

Making a Drawing Template

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

- 1. Starting from Scratch for the Last time**
- 2. Creating New Layers in an AutoCAD Drawing**
- 3. Setting Your Object Snaps**
- 4. Modifying Text Styles**
- 5. Creating a Mechanical Dimension Style**
- 6. Modifying the Drawing Units**
- 7. Setting the Drawing Limits**
- 8. Modifying the Linetype Scale**
- 9. 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 any other system variable changes to make the drawing clear to read. In this chapter, you will again repeat all of the alterations, but now those settings will be saved as a mechanical template, so in the future, you will benefit from the work done today. A mechanical template is not the only type of drawing model required by designers, engineers and architects, but architectural, civil, electrical and metric templates are useful when creating all sorts of various types of prints. To create a template, you need to start the drawing as usual.

Starting from Scratch for the Last Time

Pick the New Drawing tool on the Standard toolbar and the Startup window will appear. Decide on the English (feet and inches) option since this will use the conventional measurement used in the United States (See Figure 8.1). The metric choice will not just align your system variable for the metric system, but the decimal numbers such as 1.500 will be displayed as 1,500 since the comma is used as a decimal point in the metric system. If you are building a mechanical or metric template outside the United States, then you would pick the metric radial button.

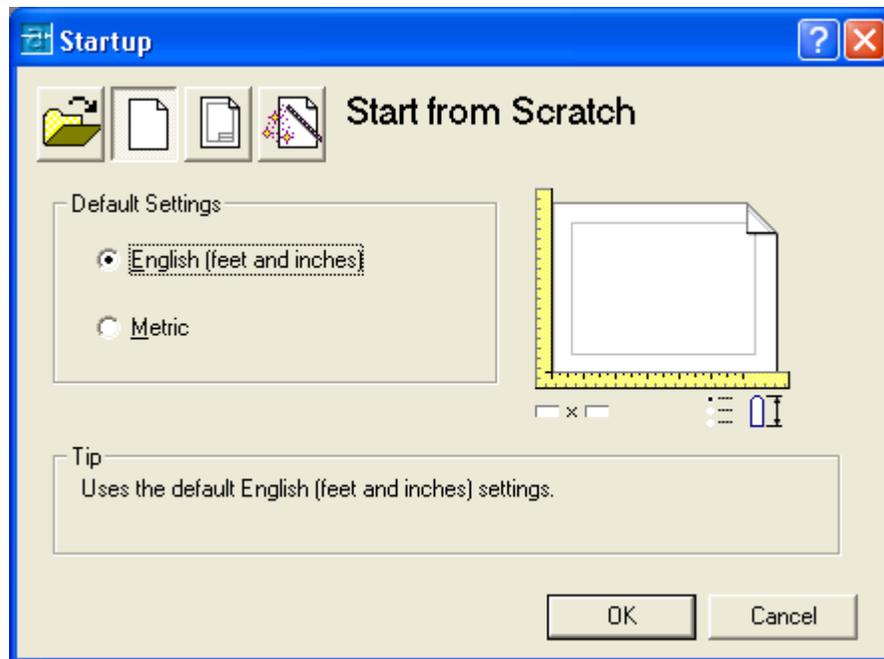


Figure 8.1 – Starting from Scratch

This is the AutoCAD traditional Start from Scratch window. Newer releases of the 2000+ AutoCAD software will start with the Show Today Startup Dialogue which is an Internet interface to Autodesk, the software developer of AutoCAD. Many companies choose not to use the newer startup window, but this strictly a personal preference. As in most software that is now around twenty years in development, there are several different approaches resulting in the same conclusion

Creating New Layers in an AutoCAD Drawing



Figure 8.2 - The Object Properties toolbar

When expanding the orthographic drawing and adding more than one view, like a top and front view, new linetypes such as hidden, section and hatching lines will be needed to explain the views to the machinist or construction worker. You will also necessitate new layers for the drawing border and possible hatching for cross sectional views. To create new layers using the Layers Properties Manager, select the Layers command button to the left of the Layers Control list box (See Figure 8.2). Add the following layers shown in Table 8.1 using the New button in the Layer Properties Manager.

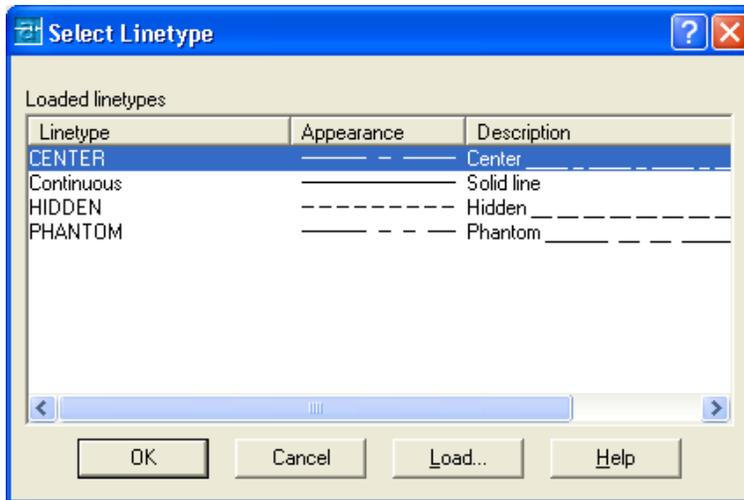


Layer Name	Color	Linetype
Border	8 (dk. gray)	Continuous
Center	Green	Center
Dimension	Red	Continuous
Hatch	254 (lt. gray)	Continuous
Hidden	Blue	Hidden
Section	Magenta	Phantom
Text	Cyan	Continuous

Table 8.1 - Mechanical Template Layers

To create this template, you need make seven new layers. To create the layers, select the New button to add a new line ready for editing. Immediately, the layer name is ready for editing, so type, “Border”, and ENTER twice to add each new layer. Continue to add the new layers until you type the Text layer name. 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 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) on the Standard Colors Row from the Standards Colors row and select the OK button. You will now see that Border layer is 8 (gray). Repeat the process until you select the Cyan color for the Text layer.

