C h a p t e r

# 8

# Making a Drawing Template

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

1. **Starting from Scratch**
2. **Creating New Layers in an AutoCAD Drawing**
3. **Setting Your Entity Snaps**
4. **Modifying Text Styles**
5. **Creating a Mechanical Dimension Style**
6. **Modifying the Drawing Units**
7. **Setting the Drawing Limits**
8. **Saving a Drawing Template**

In the first two units, the Rectangular and the Circular problem, you created layers, set Entity 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 at the end of the lesson those settings will be saved as a mechanical template, so in the future you won't have to change 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 all sorts of various types of prints. 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 Standard toolbar and the Start window will appear. Select the Start from Scratch option, with English selected in the Default Settings area since this will use the conventional measurement used in the United States (Figure 8.1). If you are building a mechanical or metric template outside the United States, then you would pick the metric radial button.
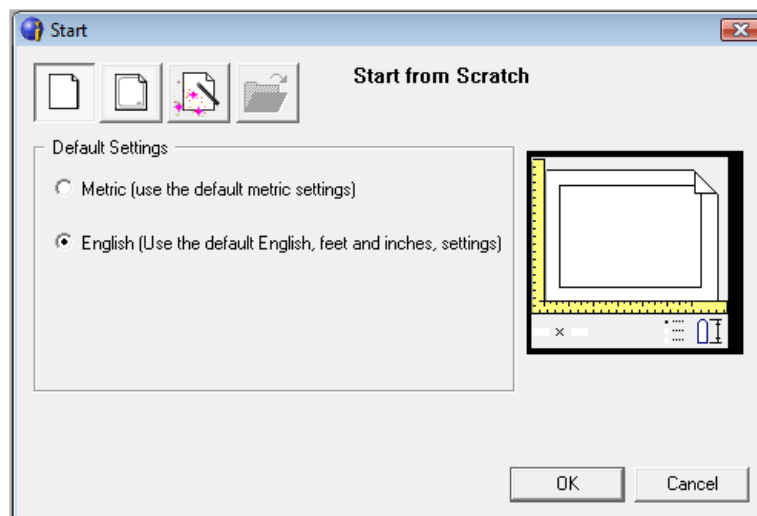


**Figure 8.1 – The Start window**

## Creating New Layers in an AutoCAD Drawing



**Figure 8.2 - The Layers section of the Entity 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 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 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 choose different colors to help distinguish between the 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 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 linetype to a Center linetype on the Center layer, Hidden linetype on the Hidden layer, and the Phantom linetype on the Section layer.  Place your mouse arrow over "Continuous" in the Center layer row and single left click with your mouse.  The Linetype window will appear in front of the Layers Properties Manager (Figure 8.3).  Scroll down, highlight the CENTER2 linetype and select the OK button.  Repeat the process for the HIDDEN2 and PHANTOM2 linetypes so that the linetypes appear as in Figure 8.4.
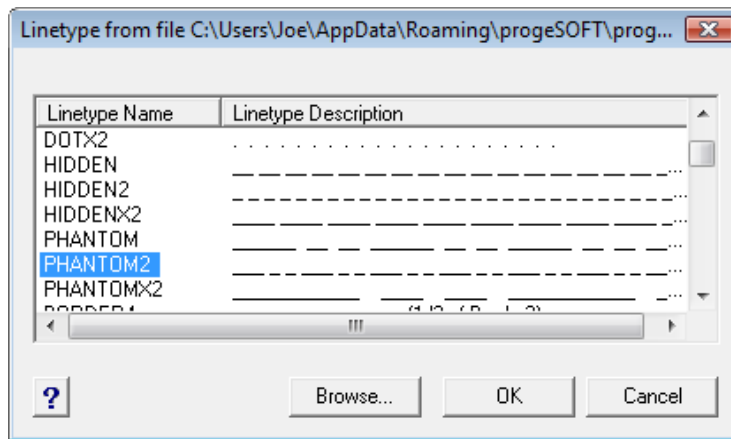


