

Making a Drawing Template

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

- **Starting from scratch for the last time**
- **Creating new Layers in an AutoCAD drawing**
- **Setting Object Snaps**
- **Modifying text styles**
- **Creating a Mechanical dimension style**
- **Modifying the drawing units**
- **Setting the drawing limits**
- **Modifying the linetype scale**
- **Saving a drawing template**

In the first two units, the Rectangular and the Circular problem, you created layers, set Object Snaps, modified the Standard dimension style, and made other system variable changes to make the drawing clear to read. In this chapter, you will again repeat all of these alterations, but you will save those settings as a mechanical template so that in the future you won't have to waste time changing the settings every time you start a new drawing. A mechanical template is not the only type of drawing model required by designers, engineers and architects. Architectural, civil, electrical and metric templates are also useful when creating prints of each type. To create a template, you need to start the drawing from scratch.

Starting from scratch for the last time

Pick the New tool on the Quick Access toolbar and the Select template window will appear. Click the down arrow to the right of the Open button, and select the Open with No Template - Imperial (feet and inches) option. This option will set the conventional measurement used in the United States (Figure 8.1). The metric choice will not only use metric units but decimal numbers such as 1.500 will be displayed as 1,500 because the comma is used as a decimal point in the metric system.

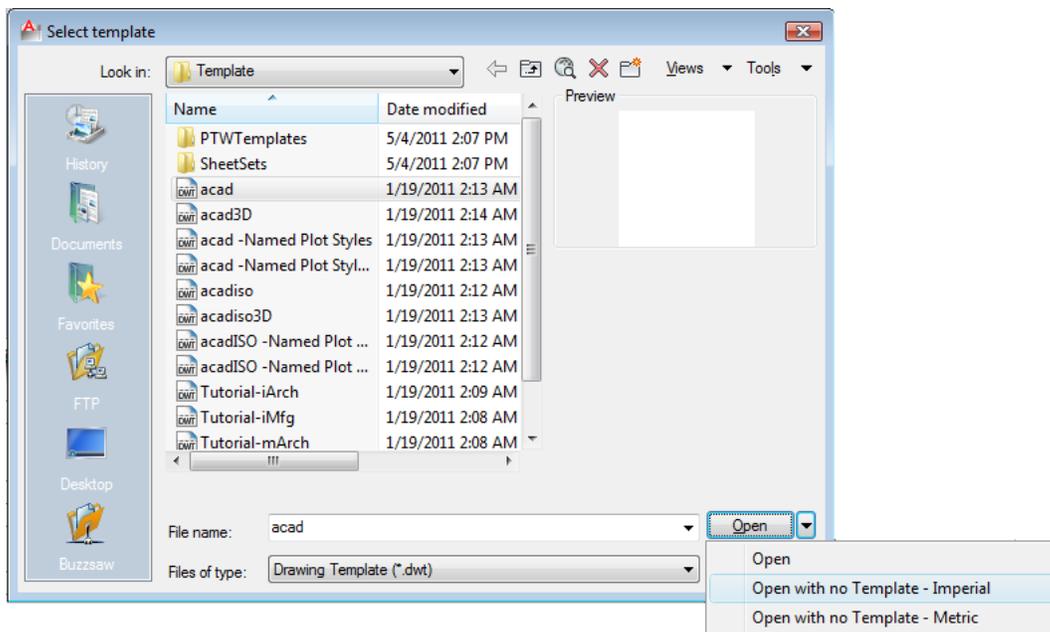


Figure 8.1 – Opening a new file with no template

Creating New Layers in an AutoCAD Drawing

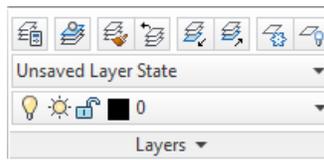


Figure 8.2 - The Layers panel

When expanding the orthographic drawing and adding more than one view, such as a top and front view, you will need new linetypes such as hidden, section and hatch to explain these views to the machinist or construction worker. You will also need new layers for the drawing border and possible hatching for cross section views. To create new layers using the Layers Properties Manager, select the Layer Properties command button in the top left-hand corner of the Layers panel (Figure 8.2). Add the following layers shown in Table 8.1 using the New button in the Layer Properties Manager.

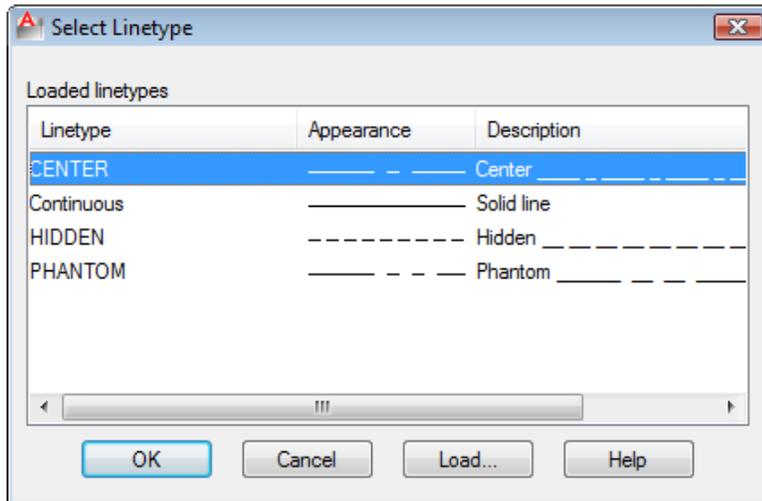


Layer Name	Color	Linetype
Border	8 (dk. gray)	Continuous
Center	Green	Center
Dimension	Red	Continuous
Hatch	254 (lt. gray)	Continuous
Hidden	Cyan	Hidden
Section	Magenta	Phantom
Text	Blue	Continuous