To finish creating and modifying layers, you need to change the Continuous linetype to a Center linetype on the Center layer, to Hidden linetype on the Hidden layer and the Phantom linetype on the Section 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 8.3). 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 and select the OK button. Repeat the process for the Hidden and Phantom linetypes so that four different linetypes are not shown in the Select Linetypes window.



Now, highlight the CENTER linetype in the Select Linetype window and select OK. You will now see that the Center layer has a CENTER linetype (Figure 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. Do the same steps for the Section layer to load the Phantom linetype.

Figure 8.3 - The Select Linetype Window

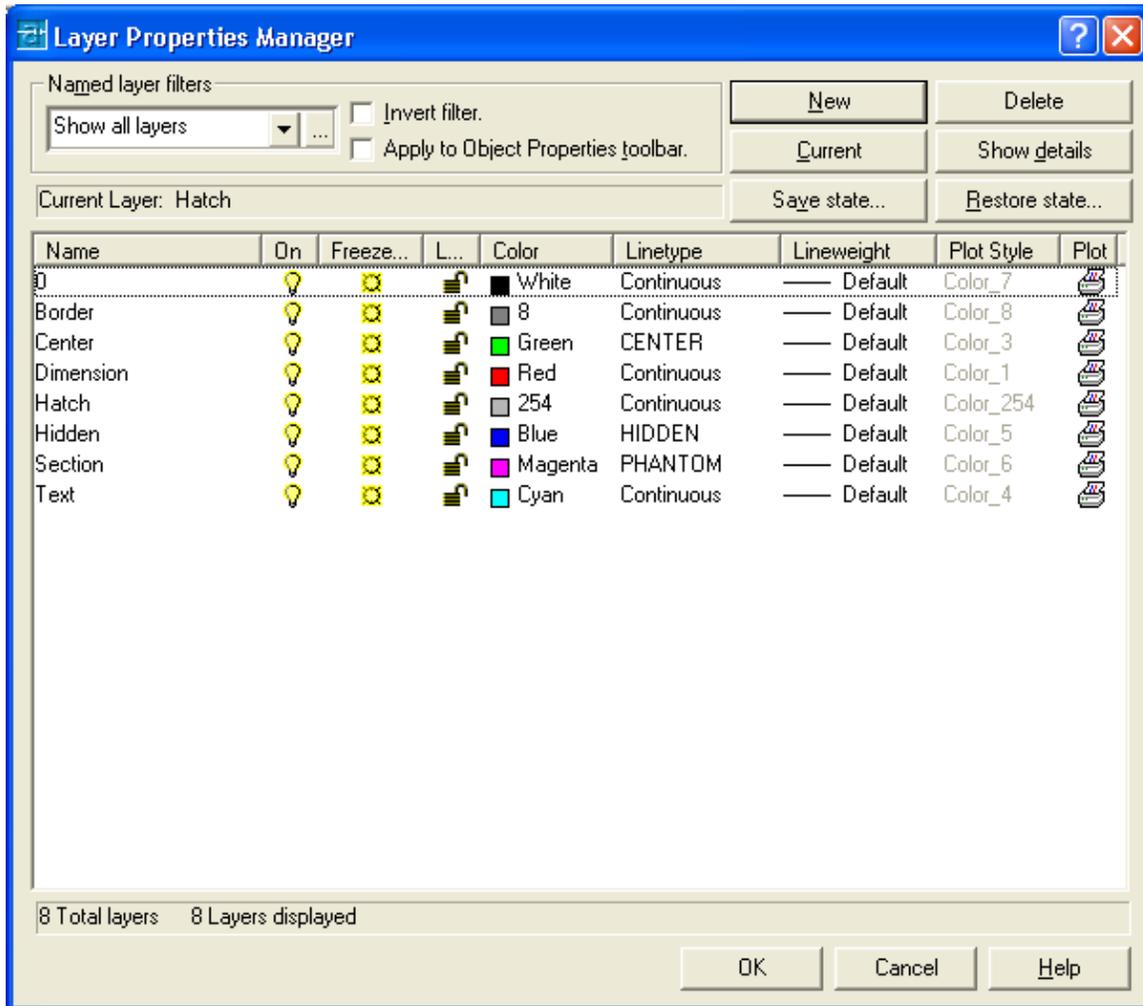


Figure 8.4 - The Layer Properties Manager window

Setting your Object Snaps

Setting Object Snaps is required to allow you to select precisely the point desired. Pick the Object Snap Setting tool on the Object Snap toolbar (the magnet). In the Mechanical template, you need to select the Object Snap Mode Center for grabbing the center point of a circle, recognized by a blue circle when in the zone (Figure 8.5). You also need to choose Endpoint, Midpoint, Quadrant, Intersection and Perpendicular.

