toolbar

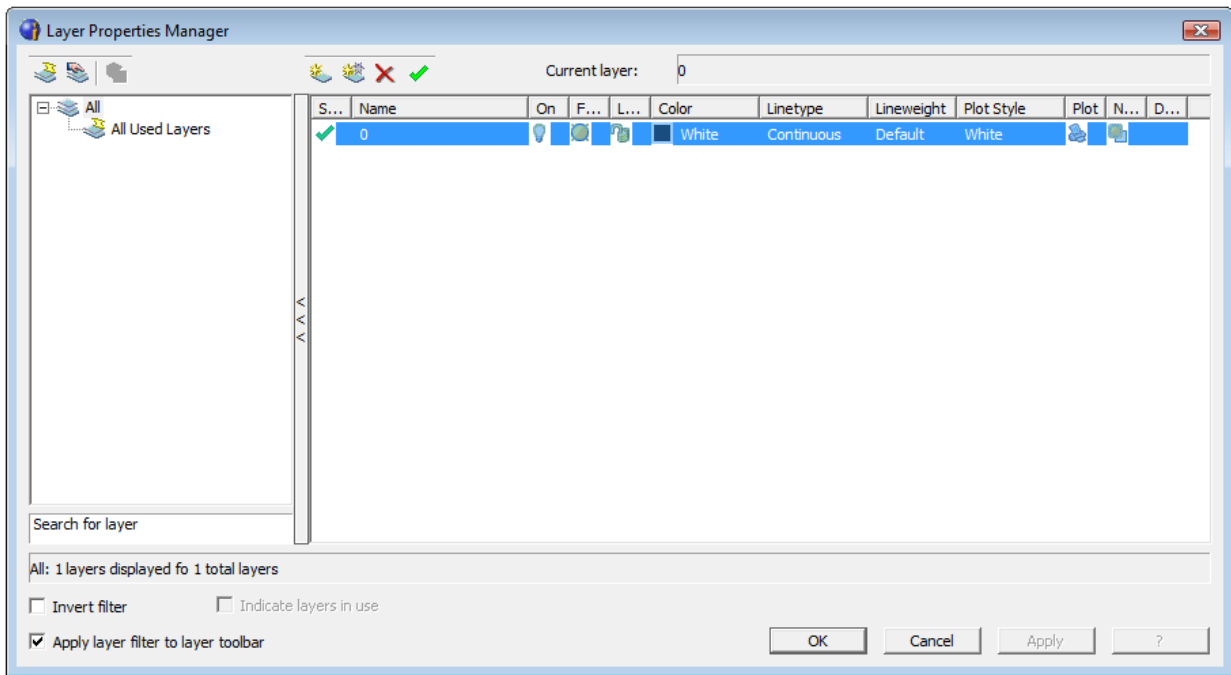


Figure 3.2 - The Layer Properties Manager window

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. Select the New button again, type “Center,” and hit ENTER again. Add one more layer, typing “Text” and ENTER 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 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.

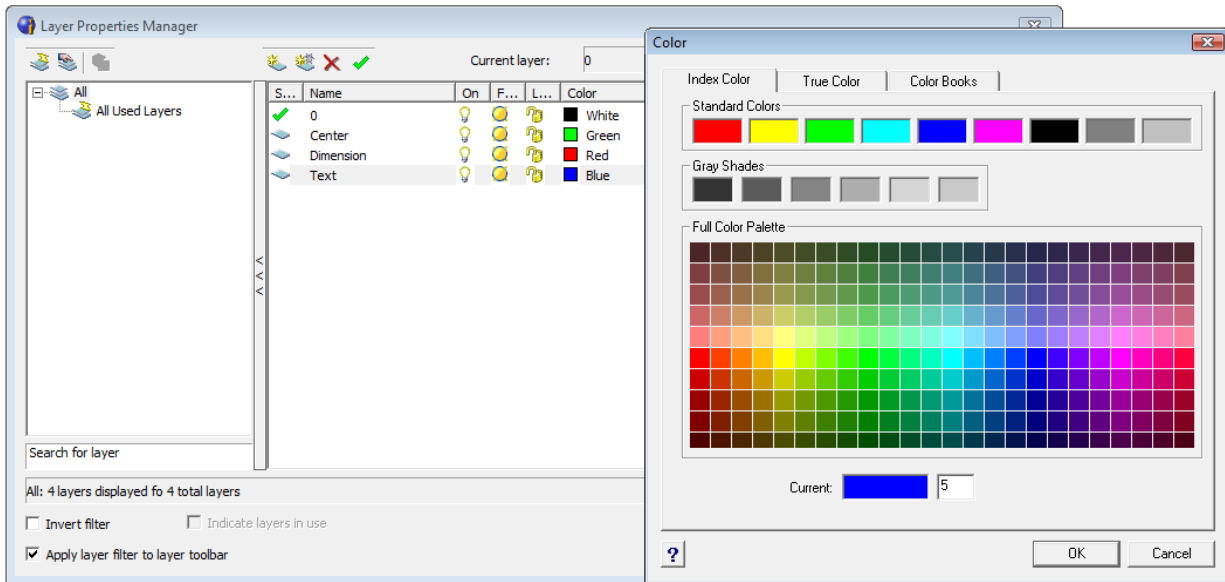


Figure 3.3 - The Select Color window

To finish creating and modifying layers, you need to change the Continuous linetype to a Center linetype 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). Scroll down, highlight the CENTER2 linetype as shown in Figure 5.5, and select the OK button. You will now see that the Center layer has a CENTER2 linetype (Figure 3.5).