Table 8.1 - Mechanical Template layers

To create this template, you need make seven new layers. To create the layers, select the New button in the Layer Properties Manager. Immediately, the layer name is ready for editing, so type, “Border” and hit ENTER twice to add another new layer. Continue to add the new layers until you have added all the layers listed in Table 8.1. Next, you need to choose different colors to help distinguish between the layers. To change colors, first place your mouse arrow over the color box in the Border layer row and single left click with your mouse. The Select Color window will appear in front of the Layers Properties Manager. Select the gray (color 8) from the Standards Colors row and select the OK button. You will now see that Border layer’s color is 8 (gray). Repeat the process until you have changed all the colors to those listed in Table 8.1.

To finish creating and modifying layers, you need to change the linetype to a Center linetype for the Center layer, Hidden linetype for the Hidden layer, and the Phantom linetype for the Section layer. Place your arrow over “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 8.3). Note that only the Continuous linetype is loaded. Select the Load button and the Load and Reload Linetypes window will appear. Scroll down, highlight the CENTER linetype and select the OK button.



You will now see that the Center layer has a CENTER linetype (Figure 8.4). Place your mouse arrow over the text “Continuous” in the Hidden layer to re-enter the Select Linetype window. Highlight the Hidden linetype in the Select Linetype window and select OK. Repeat the same steps for the Section layer to load the Phantom linetype. When finished, your Layer Property Manager should resemble the one shown in Figure 8.4.

Figure 8.3 - The Select Linetype Window

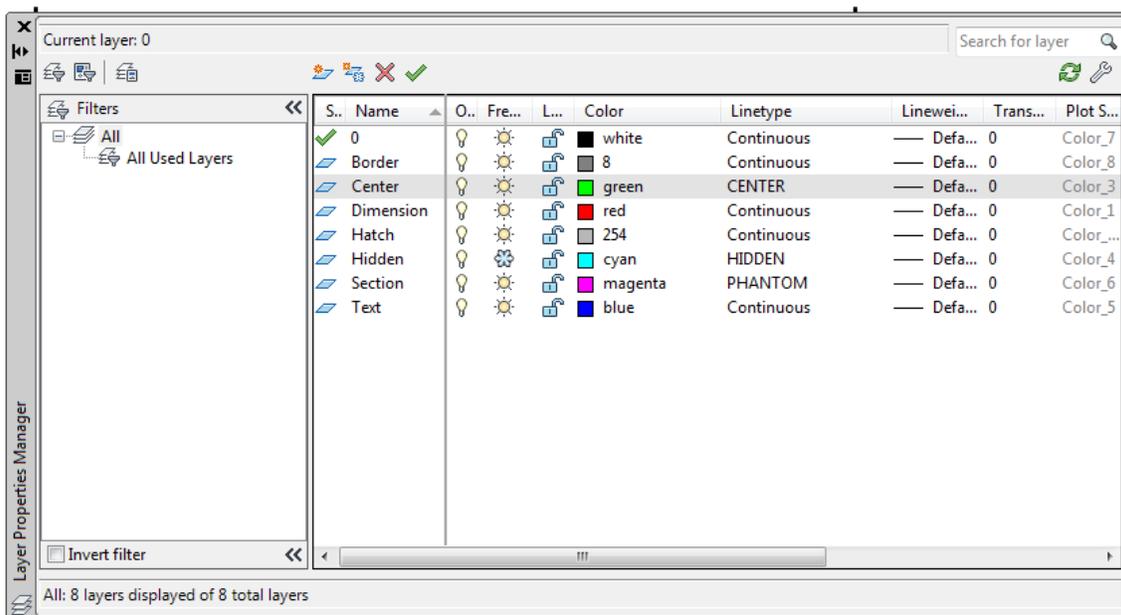


Figure 8.4 - The Layer Properties Manager window

Setting Object Snaps

Setting Object Snaps will allow you to select points precisely in relation to other objects. Bring up the Object Snaps Settings by right-clicking on the Object Snaps icon on the tray at the bottom of the screen and selecting Settings. In the Mechanical template, you need to select the Center Object Snap Mode for grabbing the center point of a circle, recognized by a green circle when in the zone (Figure 8.5). You also need to select the Endpoint, Midpoint, Quadrant, Intersection and Perpendicular Object Snap modes.



