# Placing Notes on the Circular Problem

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

1. Using the RealTime Pan and Zoom to change views in the drawing
2. Changing layers to place text on the drawing
3. Modifying Text Styles
4. Using the Multiline Text command to add text to the drawing
5. Using command codes to add special symbols to the text
6. Checking the spelling of the notes
7. Saving the drawing

Using the Real Time Pan Tool to Change the View

progeCAD offers a unique and easy way to manipulate the view of your drawing. When you click on the Real-time Pan tool on the Standard toolbar, a window will appear that explains these features. For 2D drawing you will only need to use the last two types of zooming and panning: by holding SHIFT+CTRL and the left mouse button you can zoom in and out by moving the mouse. Holding SHIFT+CTRL and the right mouse button allows you to pan around the drawing with the mouse. Remember, you are not changing the coordinates of the lines, circles and other entities that you have drawn, but rather the view point from which you are viewing.
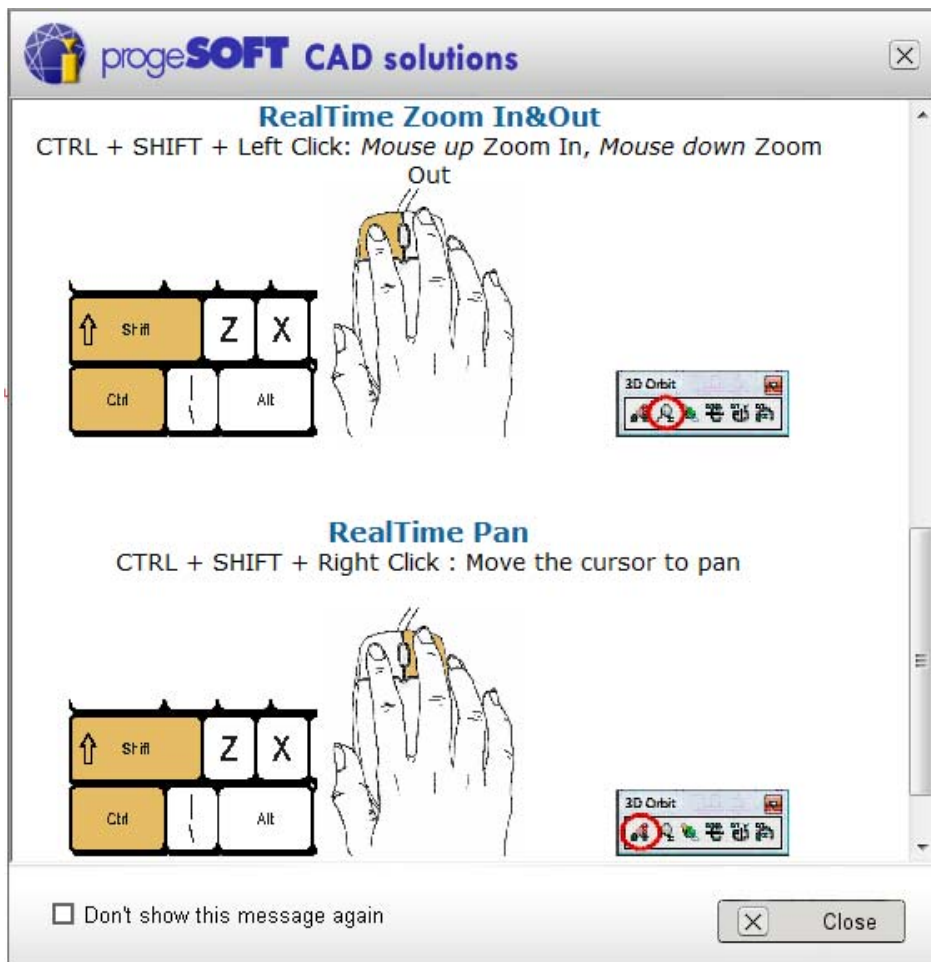

Figure 7.1 – The Real-time Pan and Zoom window

Use the RealTime Pan and Zoom tools to create space on the screen to place a few notes. See Figure 7.2 for an idea of how much to pan and zoom.