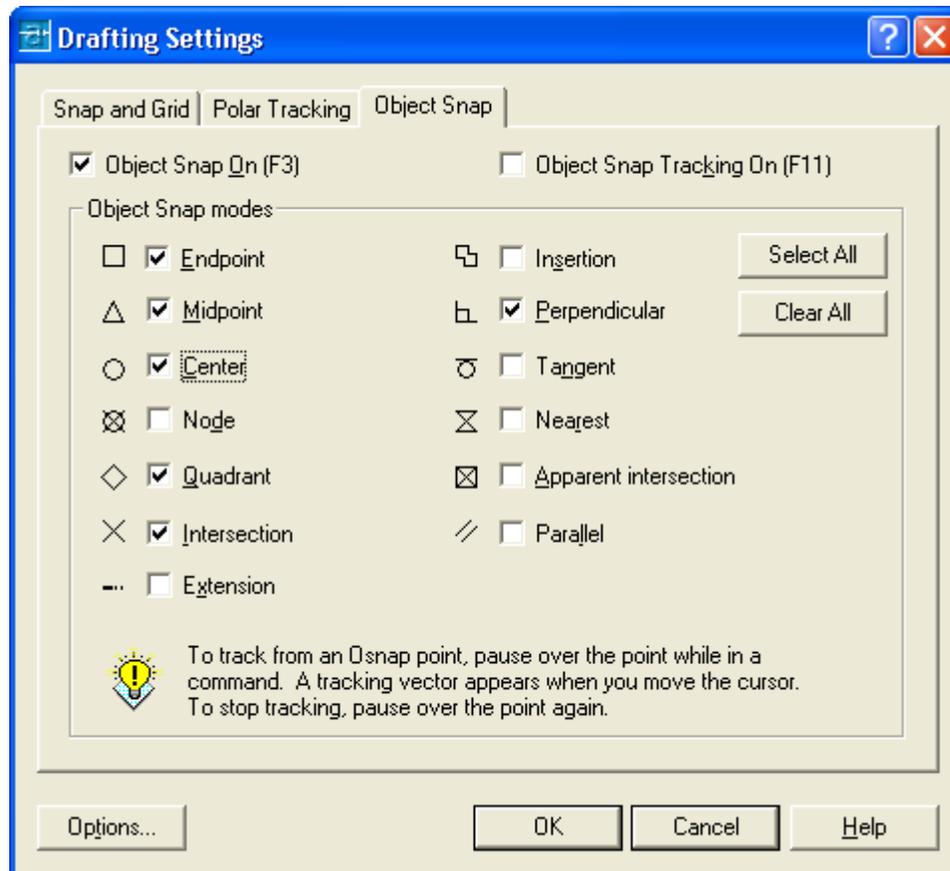


Figure 8.5 – Object Snap Settings

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 always go on the premise not to change the AutoCAD system defaults but make a new Text Style for the Mechanical template which for the principle of ease, name the Text Style “Mechanical”. Select the Text Style tool on the Text toolbar to open the Text Style window.



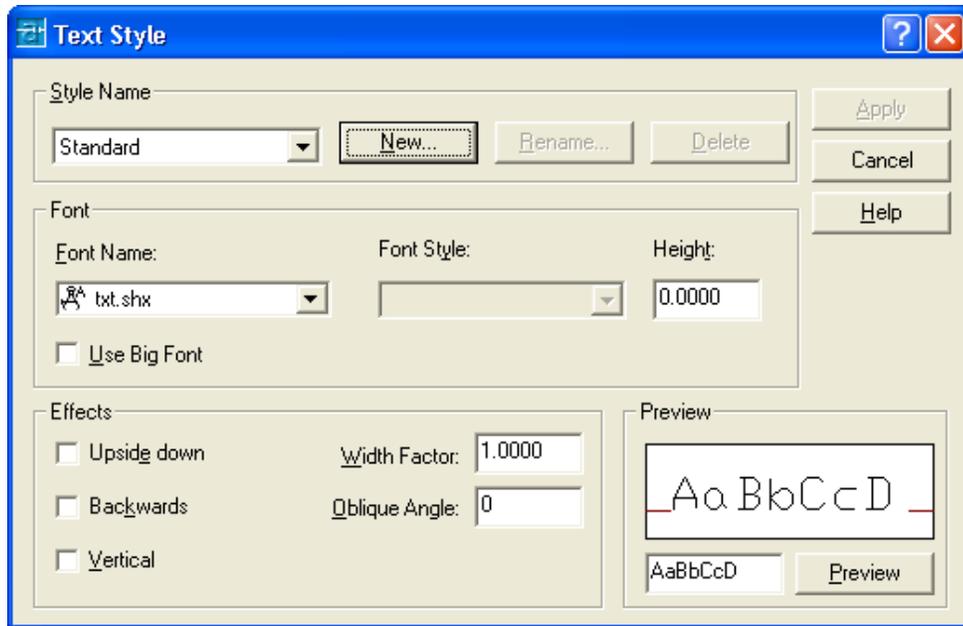


Figure 8.6 – Text Style Window

As you can see in figure 8.6, that the system is set to Standard Style and the Font is “txt”. In the Preview window, you can see that the Txt font is edgy and may not meet the visual requirements of your corporation. The Height is 0.0000, which means that whatever the setting the System Variable Textsize is registered will be the size of the text when using Mtext. 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 create a new text style, select the New button in the Text Style Window and the New Text Style dialogue box will appear as shown in Figure 8.7. Type the new Style Name as mechanical and press the OK button to create and set this style as current. To change the font for the new Mechanical Text Style, scroll through the list of Font Names and select Arial. You will see the smoother text style appear in the Preview window. (See Figure 8.8) The last step in the text process is to change the System Variable named Textsize which is set at 0.2000. At the command line (see Figure 8.9), type Textsize and Enter. Now type 0.125 to change the default text height. You can create more text styles for different purposes, following the same procedure. Visually, if your organization chooses an edgy font stay with other sharp framed fonts. If your group selects a smooth font, your additional text styles should have similar edges. Furthermore, smooth fonts print nicer than edgy fonts and are easier to read.