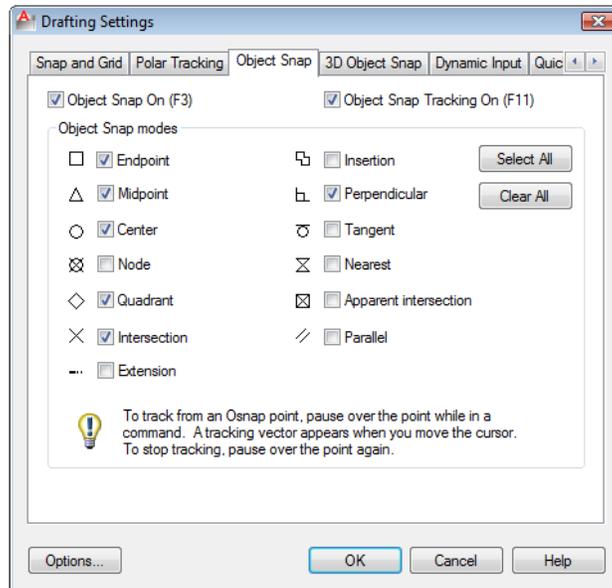


Figure 8.5 – Object Snap Settings

Modifying the Text Style

Many companies or organizations will want you to change the font that will be used in their drawings. You probably should not change many AutoCAD system defaults, but you can make a new Text Style for the Mechanical template. For the principle of ease, you will name the Text Style “Mechanical”. Open the Text Style list box on the Text panel under the Annotate tab and select Manage Text Styles.

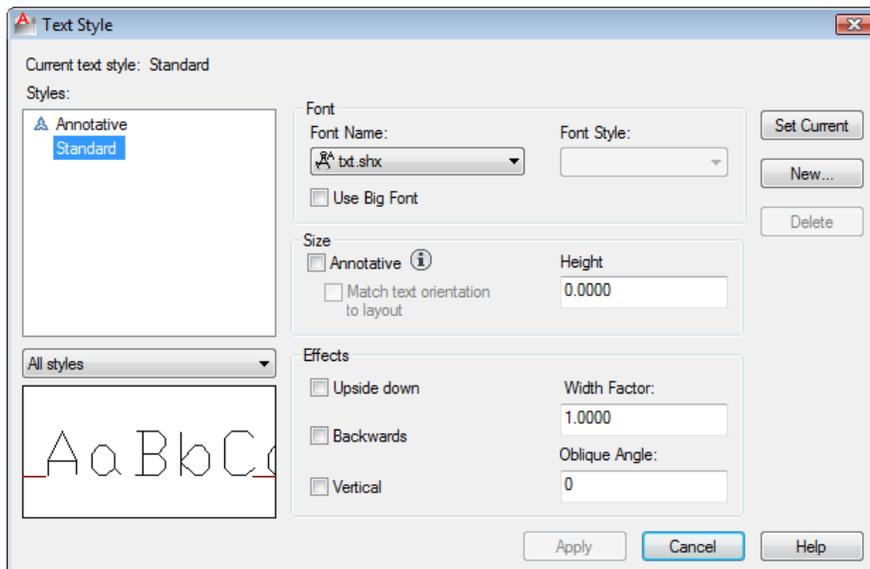


Figure 8.6 – Text Style window

As you can see in Figure 8.6, the system default is set to the Standard style and “txt” is the Font. In the Preview window in the bottom left you can see that the Txt font is edgy and may not meet the visual requirements of your corporation. The Height is set to 0.0000, which means that when using Mtext the font will be the size set in the Text panel. Notice there are check boxes for Upside down, Backwards, and Vertical text if you so desire.

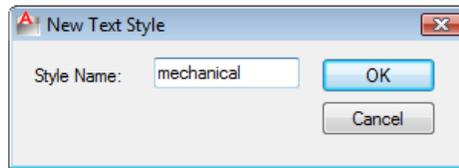


Figure 8.7 – Naming a New Text Style

To make a new text style, select the New button and the New Text Style window will appear. Type the word “mechanical” for the Style Name and select the OK button as shown in Figure 8.7. The Text Style window will show the style name as mechanical. Pick Arial for the Font name as shown in Figure 8.8. Select Apply and OK to exit the dialogue box.

To change the default text size, type textsize at the command line. You will notice the default is 0.2000. Type 0.125 and hit ENTER as shown in Figure 8.9.