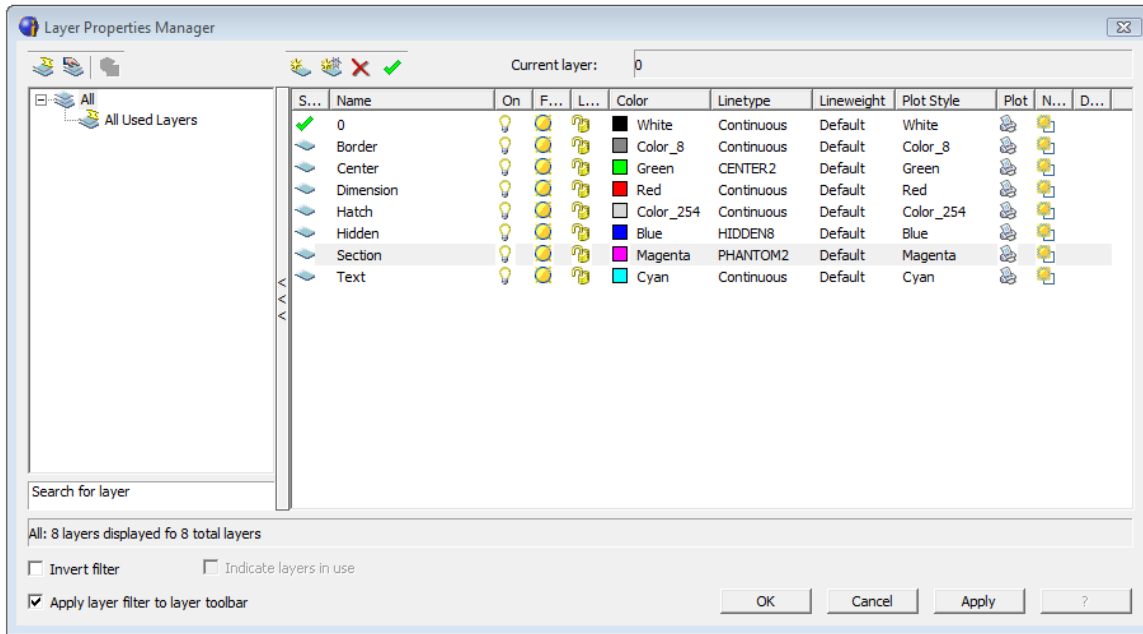**Figure 8.3  -  The Select Linetype Window**

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**Figure 8.4  - The Layer Properties Manager window**

## Setting your Entity Snaps

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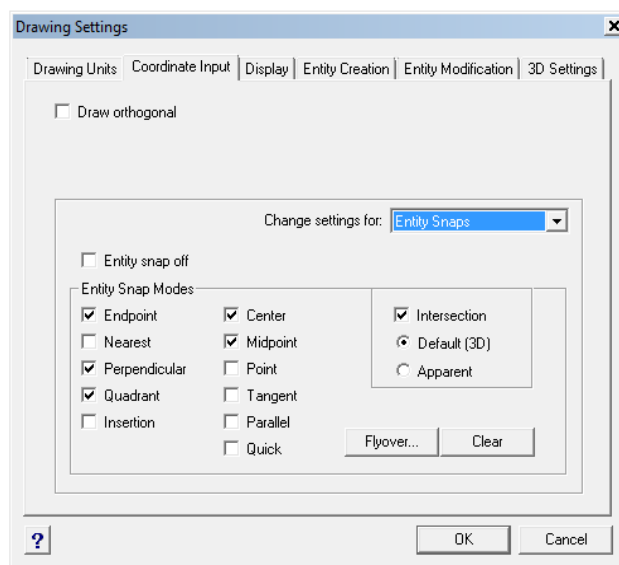


