

## Chapter 4 – Dimensioning Drawings

In this chapter, we are going to learn how to dimension the drawings we are creating. AutoCAD supplies many different tools for creating these dimensions quickly and easily. We will cover the basic tools.

The dimensions created by AutoCAD come from the dimensions you used when you drew the part. It is very important that you use the correct dimensions when you are drawing. Many students will use a scaling factor when they are drawing to make it easier to keep the entire drawing on the screen. This is completely unnecessary and undesirable. If the drawing or a line extends beyond the edge of the screen use the zoom command to show the entire drawing.

**zoom all** *{This will display everything you have drawn on the screen. If it is still too large, roll the mouse wheel towards you to reduce the size farther.}*

We are going to start by drawing the front, top, and right views of the object shown on the right. We will use a first angle projection for our drawings.

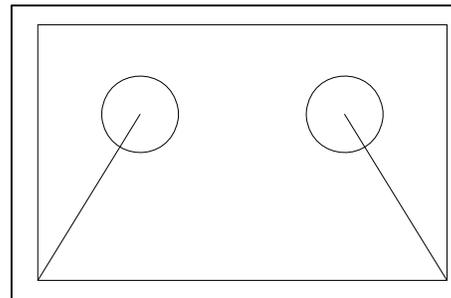
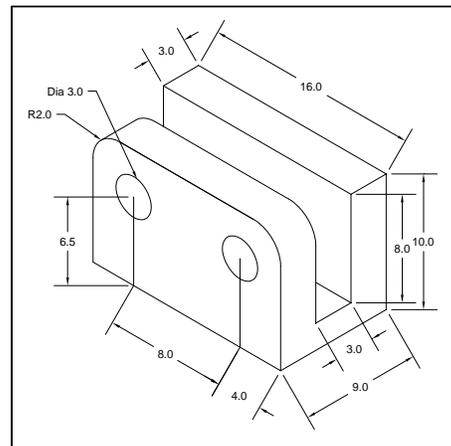
Start as you did before by creating Obj, Hide, Proj, Center, and Fold layers. Set the appropriate line types for hidden and center lines and select colors as appropriate.

You can start by drawing the front view. It is a basic box with two holes in it. Make sure you use the correct dimensions. Use exactly the same techniques you used in the last chapter to draw a simple box shown below with two holes as shown below. The two lines extending up from the bottom corners are just construction lines for the creation of the holes and they can be erased after the holes have been created.

Next we need to cut off the corners on the front plate of the part. This can be done with the **FILLET** command. Two things must be defined for the **FILLET** command before we can use it. We must set the radius of the fillet and in this case, we want to turn trimming off so that the tangent lines are left in place. Normally we would want AutoCAD to remove these lines.

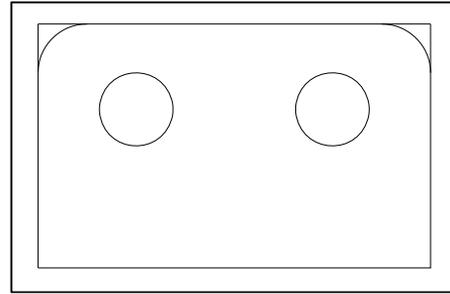
The **FILLET** command remembers the settings so we only have to make changes to the radius and trim settings when we want it to work differently than the last time we used the command. Initially the radius is set to zero and trimming is turned on. You will always need to set the radius the first time you use the command.

```
fillet
Current settings: Mode = TRIM, Radius = 0.0000
Select first object or
[Undo/Polyline/Radius/Trim/Multiple]: R           {enter R to change the radius}
Specify fillet radius <0.0000>: 2                {enter 2 to set the radius}
```



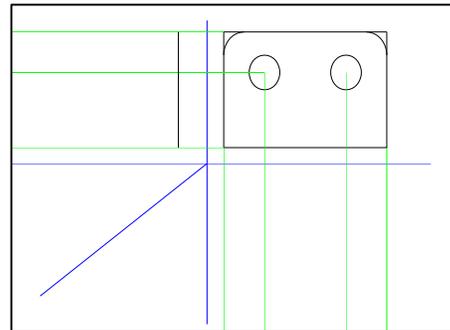
```
Select first object or
Undo/Polyline/Radius/Trim/Multiple]: T      {enter T to change the
                                                trim option}
Enter Trim mode option [Trim/No trim] <Trim>: No
```

Select the top and left side of the box to create the first rounded corner then repeat the command and select the top and right side of the box. You should be left with the object shown on the right.