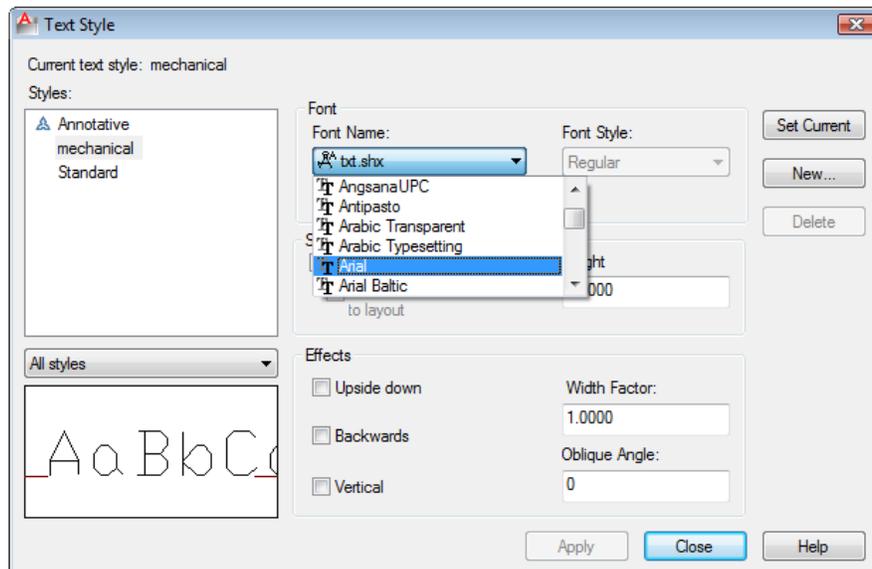


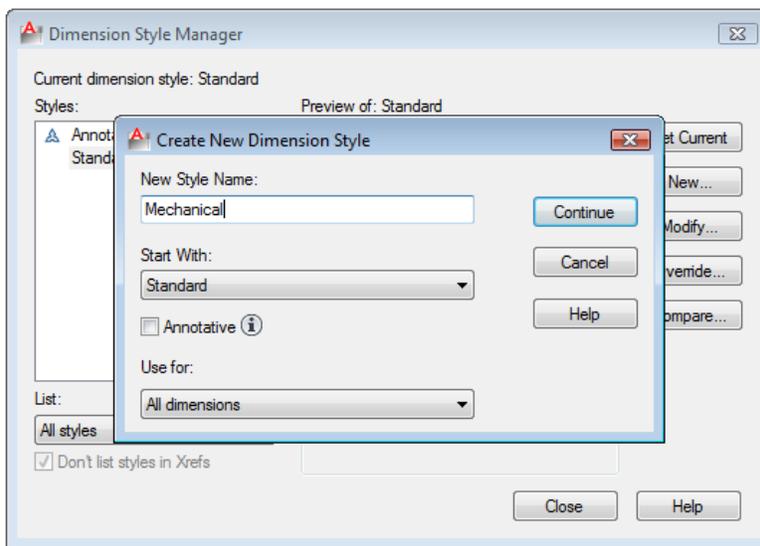
Figure 8.8 – Selecting a Font with the Text Style window



Figure 8.9 – Setting the Text Size variable at the Command Line

Creating a Mechanical Dimension Style

In your previous problems and practice exercises, you opened the Dimension Style window and modified the dimension variables on every drawing. As with Layers, Object Snaps, and Text Styles, you want to make a custom variation using the Standard Dimension Style as the starting point. Open the Dimension Style Manager window by selecting the small arrow in the bottom right-hand corner on the Dimensions panel under the Annotate tab (see Figure 8.10). Here you will see a list of Styles on the left, a Preview of the style highlighted, the Set Current, New, Modify, Override and Compare buttons, Description field and Close and Help buttons. In the past you modified the Standard Style using the Modify button, but today you will select the New button to create the Mechanical Dimension Style.



Type “Mechanical” in the New Style Name, retain the Standard style to Start With, and Use for all dimensions. Then select the Continue button to enter the dialogue box that will allow you to set all the variables for the new Mechanical Dimension Style (Figure 8.10).