**Figure 8.5 – Entity Snap Settings**

# Modifying the Text Style

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Many companies or organizations will want you to change the font that will be used in their drawings. You probably should not change the progeCAD system defaults, so instead we will make a new Text Style for the Mechanical template. For the principle of ease, name the Text Style "Mechanical."  Select the Explore Text Styles tool on the Text toolbar to open the Text Style window.
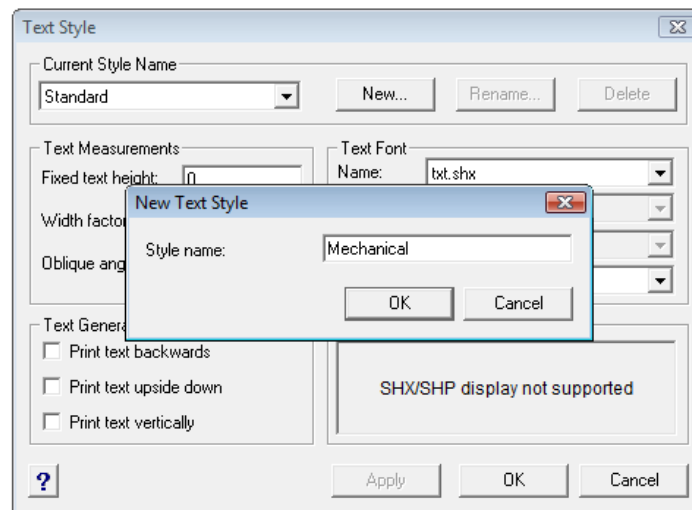


**Figure 8.6 – Naming a New Text Style**

As you can see in Figure 8.7, the system is set to Standard Style and the Font is "txt." The Fixed Text Height is 0.0000, which means that whatever the setting the System Variable Text size is registered will be the size of the text when using Mtext.  We want to change the fixed text height to 0.125:  this will eliminate line spacing issues in your notes.   Notice there are check boxes for Upside down, Backwards and Vertical text if you so desire.
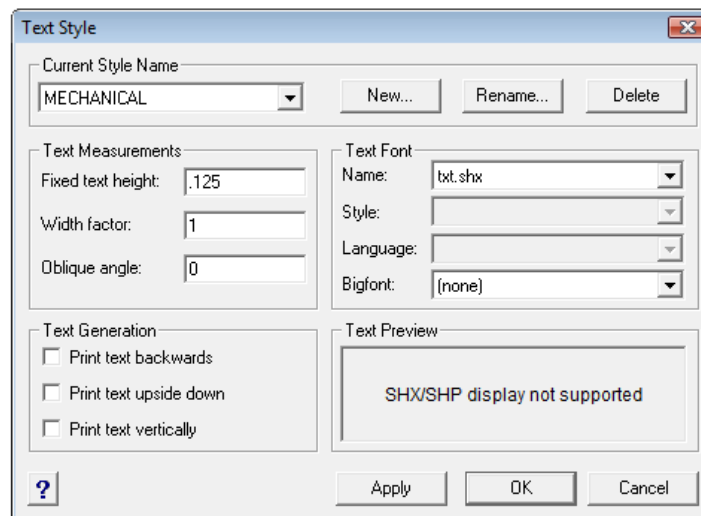


**Figure 8.7 – Changing the Fixed text height to 0.125**

# Creating a Mechanical Dimension Style

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In your previous problems and practice exercises, you opened the Dimension Style window and modified the dimension variables on every drawing. As with Layers, Entity Snaps and Text Styles, you want to make a custom variation using the Standard Dimension Style as the starting point. Open the Dimension Settings window by selecting the Dimension Style tool on the Dimension toolbar (Figure 8.8).
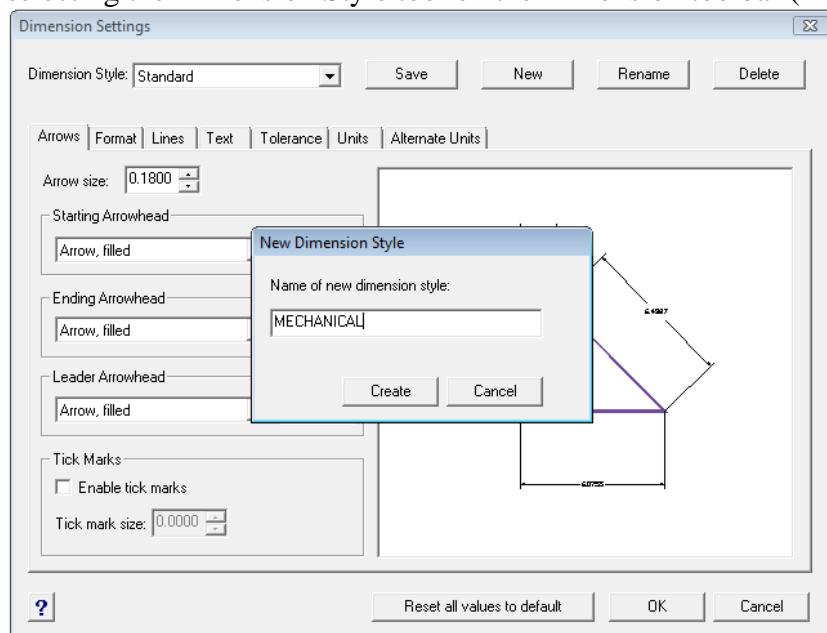