Now we can draw the folding lines and start projecting the features shown on the front view to the right and top views. Remember that we are creating a first angle projection so the top view is below the front view and the right view is to the left.

After the folding and projecting lines are drawn, change to the object layer and draw the right edge of the object in the right view. You can draw it with the mouse then trim it using the projection lines like we did in the previous assignments. When you have finished, you should have a drawing similar to the one shown on the right.



The other vertical edges in the right view can be created with the **OFFSET** command. This command will duplicate a line at a specified offset distance. Enter

```
Offset
Specify offset distance: <0.0>: 3
Select object to offset:
```

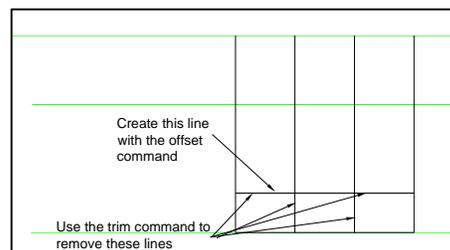
```
Select object to offset:
```

*{select the right edge of the object then click to the left of the line}  
{click on the new line you created then to the left of that line to create another line offset by 3}*

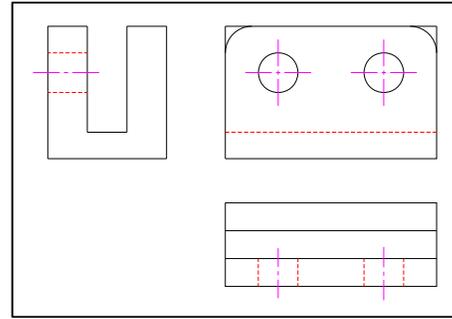
Repeat the above process by clicking on the line just created then to the left of the line to place the new offset until all of vertical lines have been created.

Next, you can draw the bottom of the object by creating a line that connects the bottoms of the left most and right most lines. Use the **OFFSET** command to create the slot in the bottom of the object then use the **TRIM** command to trim the line to the correct length. The results are shown in the figure on the right.

The **OFFSET** and **TRIM** commands are very useful and can frequently be used to eliminate typing coordinates when creating lines and other objects.



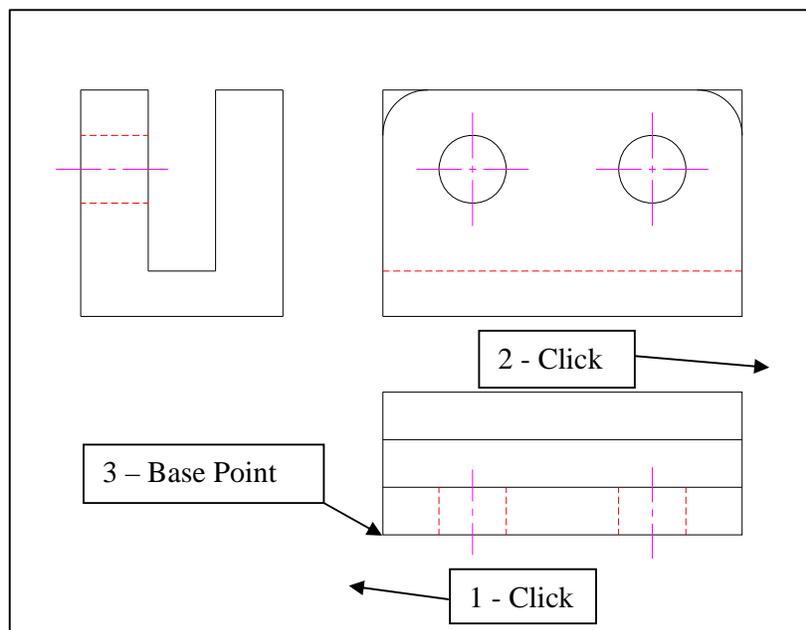
Continue as we have before and complete the three views. These are shown on the right. Remember that this is a first angle projection. The right view is on the left, the top view is on the bottom, and the front view is between the other two.