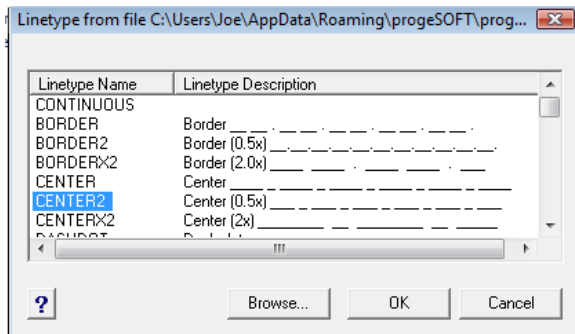


Figure 3.4 - The Select Linetype window

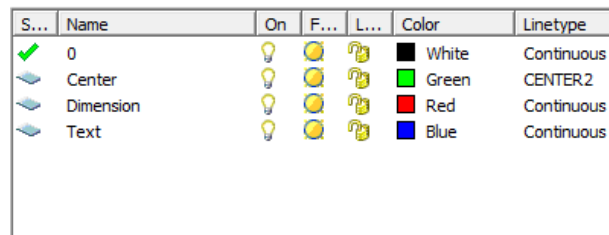


Figure 3.5 - Load or Reload Linetypes

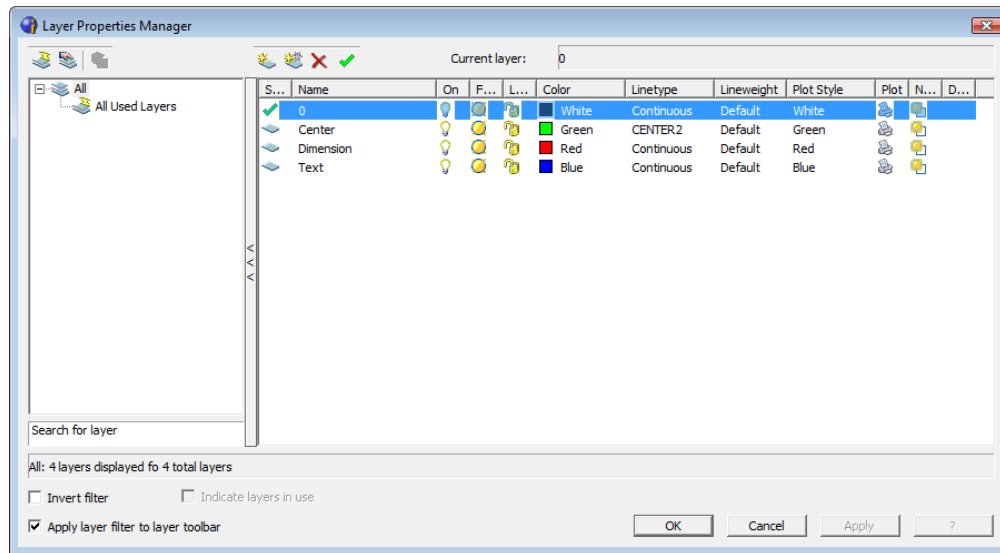


Figure 3.6 – Four completed layers

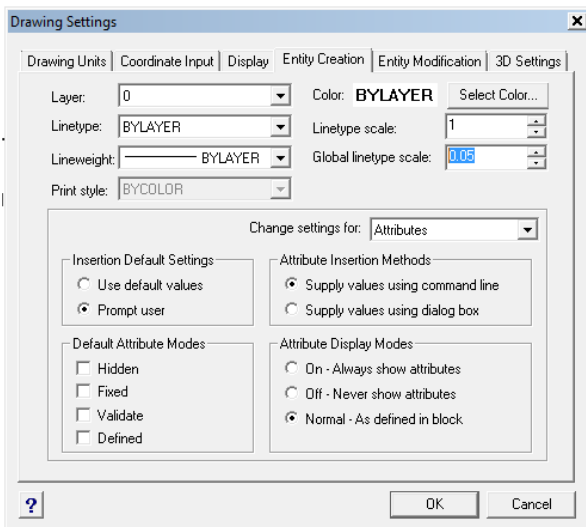


Figure 3.7 – Changing Linetype Scale

Finally, before you begin to continue drawing, you need to make a change to the Drawing Settings. Select Drawing Settings from the tools menu on the menu bar and the Drawing Settings window will appear. Change the Global Linetype Scale to 0.05 (Figure 3.7). This will make the dashes in the Center linetype appear at closer intervals so that you will be able to see the distinction on your drawing. If the Global linetype scale is left at 1, your centerlines will appear as though they are under the continuous linetype.

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 initially. 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.

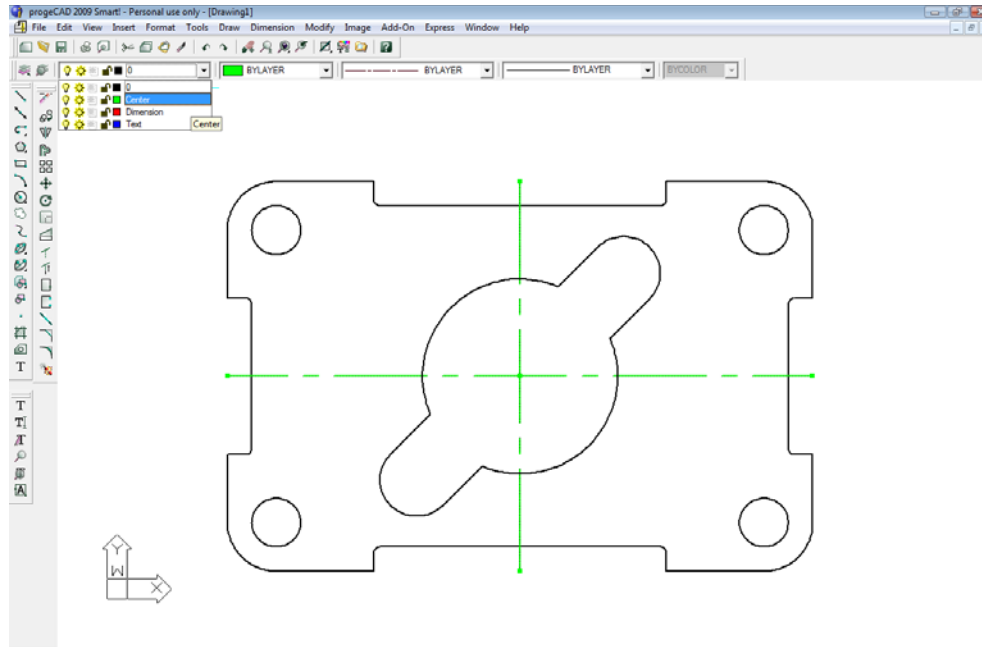
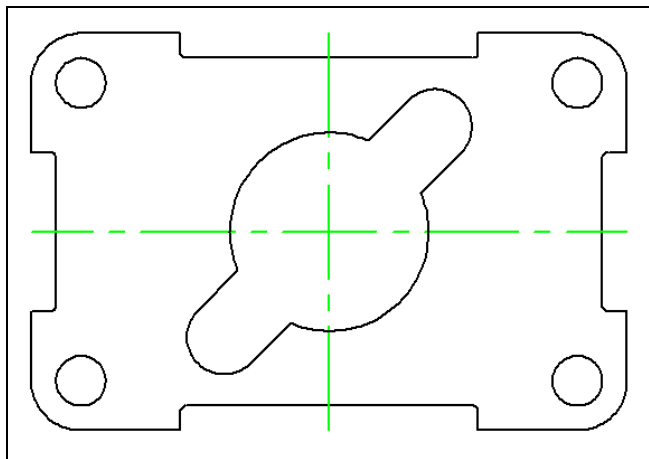


Figure 3.8 – Selecting the Center layer in the Layer Control list box



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 will be removed from the selection set. Centerlines are critical in the dimensioning process because they show symmetry in a part and reduce the amount of dimensions on the drawing. When reading prints, clarity improves the communication process.