Above the tabs there is a drop menu where you can select a Dimension Style. We want to hit New, and then name the dimension style Mechanical (Figure 8.8). Hit Create to save the new dimension style, and then make sure it is selected in the Dimension Style drop box so we can make changes to it.

**Figure 8.8 – Dimension Settings – Creating a New Dimension Style**
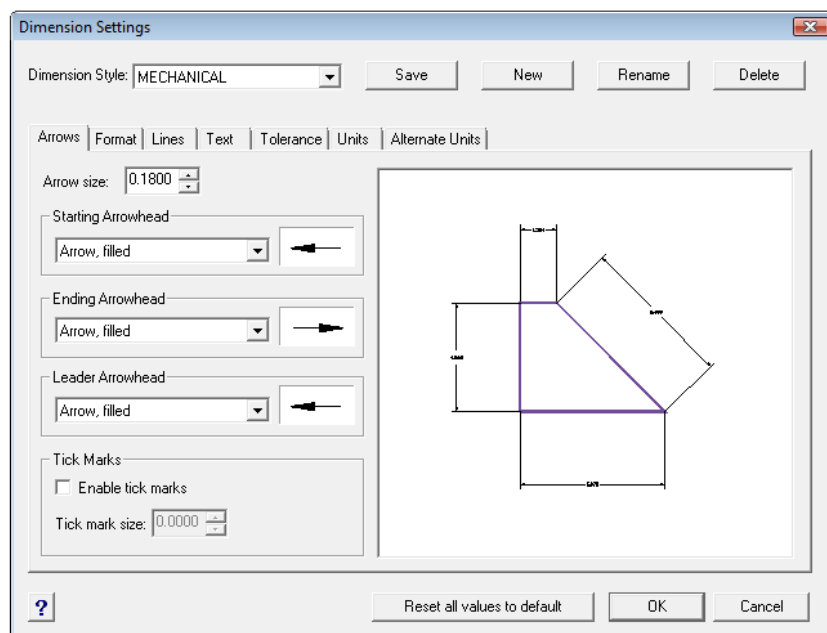


The first tab in the Dimension Settings window is the Arrows tab (Figure 8.9). Here we can make changes to the symbols at the ends of our dimension lines or change them to the architectural tick marks. We won't make any changes to these settings.

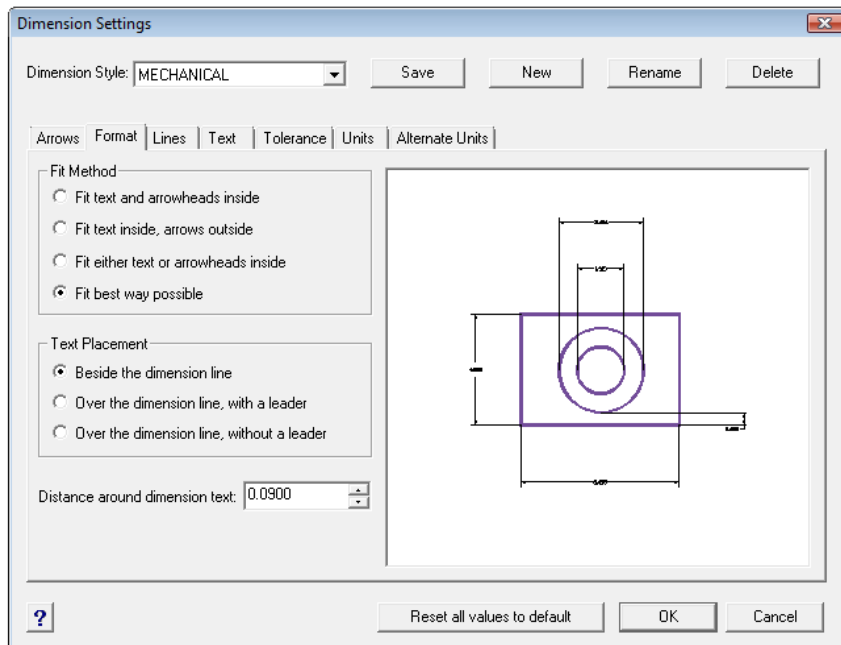**Figure 8.9 – Modifying the Dimension Style – Arrows**