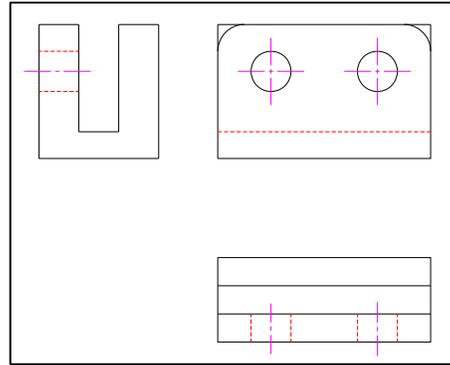
## Moving Views

If the views you have drawn are too close together to accommodate dimensions, use the **MOVE** command to move them farther apart. Use the Window option of the **MOVE** command to select only those lines and arcs that are part of the view and move them farther down the drawing. Before you start, make sure that **ORTHO** is set to **ON** so that the view can only be moved vertically. The **MOVE** command is illustrated below.

**Move**  
 Select objects: **w** *{enter w to create a window around the object}*  
 Specify first corner: *{1 - click below and to the left of all lines in the top view}*  
 Specify opposite corner: *{2 - click above and to the right of all of the lines in the top view}*  
 Select objects: *{if all of the lines have been selected, press enter otherwise continue to select lines}*  
 Specify base point or [Displacement] *{3 - click on a point in the top view you wish to use as a handle to move the view}*  
 Specify second point or *{drag the view to where you want it and click with the mouse to relocate it}*

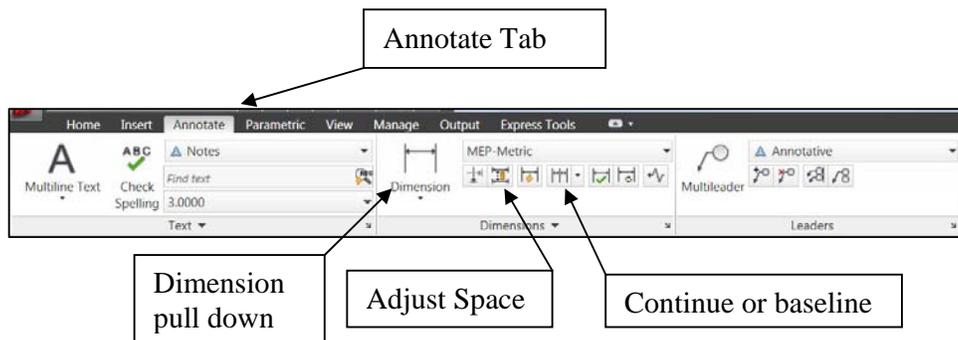


The figure on the right shows the three views after the top view has been moved. You now have space to create the dimensions.



### Adding Dimensions

AutoCAD supplies a several menu items for creating dimensions. Clicking on the “Annotate” tab to produces the ribbon shown below.

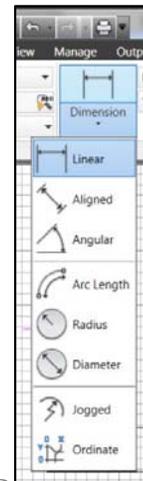


There are two very useful commands. There are:

- Adjust Space** this controls the spacing between nested dimensions
- Continue** this allows dimensions to continue so that the end of one is the beginning of the next
- Baseline** this allows multiple dimensions to start at the same base location

If you click on the “Dimension” pull down AutoCAD will produce the menu shown on the right.

- Linear** produces normal horizontal and vertical dimensions
- Aligned** produces dimensions that are aligned with part features
- Angular** is used to create angular dimensions
- Arc Length** dimensions the arc length of an arc or a polyline
- Radius** creates a radius dimension
- Diameter** creates a diameter dimension

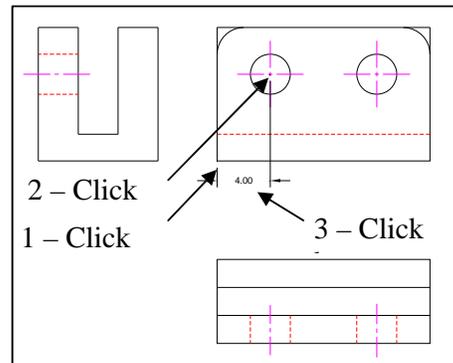


We can use these tools to create dimensions on the drawing we have created.

Remember before you start creating dimensions there are several rules to follow.