Figure 8.10 – Dimension Style Manager

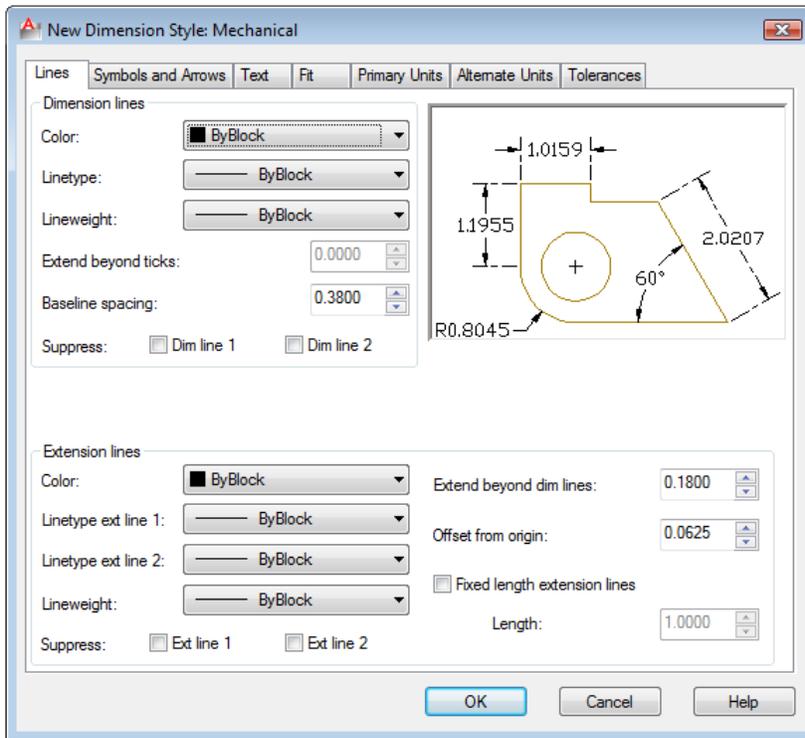


Figure 8.11 – Modify Dimension Style, Lines tab

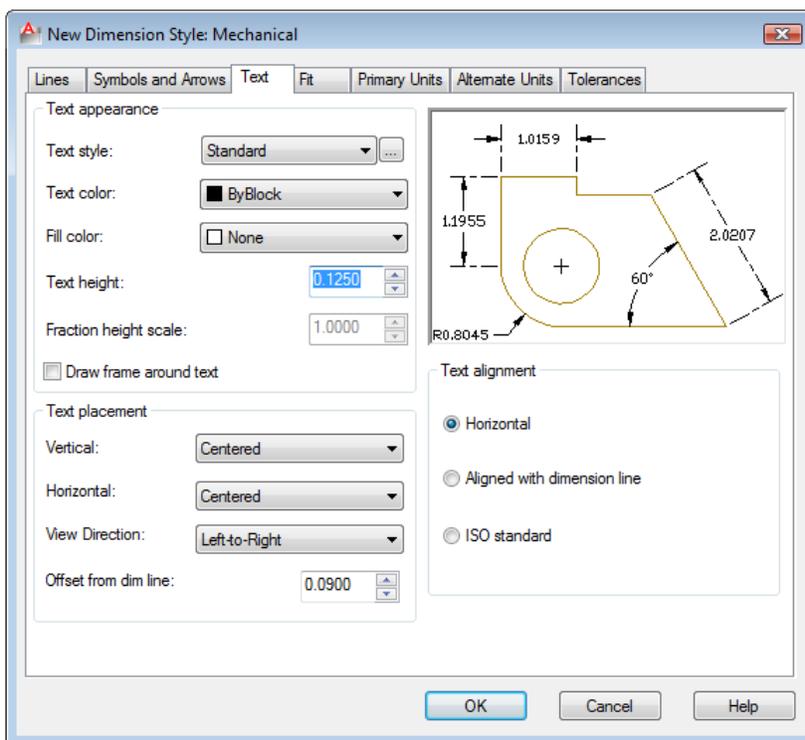


Figure 8.12 – Modify Dimension Style, Text tab

The New Dimension Style window will appear. The Lines tab shown in Figure 8.11 allows you to alter every detail of Dimension Lines and Extension Lines. The Symbols and Arrows tab allows you to change the settings for Arrowheads and Center Marks.

Proceeding to the Text tab, you can choose the Text Appearance, Text Placement and Text Alignment. Many of the establishments where you work will change the Text Height to 0.937 or 0.125. The 1/8 (0.125) text height is plenty large enough to read on prints and allows the dimension to remain close to the feature that is being described. 3/32 is the smallest text still able to be read. Here, change the Text Height to 0.1250.

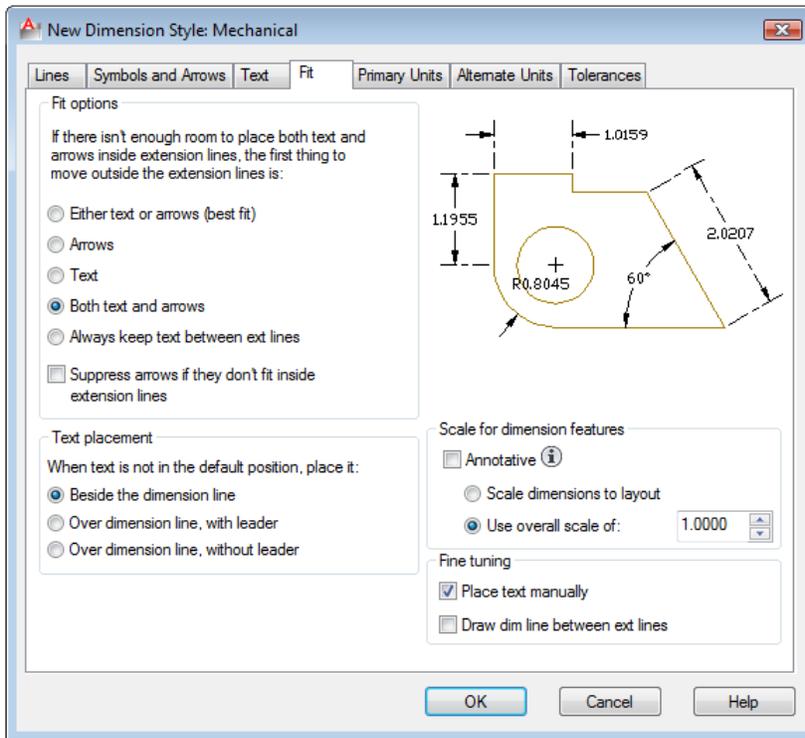


Figure 8.13 – Modify Dimension Style, Fit tab

The Fit tab allows you to adjust the dimension variables to compensate for spacing issues. You will make a couple of changes on this tab. First select the radial button for “Both Text and Arrows.” By choosing this radial button, Diameters will be flanked by regular dimension lines instead of a leader. Now check the box for placing the text manually when dimensioning. This will allow you to move the text in every direction on the final step of inserting a dimension (Figure 8.13).