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**Figure 8.10 – Modify Dimension Style – Text**

Proceeding to the Format tab, you can choose a new Fit Method or Text Placement (Figure 8.10). We won't modify any of these settings, either.
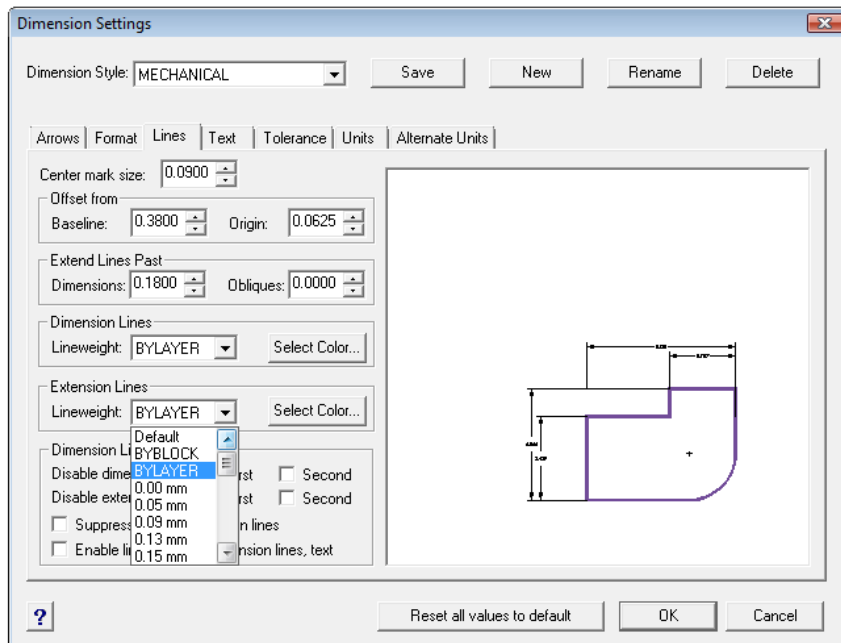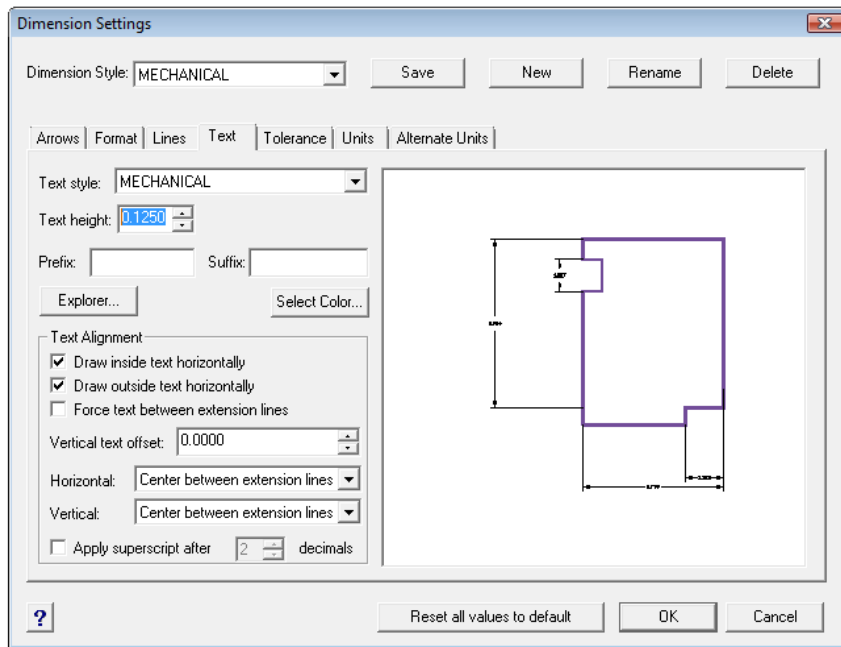


**Figure 8.11 – Modify Dimension Style – Fit**

The Lines tab allows you to modify the settings of Extension and Dimension lines (Figure 8.11). Under the Dimension Lines and Extension Lines section change the Lineweights to BYLAYER. Then you will be able to modify all lineweights through the Layer Properties manager.