1. The person who makes the part must measure it exactly the way you dimensioned it. If you dimension it in a way that is difficult to measure, you have made the part much more expensive to make without improving the quality or usefulness of the part.
2. Dimension in the view where you see the shape. Holes should be dimensioned where you see the round shape.
3. Do not dimension to hidden lines if possible. You can usually dimension the feature in a different view where it is not hidden.

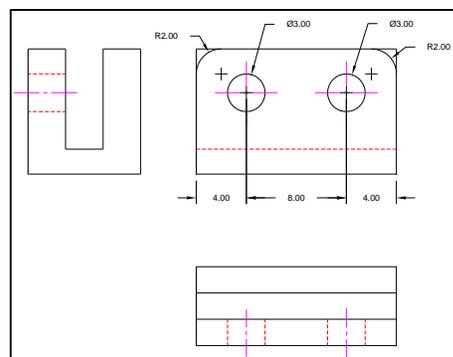
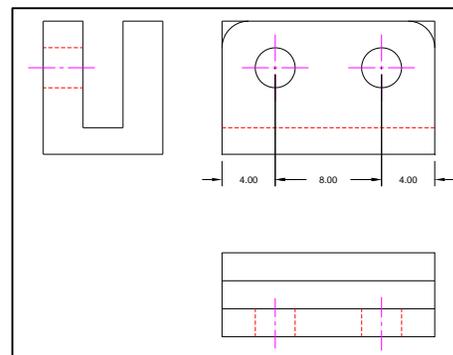
First, click on the “Linear” dimension button then (1) click on the bottom, left corner of the front view. This is the beginning point for the dimensions. We want to create a dimension from this point to the center of the left circle. Click on the center (2) of the left most circle in the front view then move the mouse to a position below the front view and (3) click to locate the text.



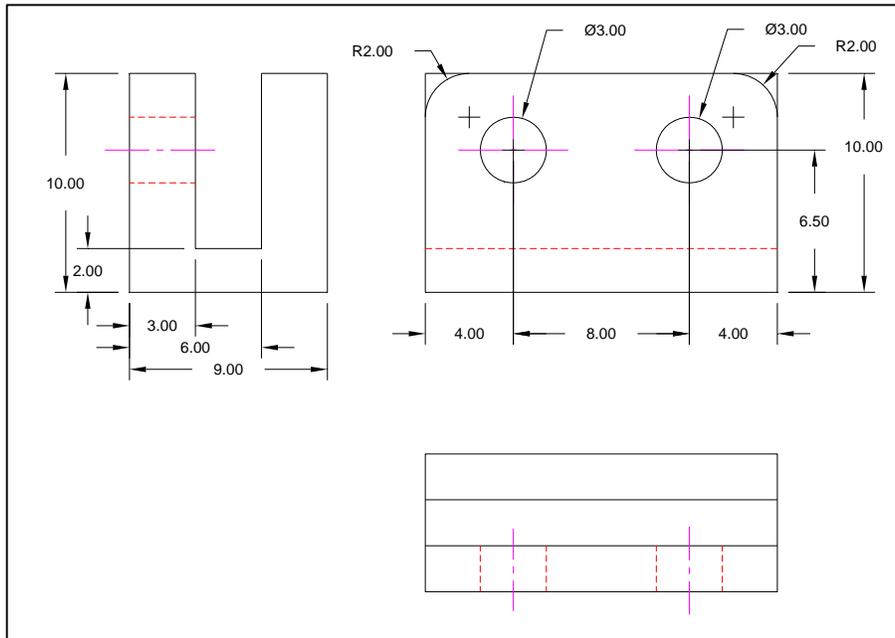
Now click on the “Continue” button and dimension to the center of the next circle then to the lower right corner of the part in the front view. Press the ESC key when you have finished. Your drawing should look like the one on the right.

Next we will dimension the holes and the rounds in the corners. Click on the dimension pull down and click on the “Diameter” button. Use this command to dimension both circles.

Next use the “Radius” button to dimension the rounded corners. Click on the button, then click on the arc you want to dimension, then move the mouse to the place you where you want the annotation to appear and click to place it. When you have finished, the drawing should look like the one at the right.



Next we will add width dimensions to the right view. We will use the left face of the right view as a base datum and dimension from there. Click on the “Linear” dimension button and create a dimension from the left face to the left side of the slot. Next click on the arrow at the right side of the “Continuous” button and click on the



“Baseline” button. Use the same technique to dimension the height of the slot and the height of the part. Add dimensions to the view until it looks like the figure below.