Figure 3.9 – 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 entity 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 now appear 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 3.11). A green center mark will appear 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 3.12).

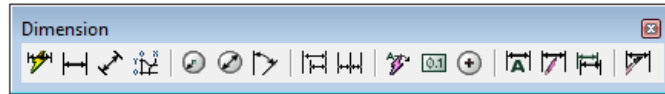


Figure 3.10 – The Dimension toolbar

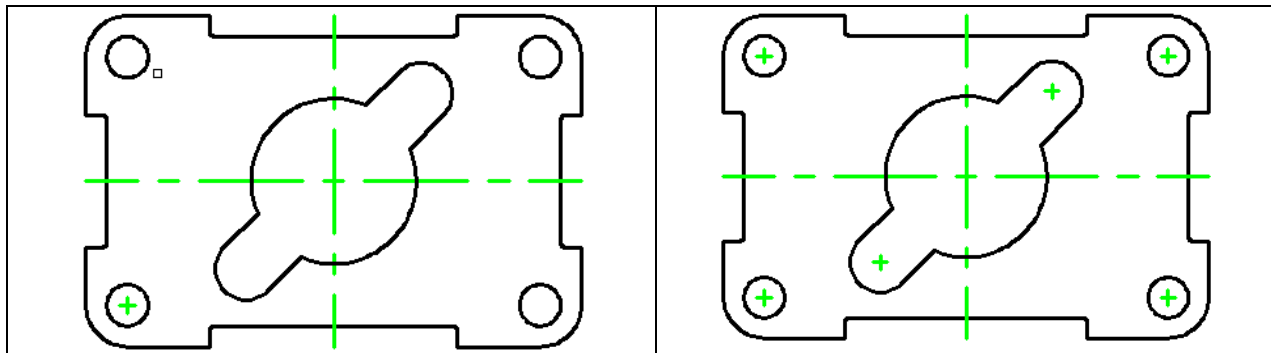


Figure 3.11 – Placing center marks

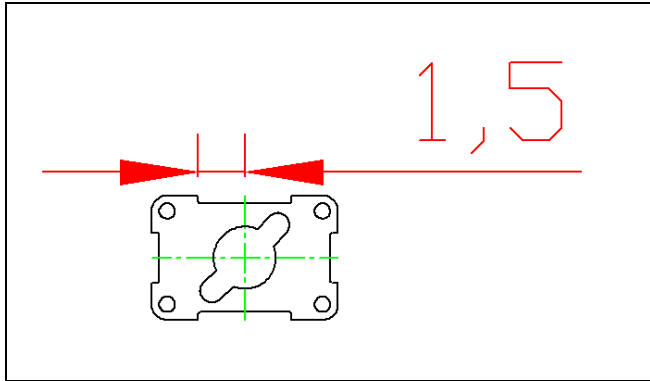
Figure 3.12 – All 6 center marks located

Did you notice how simple the dimensioning tools work? 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

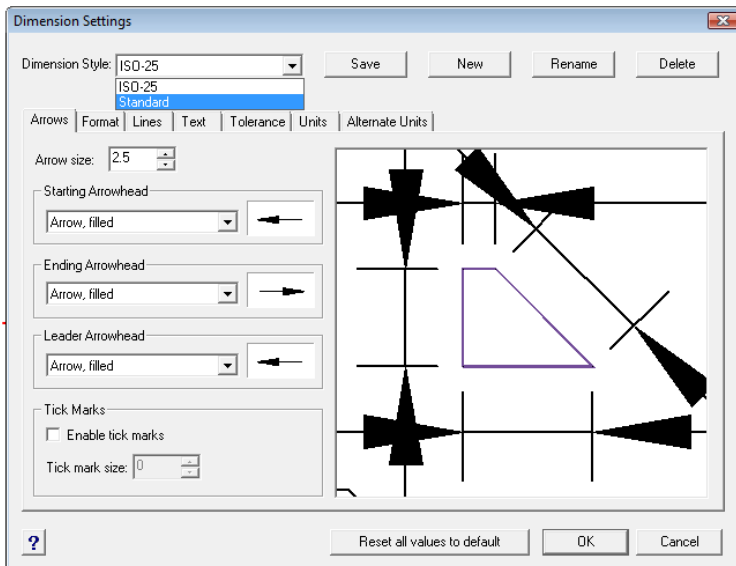
The first dimension you will place is 1.5000, from the left corner of the top indent to the centerline. Select the Linear Dimension tool on the Dimension toolbar. The command line will prompt you with “ENTER to select entity/<Origin of first extension line>:”, so select the endpoint at the top of the indent. The command line will immediately ask you to select the “Origin of second extension line:”, so select the endpoint at the top of the vertical centerline. The next prompt on the command line will be to “Angle/Text/Orientation of dimension line: Horizontal/Vertical/Rotated:”, so 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 3.13). Your dimensions may be way too big for the drawing. We’ll fix that next.





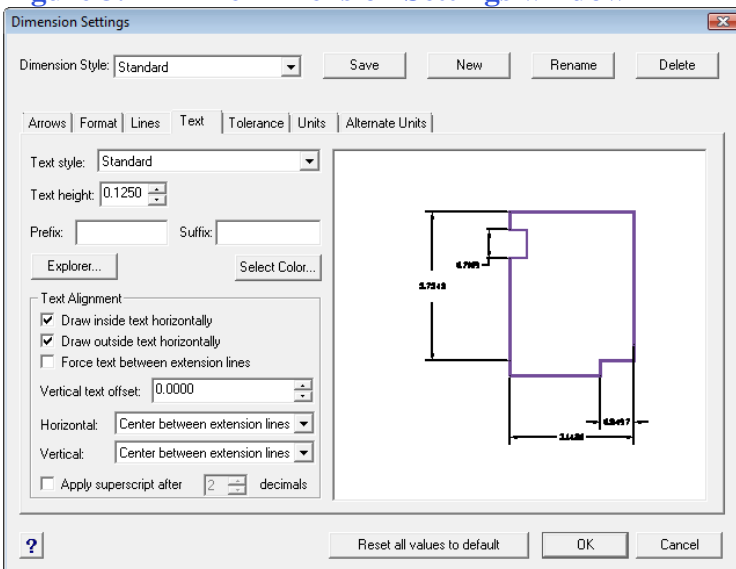
The 1,5 dimension is obviously formatted incorrectly. We will correct this by selecting the Dimension Style tool on the Dimension toolbar. The Dimension Settings window will appear, allowing us to modify the way dimensions are displayed. A more comprehensive overview of the Dimension Settings window will follow in another chapter. For now we'll just make the changes we need for this particular problem.