Under the Text tab you can choose the text appearance 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. Change the Text style to the Mechanical style you set up earlier, and set the Text height to 0.1250 (Figure 8.12).

**Figure 8.12 – Modify Dimension Style – Primary Units**
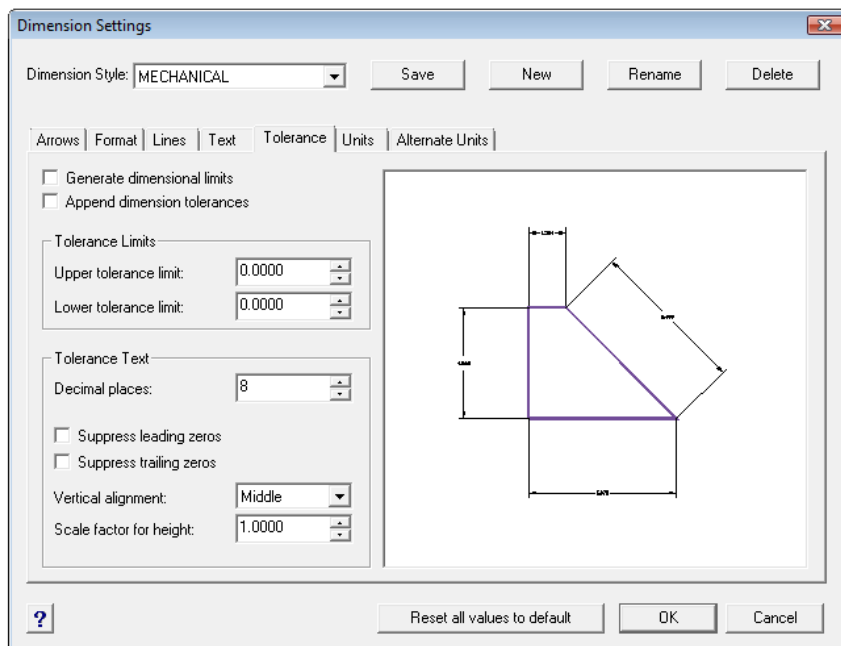


The Tolerance tab is an option mostly used by a Mechanical Designer or Engineer when they need to discuss the range of allowable error in the part when manufacturing or accepting the components at the production site. Right now this is being addressed in the notes that will be typed in every drawing you do. As the parts become more complex, this feature will be utilized to correctly show the detail dimensions (Figure 8.13).

**Figure 8.13 – Modify Dimension Style – Alternate Units**

In the Units tab, the Linear Dimension format will remain Decimal, but the Decimal places setting will change to 3 as shown in Figure 8.14. The other Linear Dimension settings are used periodically, such as Round linear distances, but the 1.0000 Scale Factor in the Measurement Scale should stay always at 1. Confusing this setting with other scale factors like line or text scale will change the value of all the dimensions in the progeCAD drawing.
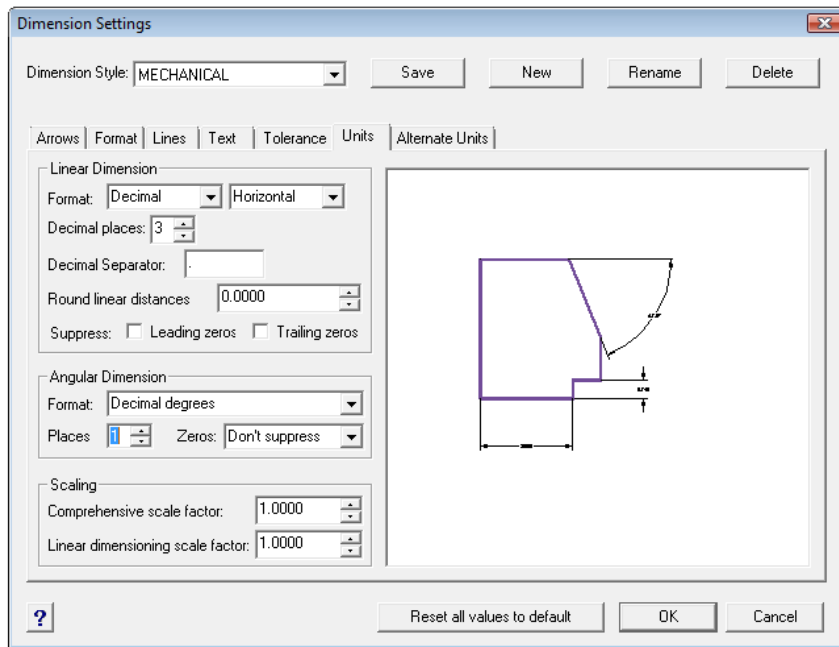
**Figure 8.14 – Modify Dimension Style – Tolerances**