## Dimension Size

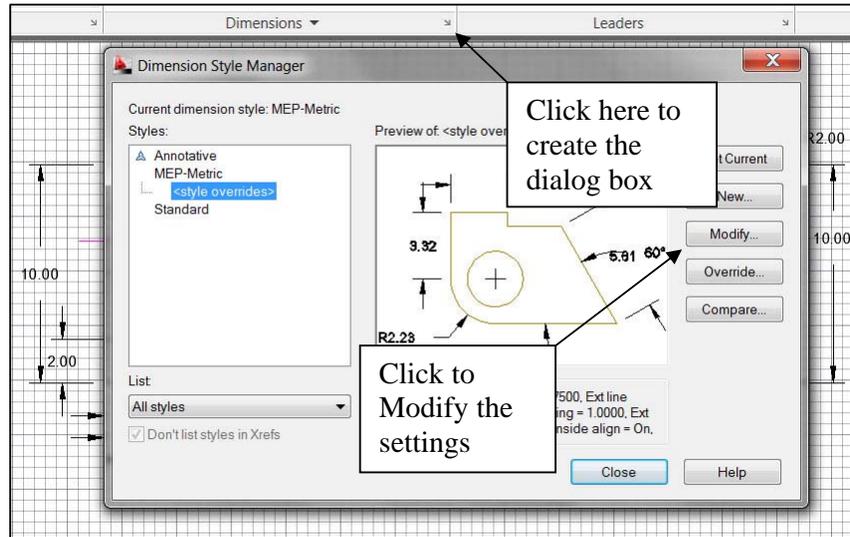
Sometimes, the dimensions are either too large or too small. You can usually fix the problem with the **DIMSCALE** command. The **DIMSCALE** command lets you change the scale of the dimensioning size by specifying a scaling factor. A factor greater than one makes the dimensions larger and a factor smaller than one makes them smaller. Changing the scaling factor affects the size of the text, arrowheads, and gap between the extension lines and the point selected for the dimension.

The **DIMSCALE** command is illustrated below. Here we are changing the dimensioning scale from 1 the default value to 4 which will make the dimensions 4 times as large.

```
dimscale
Enter new value for DIMSCALE <1.0000>: 4
```

The **DIMSCALE** command will not change dimensions that have been drawn. It only affects new dimensions. You must erase the old dimensions and recreate them with the new dimensioning scale factor. You may have to try several different dimensioning scales before getting the correct size for the dimensions.

## More Dimensioning Problems

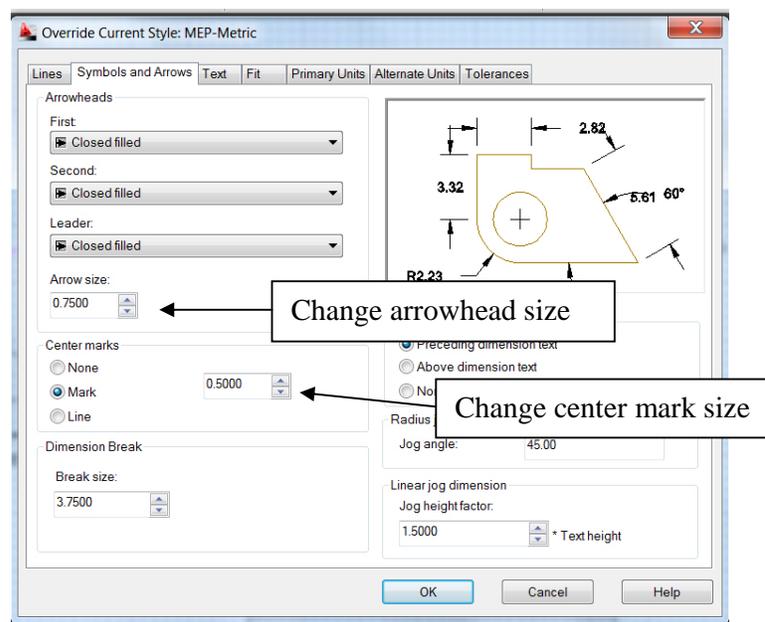


The DIMSCALE command will change the size of the dimensions but there may be other problems. A frequent problem is the number of decimals shown in the dimension. The AutoCAD default is 4 decimal places and frequently this is too many. To change the number of decimal places and many other dimensioning problems, you must open and make changes in the dimension style manager.