Figure 3.13 – Your first dimension



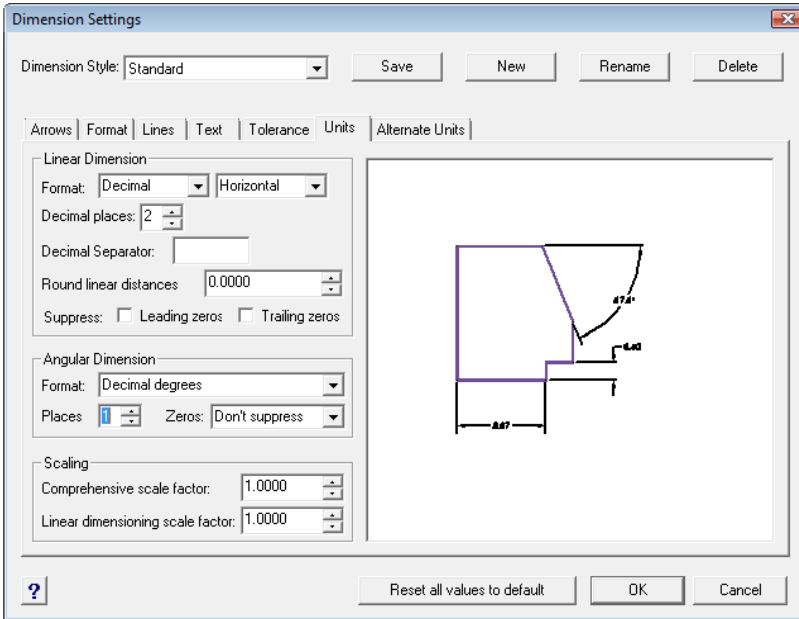
When the Dimension Settings window appears, the first thing you'll want to do is change the Dimension Style in the dropdown menu in the top left corner of the window to Standard if it isn't already (Figure 3.14). The ISO-25 dimension setting will not serve us well, as you can see Figure 3.13.

Figure 3.14 – The Dimension Settings window



The first change we'll make to the Standard dimension style is to make the text height 0.125. Select the Text tab and change the Text height value to 0.125 as shown in Figure 3.15. You can hit Save at the top of the window at any time to store the changes you've made in the selected dimension style.

Figure 3.15 – The Dimension Settings window – Text tab

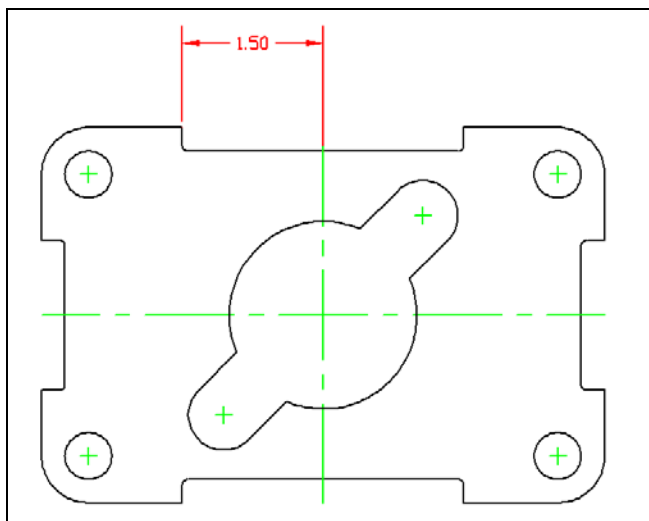


On a sheet metal part that is going to be held to (plus or minus) 0.010-inch tolerance, a four place decimal is unnecessarily accurate and may cause confusion because some companies use decimal places to convey tight tolerances. On this drawing, you need to change the dimension to a 2-place decimal. Under the Units tab, change Decimal places to 2. You can also change the Places value under Angular Dimensions to 1 (Figure 3.16).

Figure 3.16 - The Dimension Settings window – Units tab

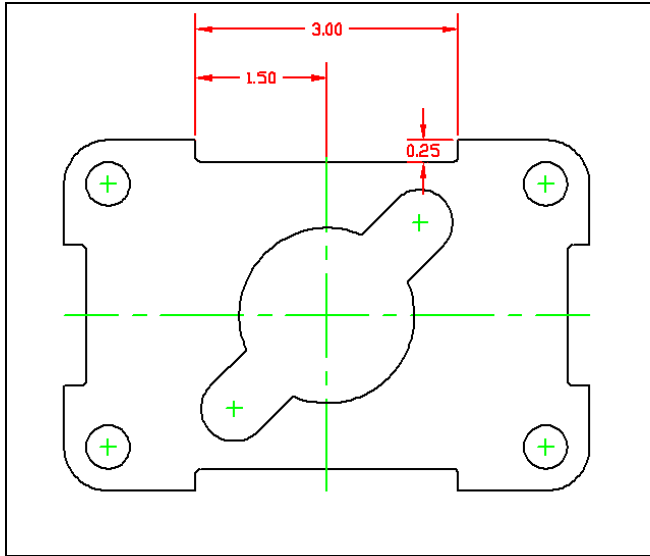
More Linear Dimensions

You may need to delete the previous dimension and place a new one for the settings to take effect. If you have done everything properly you will see that the 1.50 dimension is to our standard (Figure 3.17), so now you can place additional dimensions on the part. You do not want to be haphazard in placing addition dimensions, but rather follow a pattern to again improve efficiency and avoid errors of omission, like forgetting a dimension.



When placing new dimensions on a drawing, continue to specify extension line locations from the part (black line) and not from any existing dimension extension lines (red lines). The problem with dimensioning from 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 that dimension extension line will be hanging out in Model Space. All dimensions extension lines must reference an original entity or centerlines.

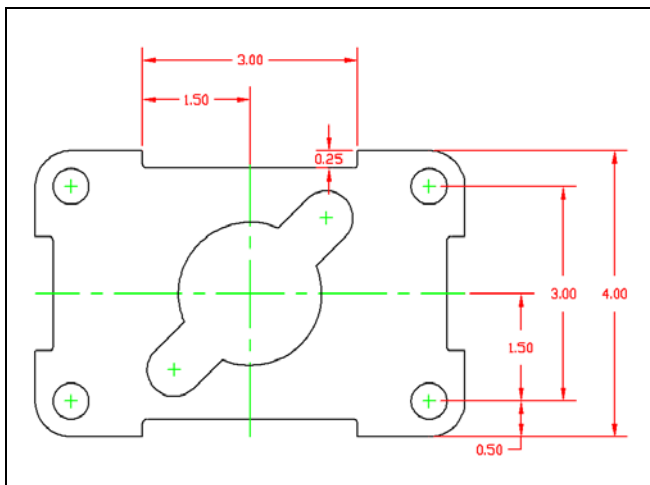
Figure 3.17 – The reformatted dimension



To continue dimensioning the top of the part, place 3.00 and 0.25 dimensions as shown in Figure 3.17. The Linear Dimension tool allows you to place horizontal or vertical dimensions by pulling the dimension in the third step of the tool towards the direction of placement. Do not worry that the lower dimension extension line of the 0.25 dimension is on the part. Select the dimension after you have finished placing it and green handles will appear. Grab the one at the end of the extension line that overlaps the part and drag it back so that it is parallel with the dimension label. Hit ESC to deselect the dimension.