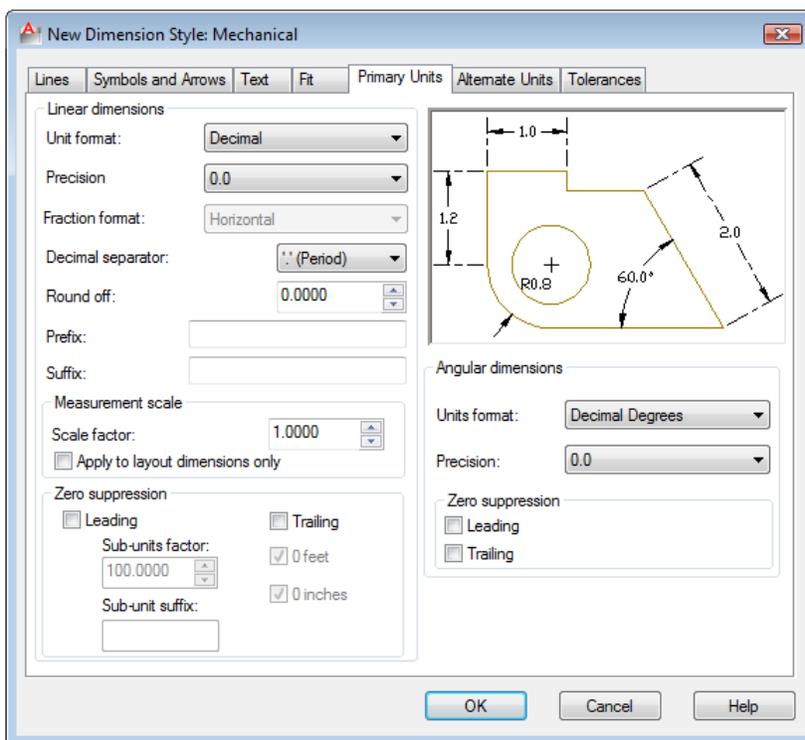


Figure 8.14 – Modify Dimension Style, Primary Units tab

In the Primary Units tab, the Unit format will remain Decimal, but change the Precision settings for Linear and Angular dimensions to 0.0 as shown in Figure 8.14. The other Linear Dimension settings are changed periodically, such as Round Off, but the 1.0000 Scale Factor in the Measurement Scale should stay always at one. Confusing this setting with other scale factors like line or text scale will change the value of all of the dimensions in the AutoCAD drawing.

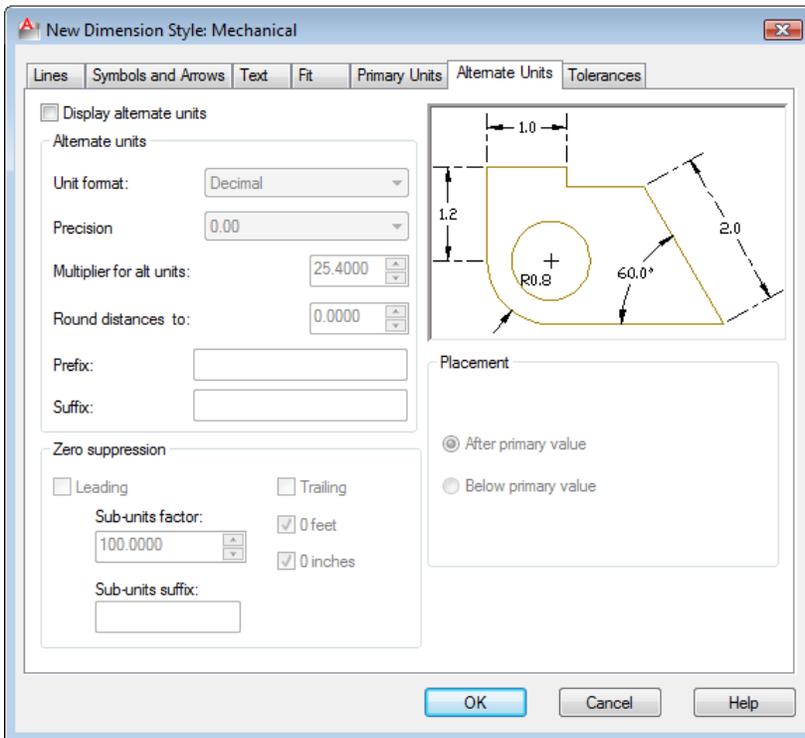


Figure 8.15 – Modify Dimension Style, Alternate Units tab

The Alternate Units tab would be used if you wanted to list the metric value of the English unit below or along side the dimension text. For example, if you checked “Display alternate units,” a dimension showing one inch will have the (25.40) alternate dimension in parenthesis below the 1.000 text. You do not need to check the “Display alternate units” checkbox on the Mechanical template.