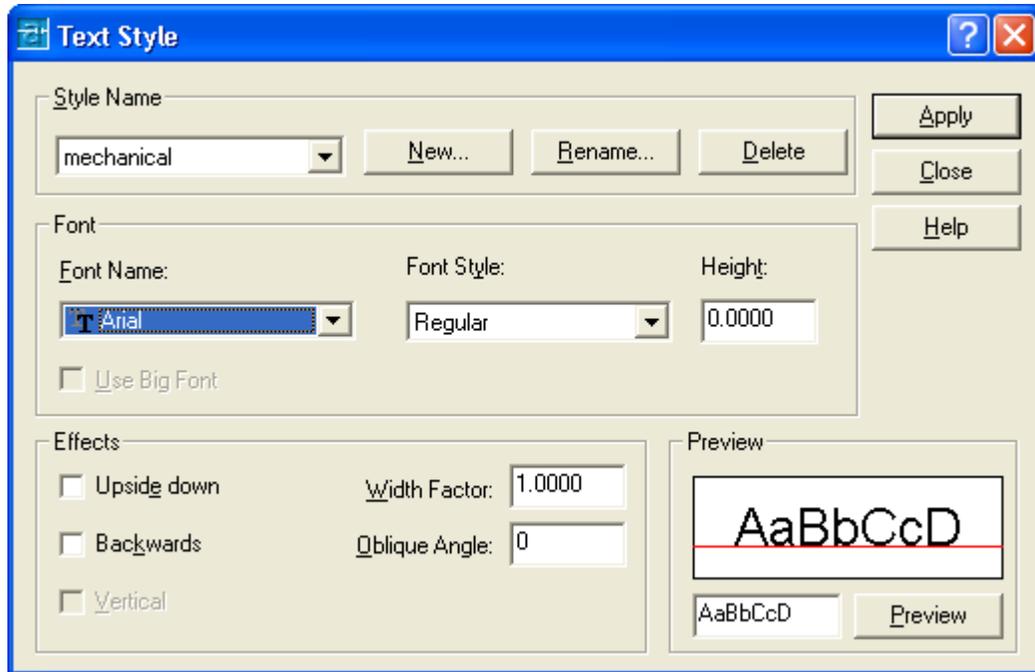


Figure 8.8 – Selecting a Font with the Text Style Window

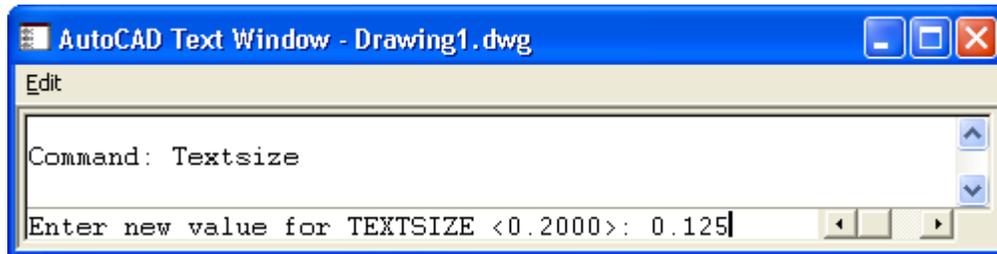


Figure 8.9 – Setting the Text Size Variable at the Command Line

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 Dimension Style tool on the Dimension toolbar. (See Figure 8.10) There is 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 have modified the Standard Style using the Modify button, but today you will select the New button to make the Mechanical Dimension Style.



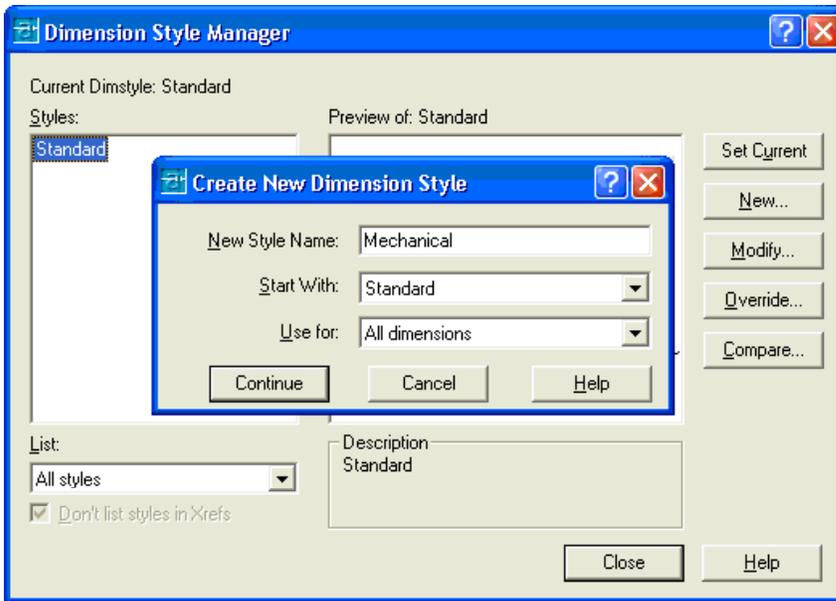


Figure 8.10 – Dimension Style Manager

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

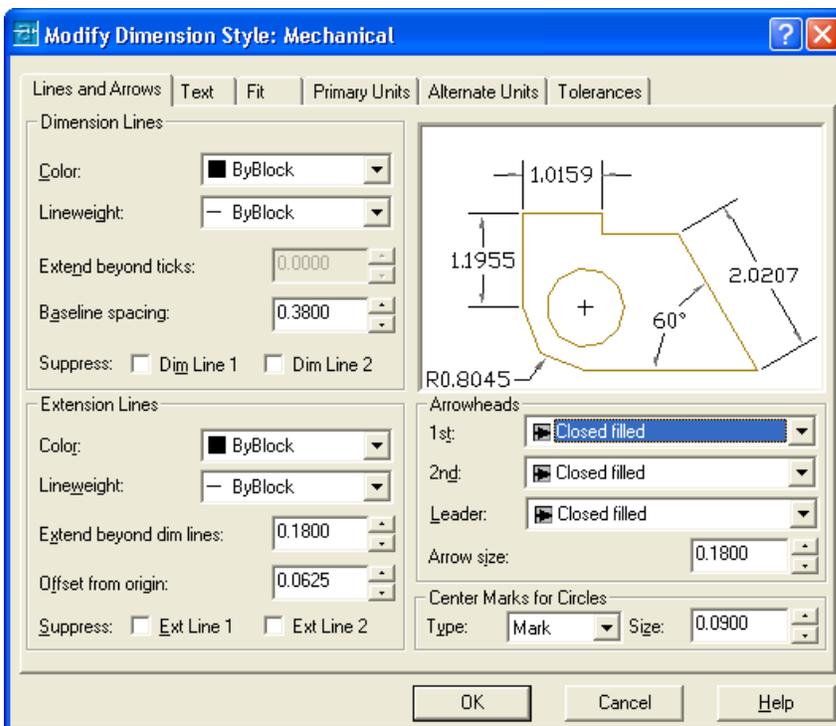


Figure 8.11 – Modify Dimension Style – Lines and Arrows

To control the Standard dimension style which you will be using to dimension the Circular problem, select the Dimension Style tool on the Dimension toolbar and the Dimension Style Manager will open as shown in Figure 8.10. Select the Modify button to make changes to the Standard style. The Lines and Arrows tab shown in Figure 8.11 allows you to alter every part of the Dimension Lines, Extension Lines, Arrowheads and Center Marks.