The Angular Dimensions are to be Decimal Degree and set to 1 Places to allow the reader of the print to see the angles in at least a tenth of a degree (Figure 8.14).
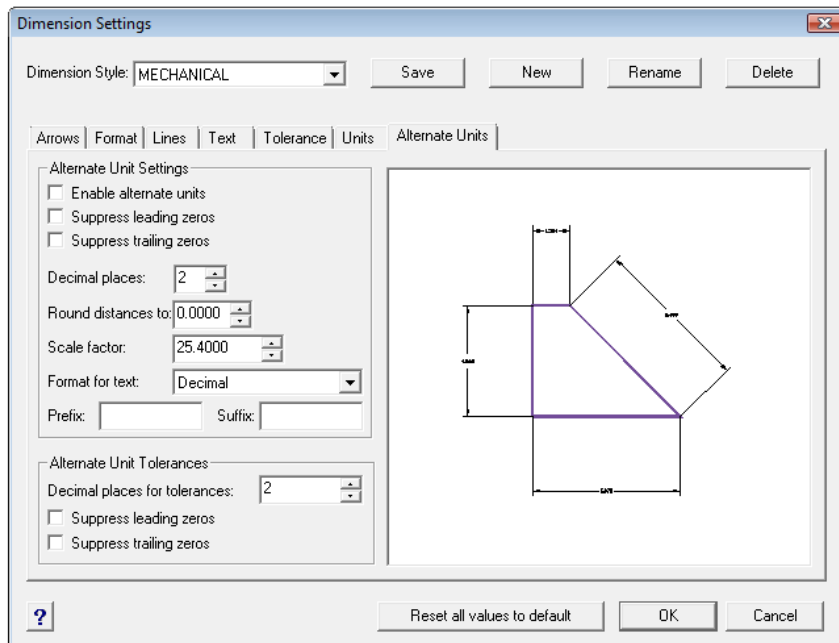


The Alternate Units tab would be used when you want to list the metric value of the English unit below or alongside the dimension text. For example, if you checked "Enable alternate units," a dimension showing one inch will have the (25.40) alternate dimension in parenthesis below the 1.000 text.

To save the changes that you made in the Mechanical Dimension Style window, select the Save button. Now press the OK button to finish the procedure (Figure 8.15).

**Figure 8.15 – Modify Dimension Style – Tolerances**

In the other templates that you will create, arrowheads will change to architectural ticks, and 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