Figure 3.18 – 3 linear dimensions

You will find that many times you will have to use a dimension's handles to modify its alignment and the placement of the label. This will require some trial and error but eventually you will find a pattern that works best. Try as best you can to match the placement of the dimensions as shown in this chapter.



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 3.19). 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. Use Entity Snaps to align the dimensions; this makes the dimensions much easier to read.

Figure 3.19 – 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 (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.

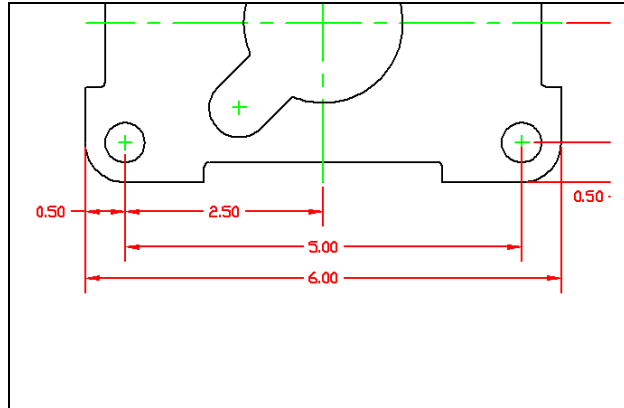


Figure 3.20 – Dimensioning the bottom

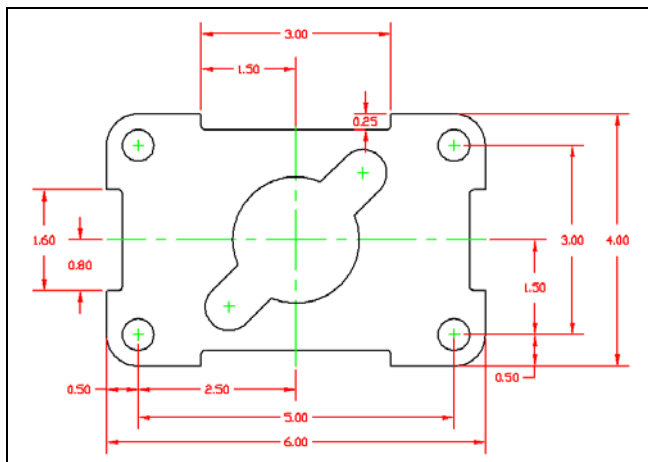


Figure 3.21 – Dimensioning the left

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.

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 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 Esnap settings.



Select the ESnap Settings tool on the Object Snap toolbar to open the Drawing Settings window. Select the Clear button to clear all previous Esnap settings. Check the Intersection checkbox and press OK to close the window. Now the other Esnaps will not interfere with your ability to select a reference point. Feel free to change Esnap 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 3.22. Next, place the 3.00 aligned-dimension on both 0.75 radius centerlines and position the 3.00 dimension text as shown in Figure 3.23.

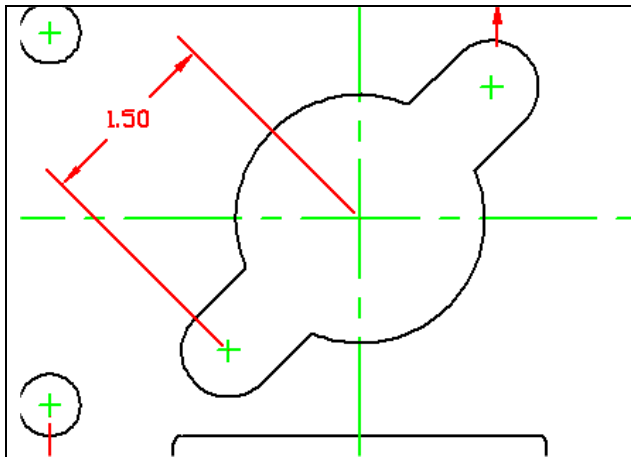


Figure 3.22 – 1st aligned dimension

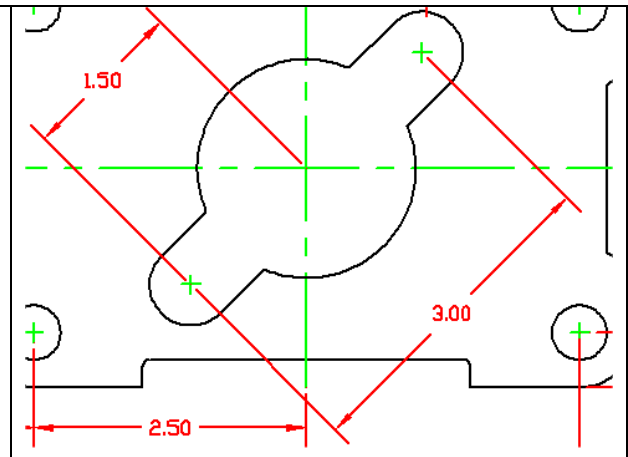


Figure 3.23 – 2nd Aligned dimension

Using the Radius Dimension Tool

The Radius Dimension tool is as easy to use as the Center Mark tool. Once you have selected 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.