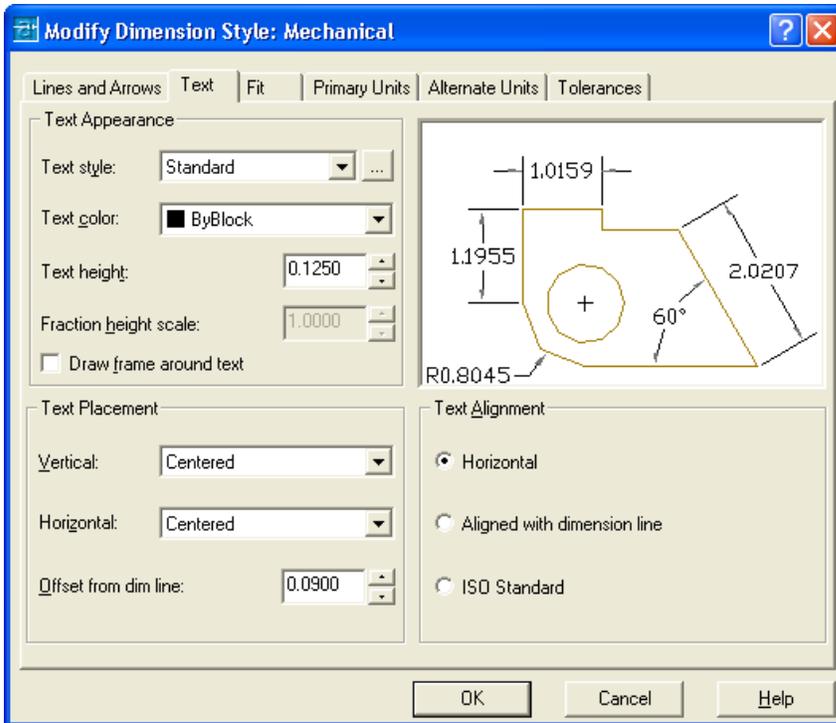


Figure 8.12 – Modify Dimension Style – Text

Proceeding to the next Text tab, you can choose the Text Appearance, Text Placement and Text Alignment. Many of the establishments where you will 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 the print and allows the dimension to remain close to the feature in which is being described. 3/32 is as small of text to be printed and is still able to be interpreted. You will change the Text Height to 0.1250.

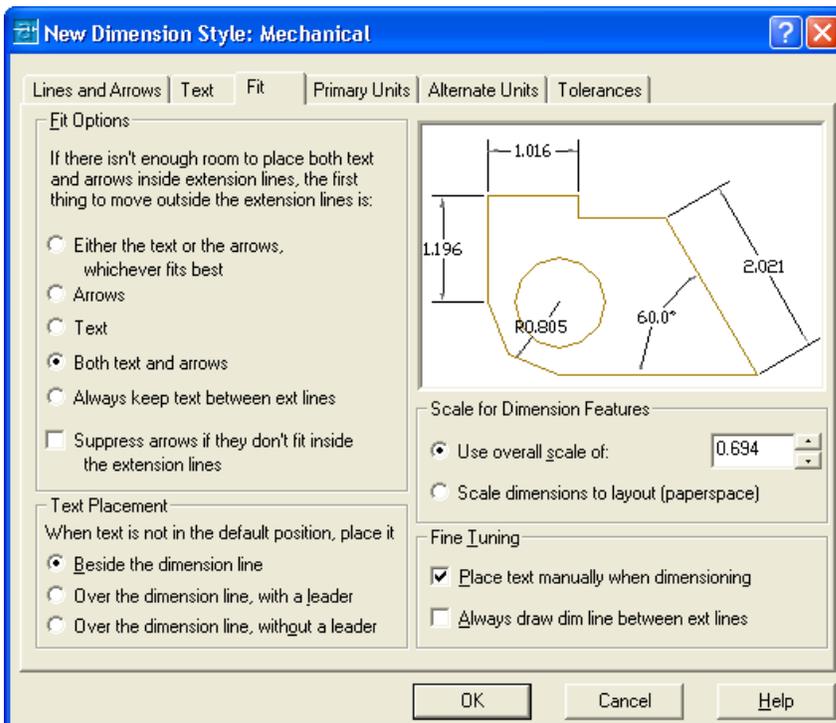


Figure 8.13 – Modify Dimension Style – Fit

The Fit tab allows you to adjust the dimension variables to compensate for room issues. You will make a couple of changes on this tab. First select the radial button for “Both Text and Arrows”. Choosing this option button, the Diameters will be flanked by regular dimension lines instead of using 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. (See Figure 8.13)