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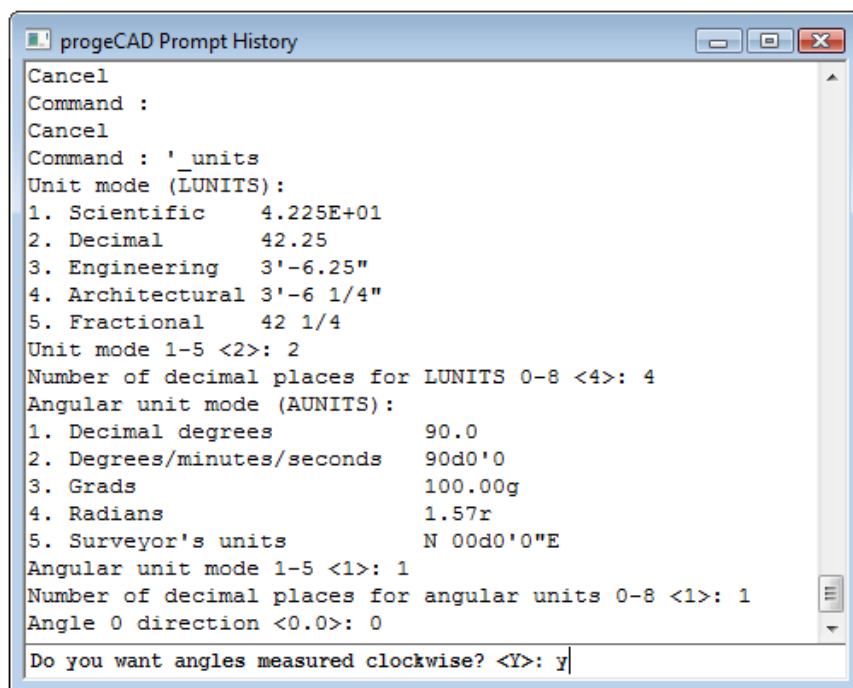


When you observe the units on the status bar, the system default is a four decimal place number, and the angular precision shows figures rounded to a whole numbers. From the menu bar, select Format, then Units to modify these adjustable units in the progeCAD drawing. You can see the Command Line prompts in Figure 8.16. Select Unit Mode 2, then 4 decimal places for Linear Units, then Angular Unit Mode 1, with 1 decimal place for angular units, 0 for Angle 0 direction, and Y for angles measured clockwise (Figure 8.16).

**Figure 8.16 – Setting Drawing Units**

## Setting the Drawing Limits

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Again in on the menu bar select Format. You will see the option to select the Drawing Limits. When selected, at the Command line, the remark "*Limits are off:   ON/<Lower left corner> <0.0000,0.0000>:*" will appear. Type "on" to turn the limits on, and then hit Enter to repeat the _limits command.  Hit Enter again to retain the 0,0 lower left limit, but when the second statement "*Upper right corner <100.0000,60.0000>:*" appears, type "100,60" to make a larger field on the graphical display when the drawing begins (Figure 8.17).

```
Command :_limits
Limits are off:  ON/<Lower left corner> <0.0000,0.0000>: on
Command :
Command :_limits
Limits are on:  OFF/<Lower left corner> <0.0000,0.0000>:
Upper right corner <100.0000,60.0000>: 100,60

Command :
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**Figure 8.17 – Setting the Drawing Limits at the Command Line**

# Saving a Drawing Template

To save your new template, select the Save tool from the Standard toolbar and the Save Drawing As window will appear as shown in Figure 8.18. You can save the template anywhere such as a floppy drive, server drive, or your own personal directory.
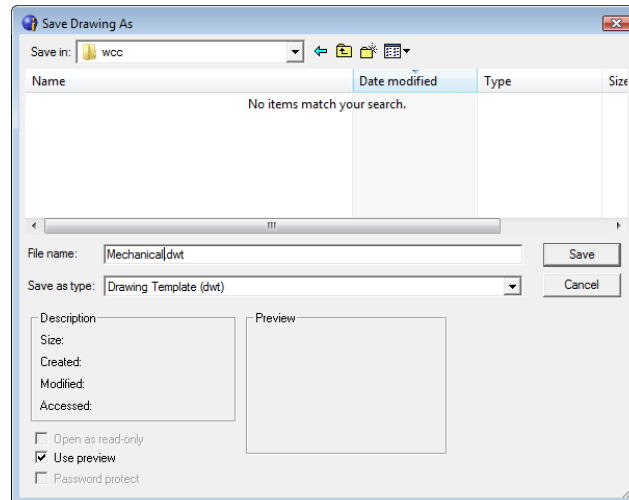


**Figure 8.18 – Saving the Mechanical Template**

Type the file name "Mechanical" and choose the File Type to be a Drawing Template File (*.dwt) as shown in Figure 8.18.

To make any changes to the template if an omission is discovered, open the template (dwt) in progeCAD, 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 100-17 \* - 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.**