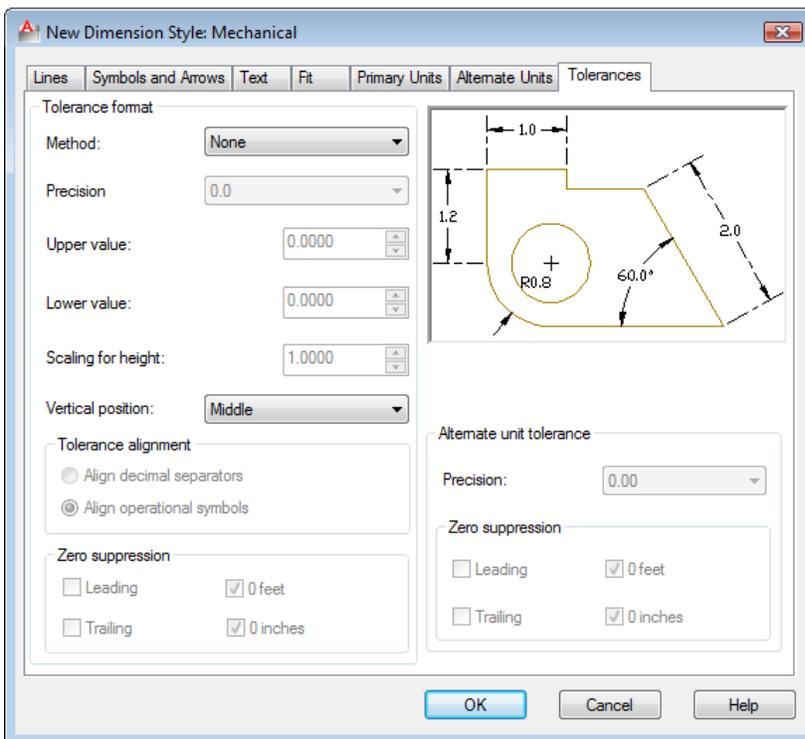
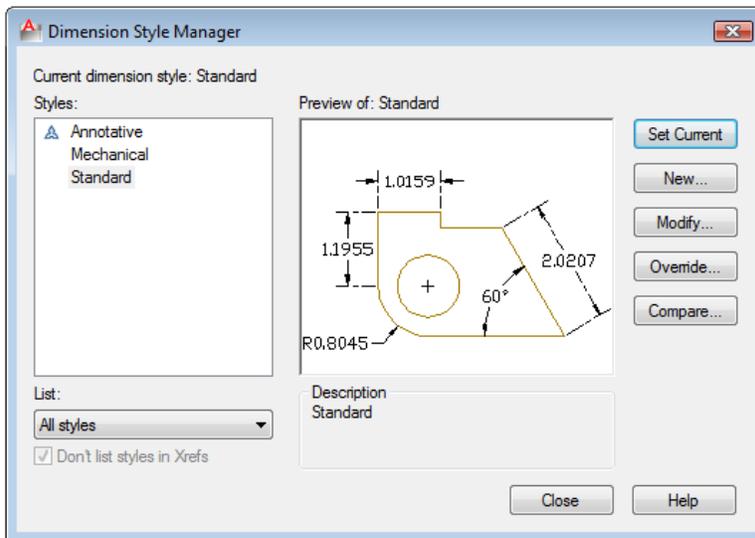


Figure 8.16 – Modify Dimension Style, Tolerances tab

The Tolerance tab is an option mostly used by mechanical designers or engineers when they need to discuss the range of allowable error in the part when manufacturing or accepting the components at the production site. Right now this is being addressed in the notes that you type in every drawing you do. As parts become more complex, this feature will be utilized to correctly show the detail dimensions (Figure 8.16).

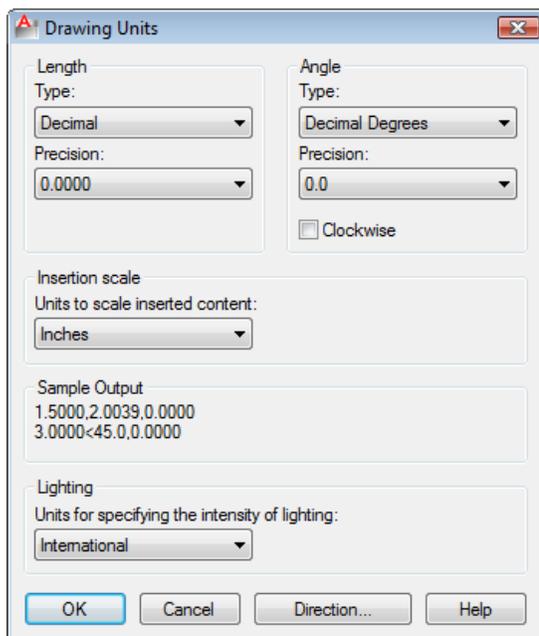


To save the changes that you made in the Mechanical Dimension Style window, select the OK button. When you have more than one style listed in the Styles window in the left side of the dialogue box, highlight the style you desire and press the Set Current button. Now press the Close button on the first window (Figure 8.17) to finish the procedure.

Figure 8.17 – The finished Mechanical dimension style

In the other templates that you will create, you might change arrow heads to architectural ticks and the Unit Format from Decimal to Architectural. You will want to make changes to the dimension variables based upon what your customer or industrial workers expect to see in drawings. Remember, in most cases you are the detailer of the design, but others will remark on your choices when they have to read your product.

Modifying the Drawing Units



When you observe the units on the status bar, the system default is a four decimal place number and the angular precision shows figures rounded to a whole numbers. Type “Units” and ENTER at the command line to modify these adjustable units in the AutoCAD drawing. You may want to change the Angle Precision to 0.0. Many of you will use this dialogue box to change from decimal to architectural units in another template.