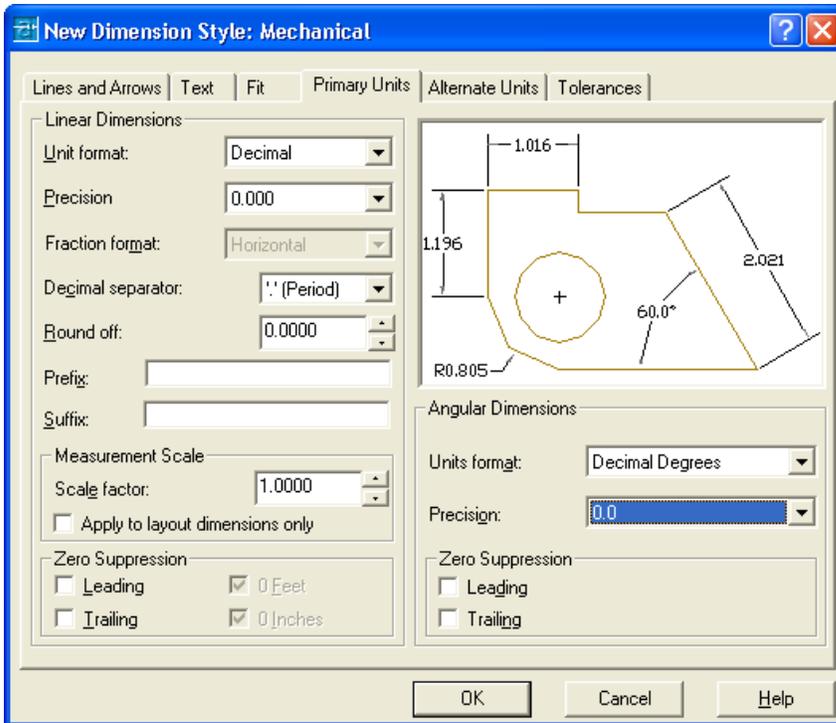


Figure 8.14 – Modify Dimension Style – Primary Units

In the Primary Units tab, the Unit format will remain Decimal, but the Precision setting will change to 0.000 as shown in Figure 8.14. The other Linear Dimension settings are used periodically, such as Round Off, but the 1.0000 Scale Factor in the Measurement Scale should stay always at one since confusing this setting with other scale factors like line or text scale will change the value of all the dimensions in the AutoCAD drawing.

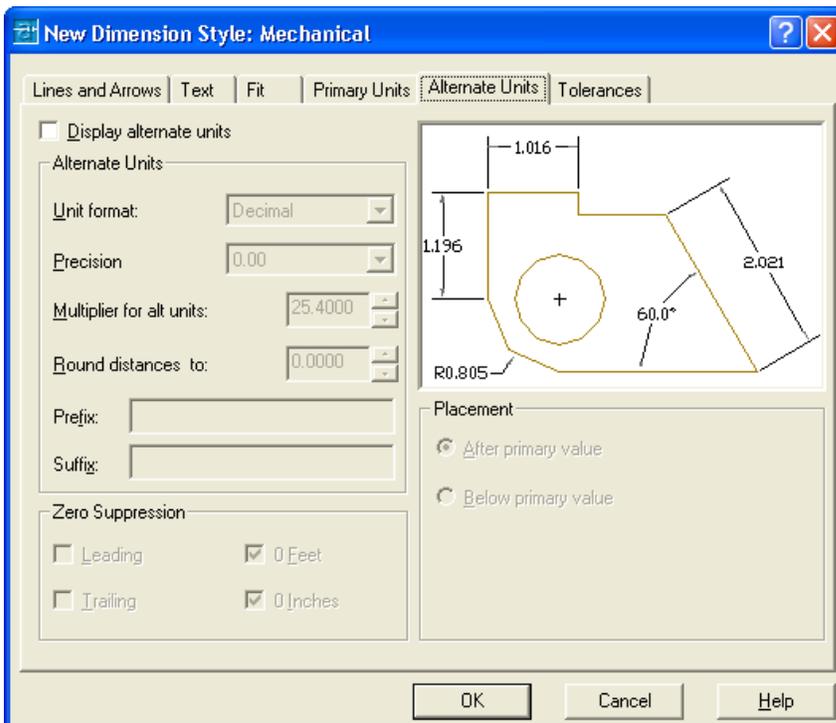
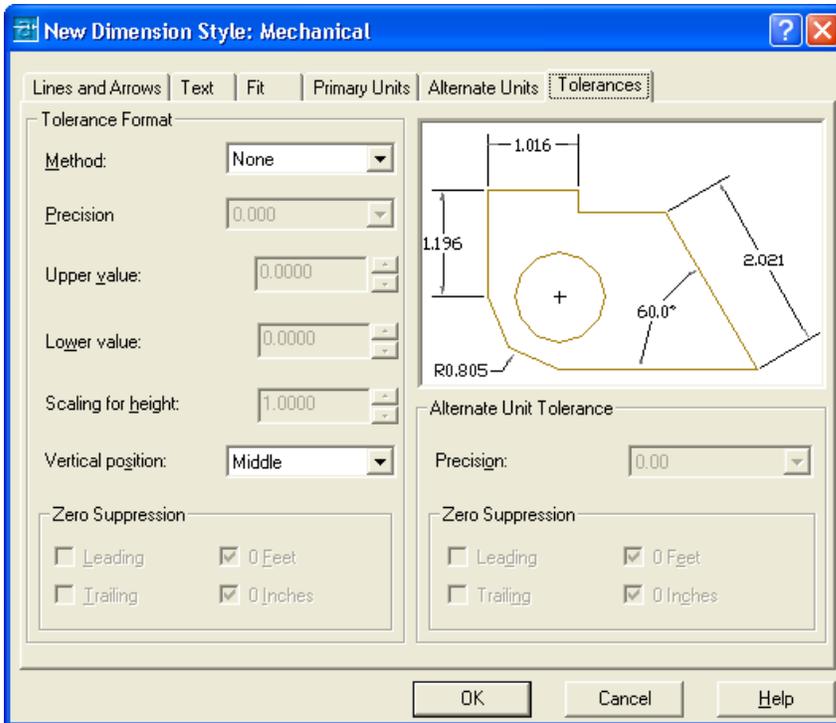


Figure 8.15 – Modify Dimension Style – Alternate Units

The Angular Dimensions are to be Decimal Degree and set to 0.0 precision to allow the reader of the print to see the angles in at least a tenth of a degree.