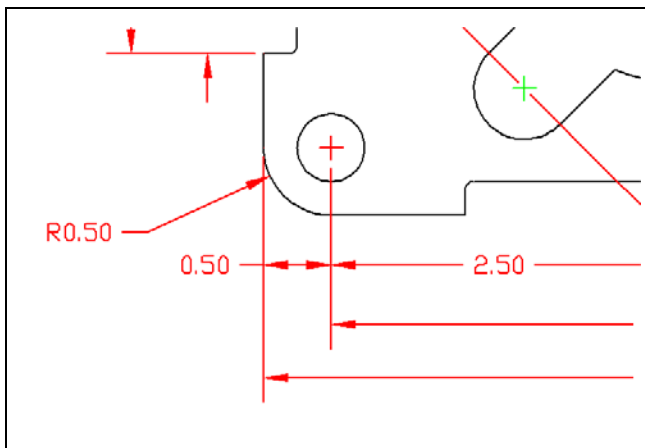
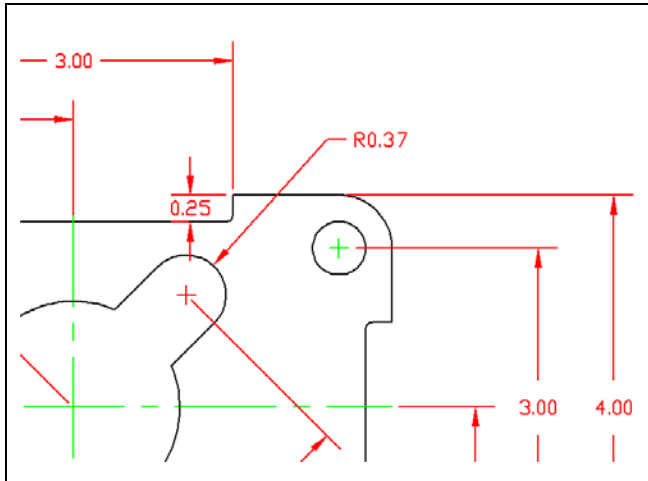


Figure 3.24 – The first radius dimension

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 will be placed on the drawing with the radius dimension (Figure 3.24).



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 R0.37 dimension text on the drawing (Figure 3.25).

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 and place the R0.06 dimension text as shown in Figure 3.26.

Figure 3.25 – Second radius dimension

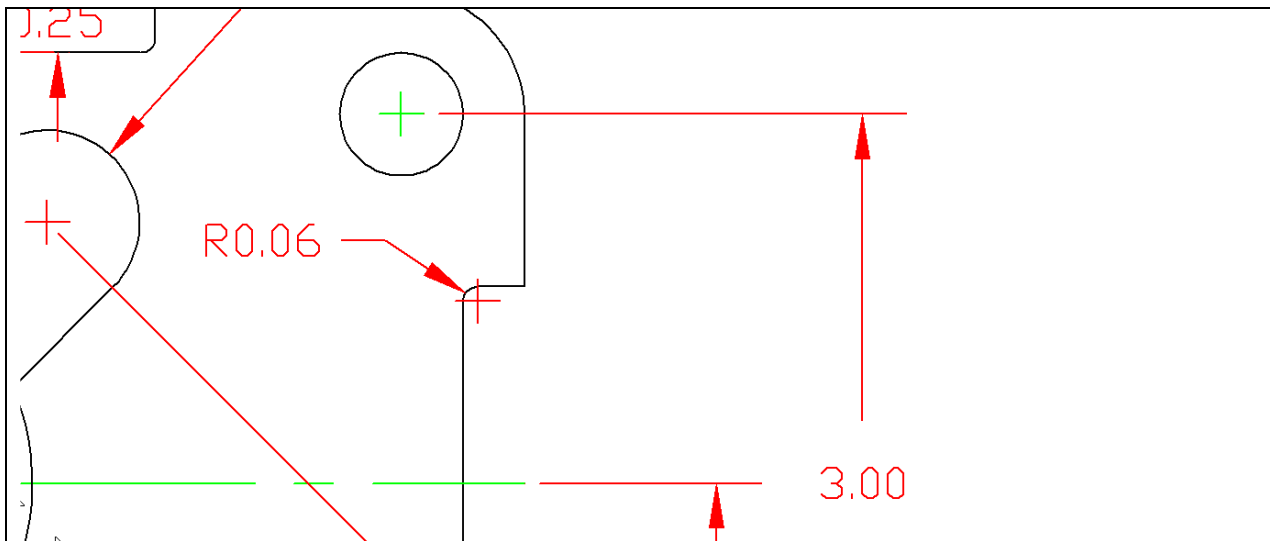


Figure 3.26 – Placing the last radius

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.



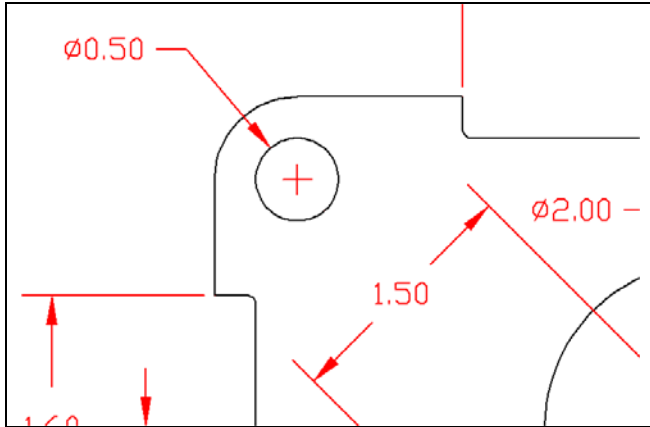


Figure 3.27 – The 0.50 diameter

Select the Diameter Dimension tool on the Dimension toolbar and 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).

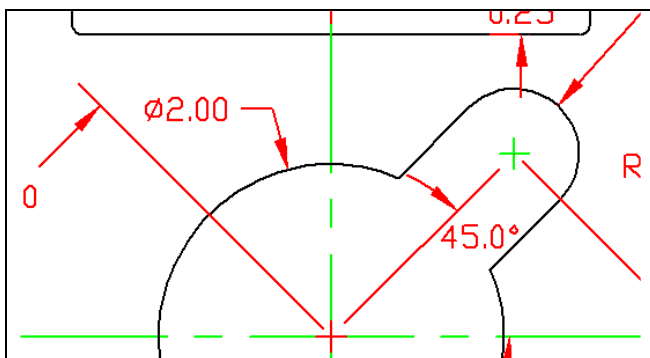


Figure 3.28 – The 2.00 diameter

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).

Using the Break Command

The Break command is a rarely used because the Trim command is so much more functional. Break is designed to abstractly remove portion of a line segment. Therefore, the Break command is perfect for your use here 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 Esnap 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).