Figure 8.18 – Setting Drawing Units

Setting the Drawing Limits

Type “Limits” at the command line to set the drawing limits. You will be prompted to “*Specify lower left corner or [ON/OFF], 0.0000, 0.0000>:*”. Hit ENTER to retain the 0,0 lower left limit, but when the second statement “*Specify upper right corner <12.0000,9.0000>:*” appears, type 100,60 to create a larger field on the graphical display (Figure 8.19).

```
..... Command: limits
Reset Model space limits:
Specify lower left corner or [ON/OFF] <0.0000,0.0000>:
Specify upper right corner <12.0000,9.0000>: 100,60
Command:
```

Figure 8.19 – Setting the Drawing Limits at the Command Line

Modifying the Linetype Scale

The Linetype scale is the last system variable that you will change in the Mechanical template. At the Command Line, type “ltscale” and hit ENTER. At the prompt “*Enter new linetype scale factor <1.0000>:*” type 4. You will be able to see your hidden, section and centerlines better when the drawing field is larger (Figure 8.20).

```
..... Command: LTSCALE
Enter new linetype scale factor <1.0000>: 4
Regenerating model.
Command:
```

Figure 8.20 – Modifying the Linetype Scale at the Command Line

Saving a Drawing Template

To save your new template, select Save Drawing As from the quick access toolbar and the Save Drawing As window will appear as shown in Figure 8.21. Save the template in your 2D drawing folder.

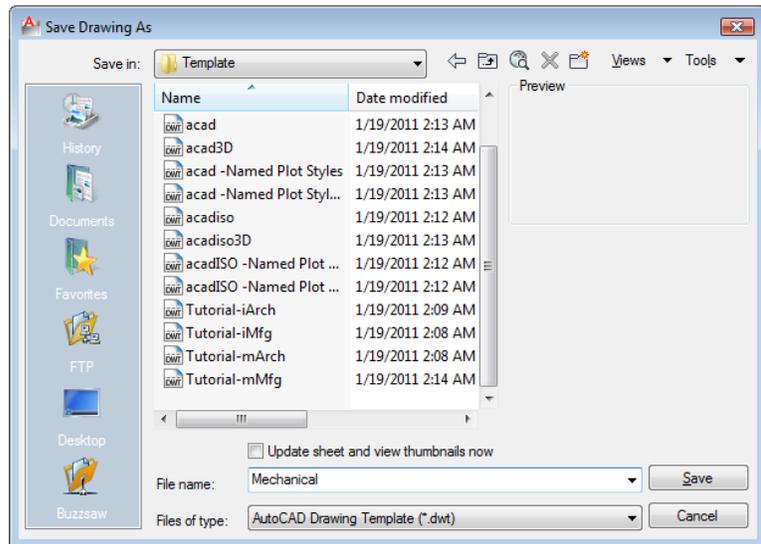


Figure 8.21 – Saving the Mechanical Template

If you are using the same computer every day, save the file in the AutoCAD template folder, type the file name “Mechanical” and choose the File Type to be AutoCAD Drawing Template File (*.dwt) as shown in Figure 8.21. After clicking the Save button, a Template Description window will appear for you to type the Description, “Mechanical Drawing Template” and maintain the English measurement system.

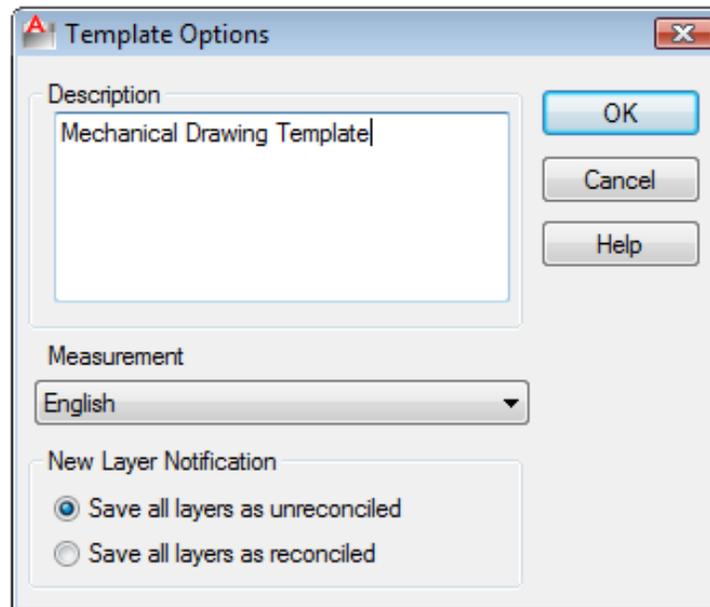


Figure 8.22 – Creating a Template Description

To make any changes to the template, open the template (dwt) in AutoCAD, make the changes, and save your work again as a template. If you do not update your basic drawing templates, then you will forever be altering drawings manually. Your next step in the World Class CAD training is to make an Architectural and a Metric template. You may continue with additional patterns for Civil and Electrical Engineering for future projects.

*** World Class CAD Challenge 02-17 * - This task is not timed but we do challenge you to test your Mechanical template on some sample drawings to discover whether you would be required to modify any system variables or create a layer that is forgotten. Continue to test your templates, each time you can write down your omission and open the template, make the change to maintain your World Class status.**