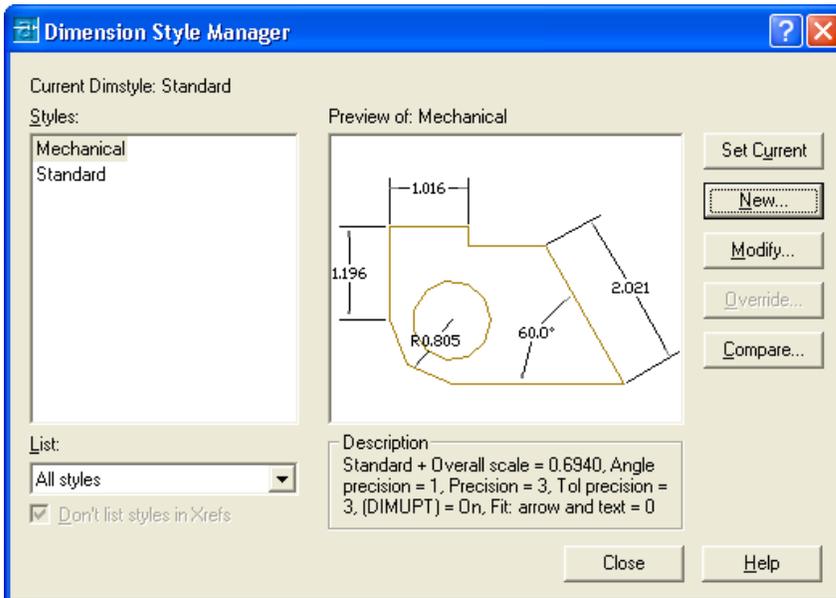
The Alternate Units tab would be used when you want to list the metric value of the English unit below or along side the dimension text. For example, if you checked “Display alternate units”, a dimension showing one inch will have the (25.40) alternate dimension in parenthesis below the 1.000 text.

You do not need to check the “Display alternate units” checkbox on the Mechanical template.



The Tolerance tab is an option mostly used by a Mechanical Designer or Engineer, where 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 will be typed in every drawing you do. As the parts become more complex, this feature needs to be utilized to correctly show the detail dimensions. (Figure 8.16)

Figure 8.16 – Modify Dimension Style – Tolerances



To save the change that you made in the Mechanical Dimension Style window, select the OK button. Now 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 (see Figure 8.17) to finish the procedure.

Figure 8.17 – Modify Dimension Style – Tolerances

In the other templates that you will want to create, arrow heads will change to architectural ticks, the Unit Format will change 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 their drawing. 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

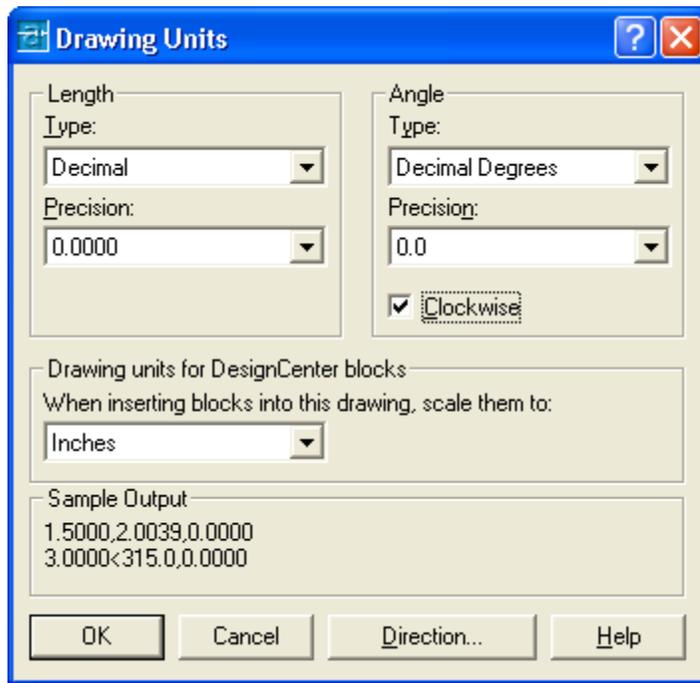


Figure 8.18 – Setting Drawing Units

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. On the pull down menu, select Format, then Drawing Units to modify these adjustable units in the AutoCAD drawing. You may want to change the Decimal Degrees to 0.0. Many of you will use this dialogue box to change from decimal to architectural units in another template.

Setting the Drawing Limits

In the Format menu, you will see the option to select the Drawing Limits. When selected, at the Command line, the remark “Specify lower left corner or [ON/OFF], 0.0000, 0.0000>:” will appear. Enter to retain the 0,0 lower left limit, but when the second statement “Specify upper right corner <12.0000,9.0000>:” is presented, type 100,60 to make a larger field on the graphical display when the drawing begins. (See Figure 8.19)

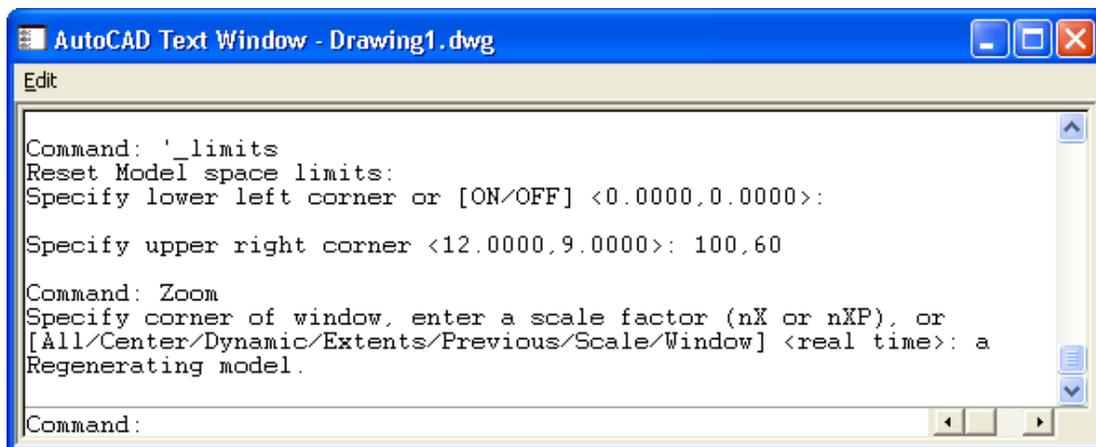


Figure 8.19 – Setting the Drawing Limits at the Command Line

Modifying the Linetype Scale

The Linetype scale is the last system variable that you will change in the Mechanical template. At the Command Line, key “ltscale” and Enter and 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. (See Figure 8.20)