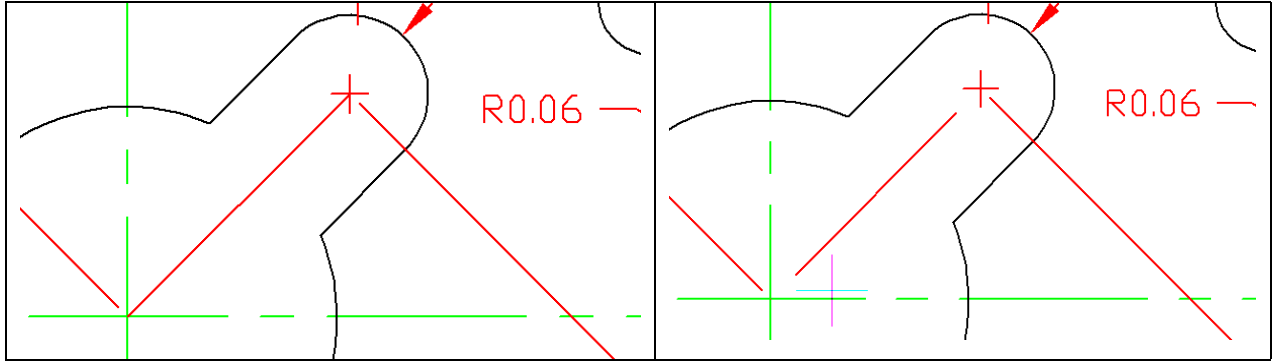


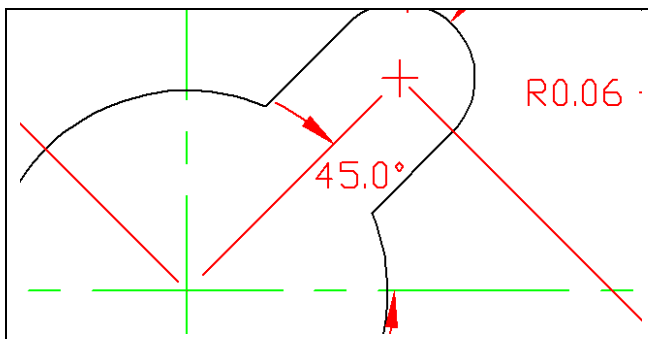
Figure 3.29 – Adding a line segment

Figure 3.30 – Breaking the line segment

Select the Break command on the Modify toolbar. The command line will prompt you to “Select entity to break:”, 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 top end of the line segment. Next the command line will prompt you to select the “First break point/<Second break 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 the bottom 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. It is located on the Dimension toolbar. Select the Angular Dimension tool and select two converging lines (non-parallel) to place the degree measurement on the drawing. For the first pick, select the red dimension extension line you just drew. 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).

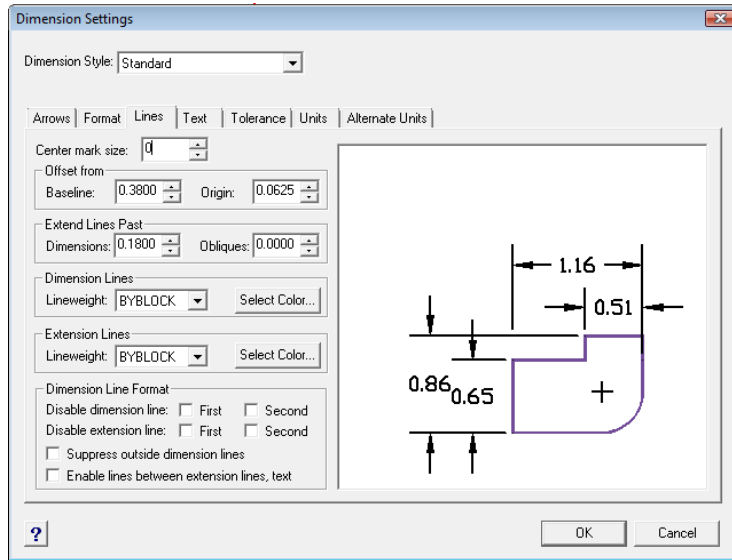


Once again you may need to use the handles of the dimension to get the alignment and label placement correct. There are a few changes you need to make to finish the dimensioning of the part and add to the clarity of the information. Three red centerlines need to be removed from the drawing. Also, you need to add “4 holes” under the 0.50 diameter dimension text.

Figure 3.31 – An angular dimension

Using Dimension Settings to Modify Single Dimensions

To remove the center marks from your radius and diameter dimensions, you must change each one individually. Start with the 0.37 radius dimension. Double-click it and the Dimension Settings window will appear, only this time you will notice that the Save and New buttons are gone at the top of the window. This is because you will be modifying the dimension settings for only one entity, the 0.37 radius.



Select the Lines tab and change the Center mark size to 0 (Figure 3.32). This will eliminate the center mark from the drawing. You will notice that all necessary center marks were placed on the center layer before you began placing dimensions.

Figure 3.32 – Setting Center Mark size to 0

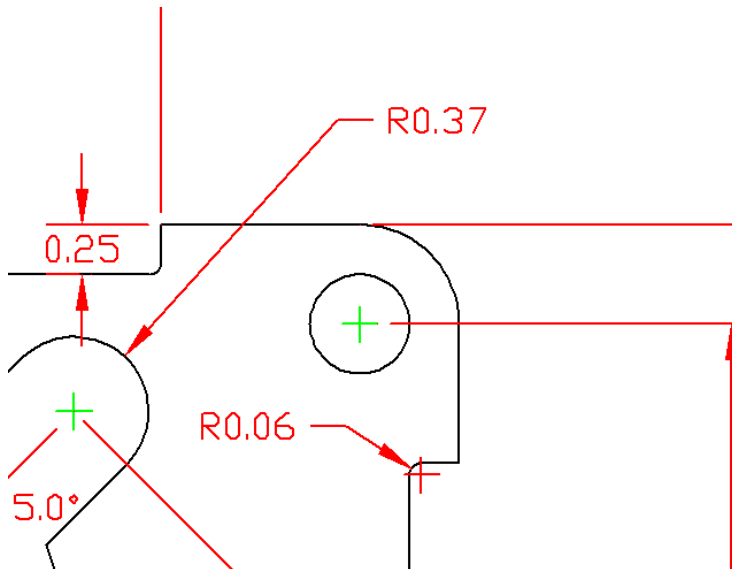


Figure 3.33 – The 0.37 dimension with no center mark

Using this same technique, remove the center marks for every radius and diameter dimension on the drawing. Now you only have one more thing to add before being completed with dimensioning the rectangle problem.

Using the Edit Text Tool

To add text to the $\emptyset 0.50$ dimension stating that this description applies to “4 holes,” select the “Text...” tool on the Text toolbar. Select the 0.50 dimension label and the Entity Properties window will appear. Type “4 holes” next to the “<>” and hit OK to add the note to the label (Figure 3.34). The “<>” in the Override text field stands for the dimension label, so don’t remove it. You can see the label in Figure 3.35.