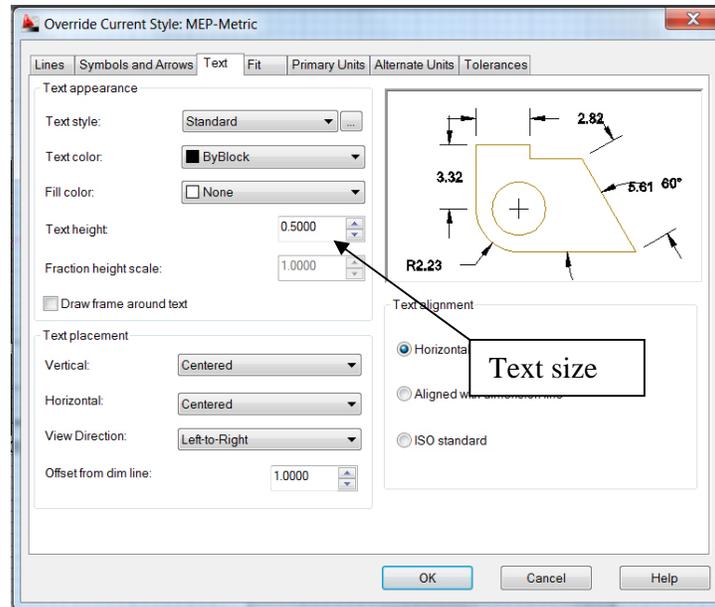
You open the dimension style manager by clicking on the small arrow to the right of the dimension tab. This produces the dimension style dialog box. In this box click on the “Modify” button then in subsequent dialog boxes, make appropriate changes to the dimensioning.

We will not cover the dimension styles in detail here but we can point out some of the more commonly used style components that can be changed. When you click on the “Modify” button, the style manager opens. It is a dialog box with several tabs at the top. Clicking on a tab changes the dialog box allowing you to change different features of the dimension.

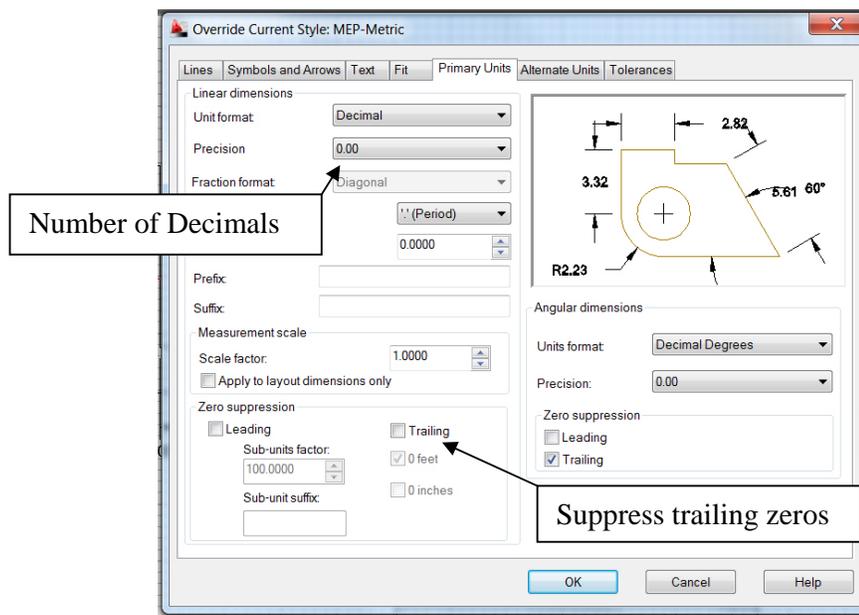
If you click on the “Symbols and Arrows” tab, the following dialog box will be created. You can use this dialog box to change the size of arrowheads and the size of the center marks for circles and arcs.



You can change the text height used for dimension text with the dialog box under the “Text” tab.



The number of decimals shown in the dimension can be changed with the “Primary Units” tab.



When you have made the changes you want to make, click on the “OK” button at the bottom of the dialog box. The “Style Manager” dialog box will appear. Click on the “Set Current” button then on the “Close” button. This will make the changes you have made the current settings for AutoCAD.

You must erase and redraw the dimension to see the changes you have made. Like many items in AutoCAD, the changes are not dynamic.