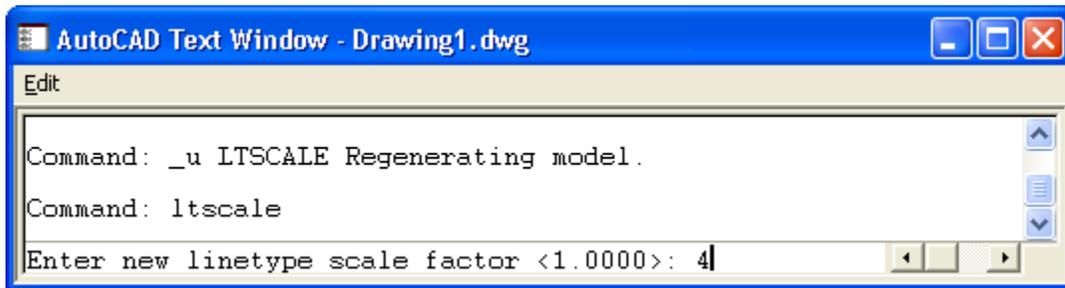


Figure 8.20 – Modifying the Linetype Scale at the Command Line

Saving a Drawing Template

To save your new template, select Save Drawing As and the Save Drawing As window will appear as shown in figure 8.21. You can save the template anywhere such as a floppy drive, server drive or in the AutoCAD template folder.

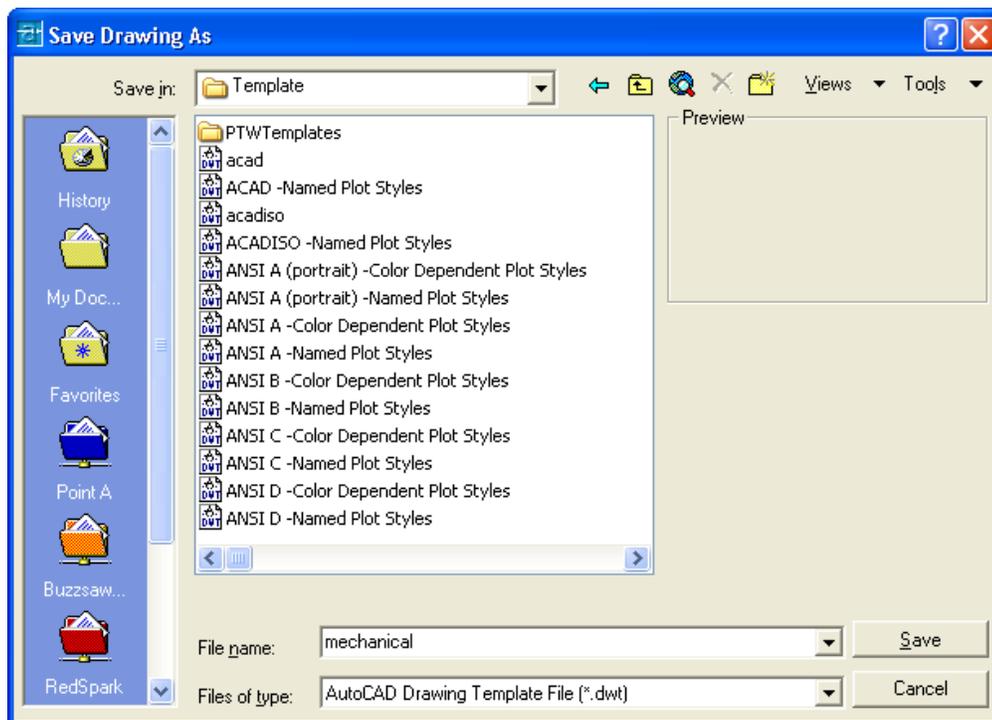


Figure 8.21 – Saving the Mechanical Template

If you are using the same computer everyday, go ahead and 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 depressing the Save button, a Template Description window will emerge for you to type the Description “Mechanical Drawing Template” and maintain the English measurement system. Always pick the English measurement if you are working on drawings where you desire non-European looking dimensions.

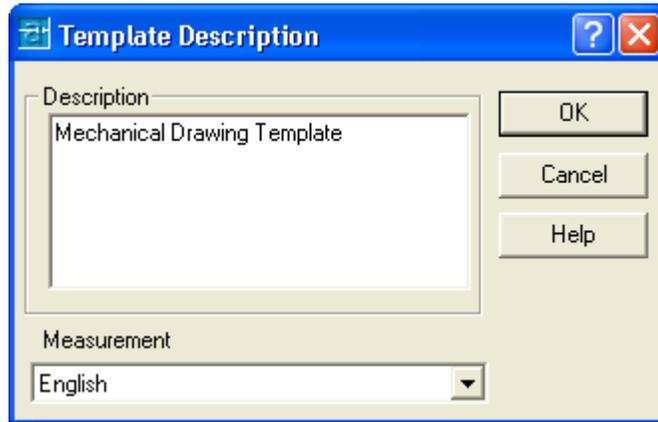


Figure 8.22 – Creating a Template Description

To make any changes to the template if an omission is discovered, open the template (dwt) in AutoCAD, make the changes, and save your work. If you do not update your basic guiding tools, then you will forever be altering the drawings manually. Your next step in the World Class CAD training package 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 * - 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.**