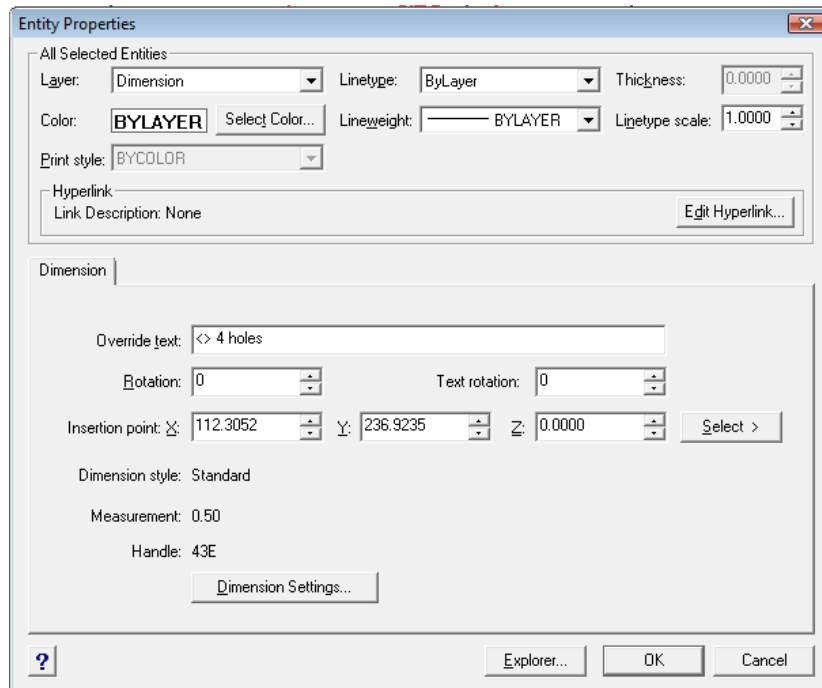


Figure 3.34 – Adding text to a dimension

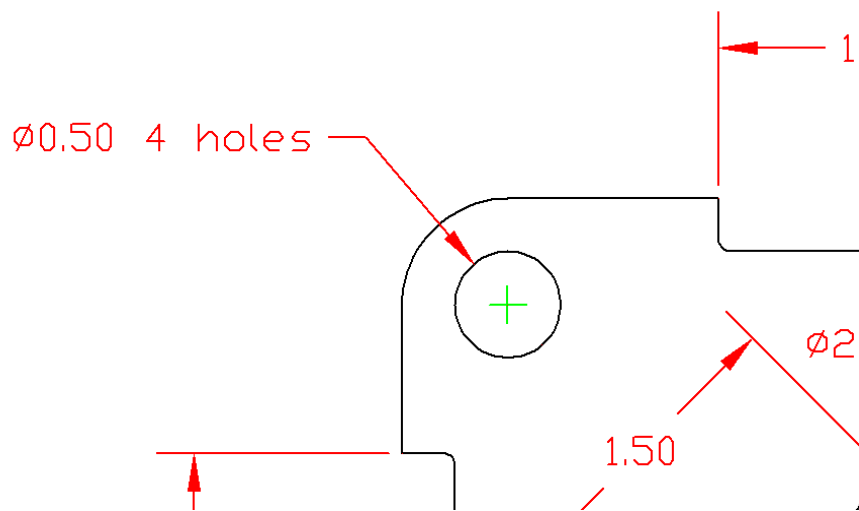


Figure 3.35 – The added label

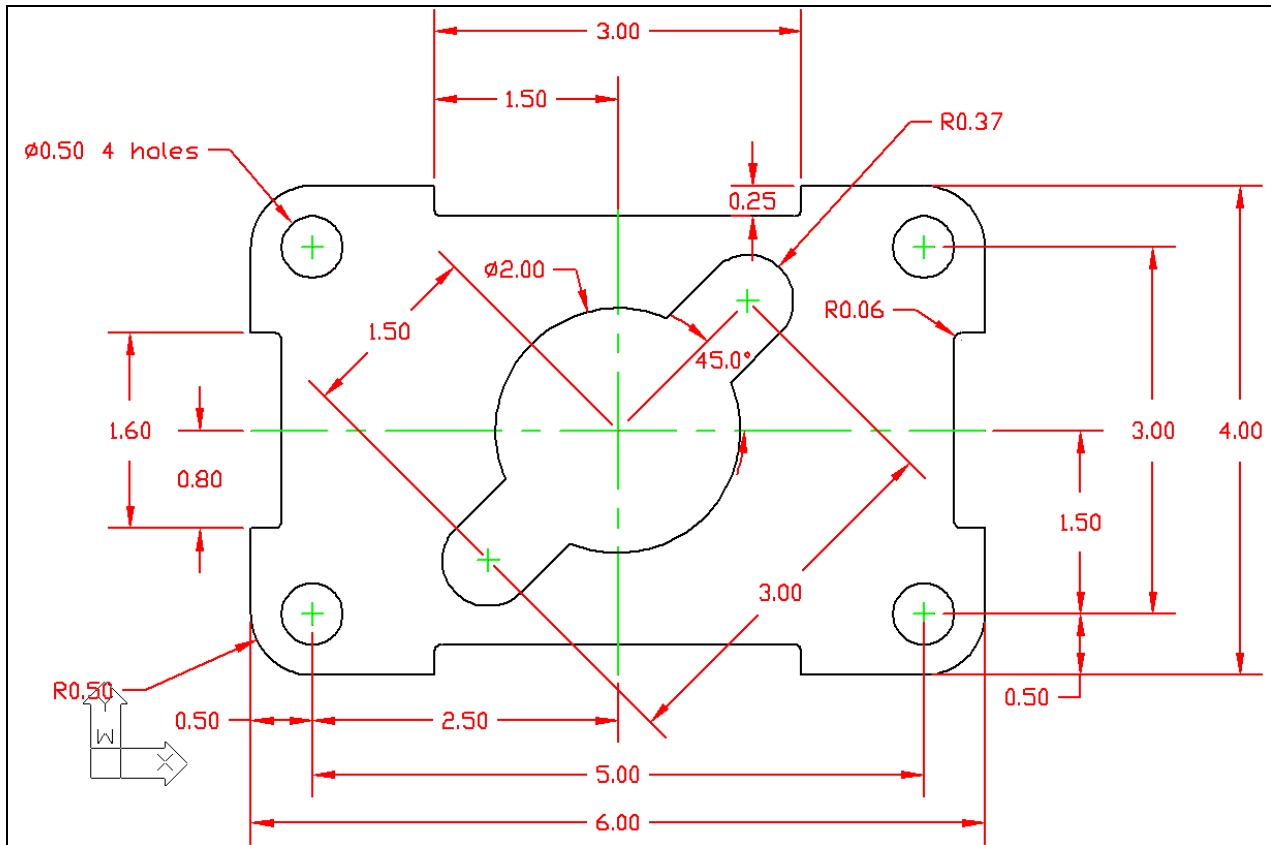


Figure 3.36 – Finished dimensioning the rectangular problem

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

*** World Class CAD Challenge 100-08 * - 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.**