Changing Layers to Place Text on the Text Layer

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Next, you need to switch to the Text layer by clicking on the down arrow in the Layer Control List Box on the Entity Properties toolbar. Scroll down the list box and highlight the Text Layer (Figure 7.2). Once you left click on the Text Layer, the Text layer will show in the Layer Control window and become the current layer in the drawing. Now you are ready to add text to the drawing.
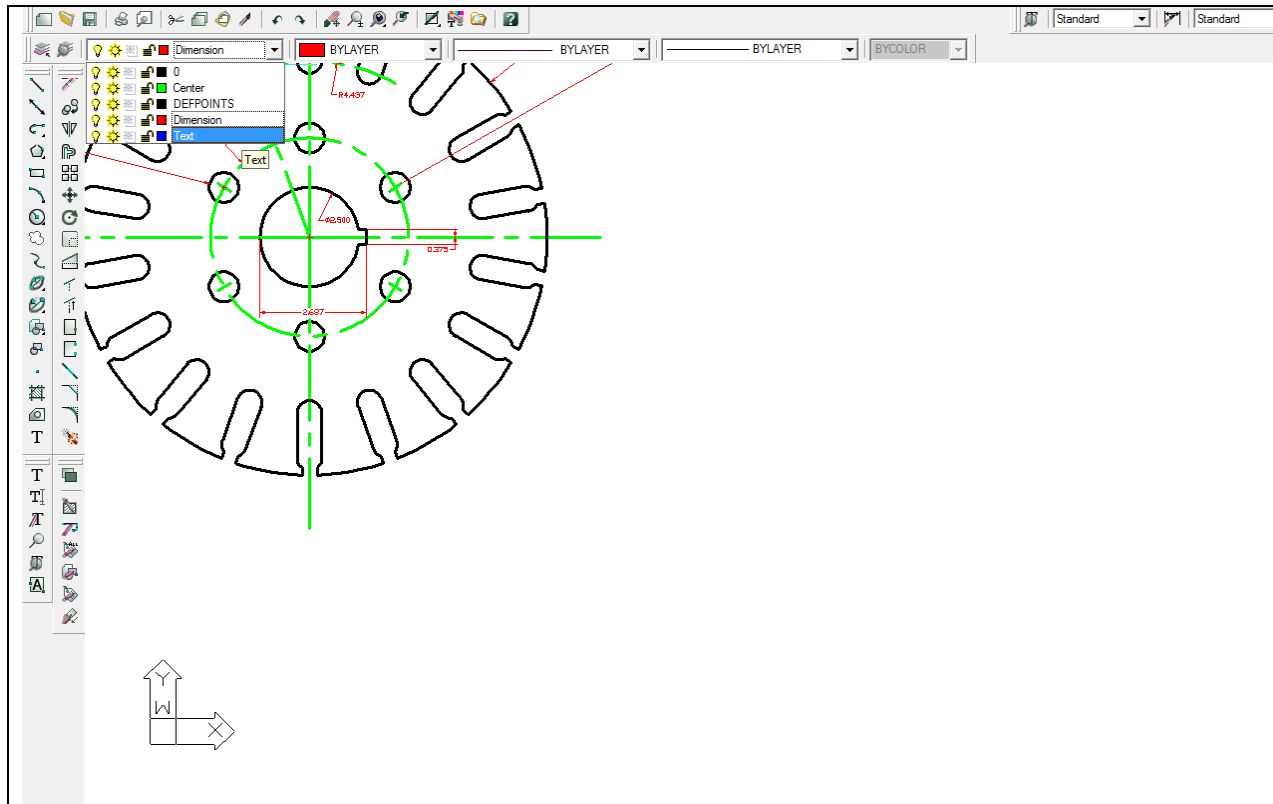
Figure 7.2 – Selecting the Text Layer

Modifying Text Styles

Before you can begin placing the notes using the Multiline Text tool, you need to modify the Standard text style by selecting the Explore Text Styles tool from the Text toolbar.   When the Text Style window appears change the Fixed text height to 0.125.   This is the only change you need to make. When you hit OK you will be prompted to save the text style. Hit Yes to continue and save the change.
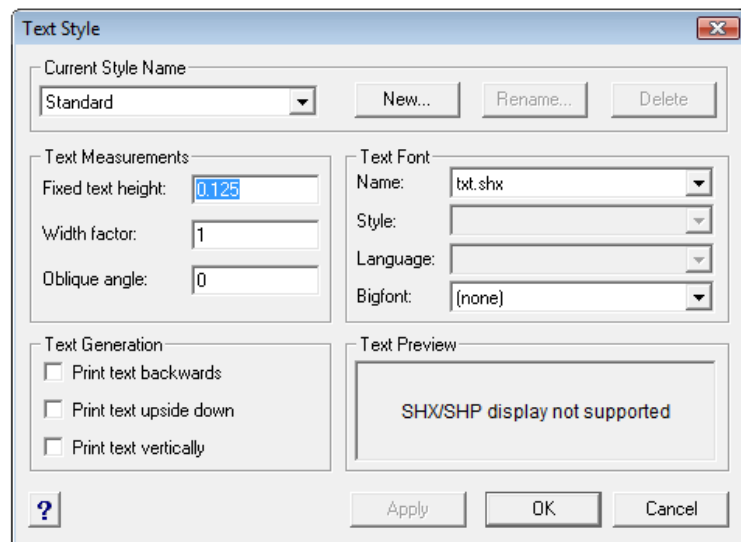

Figure 7.3 – Modifying the Standard Text Style

Using the Mtext Command to Place Text in the Drawing

To place text on the drawing, select the Multiline Text tool, or Mtext, on the Draw toolbar.   The command line will prompt you to select the first corner of the block of text. Select your first point to the bottom right of the circle (Figure 7.4). The command line will prompt you to specify the opposite corner. Make the Mtext window as large as you can, as shown in Figure 7.4.

The Multiline Text window will appear (Figure 7.5). Place your cursor in the text field. Now you are ready to type your notes, using ENTER when you need to start a new line.   The Multiline Text Editor is very much like a word processing program. You may need to increase your Zoom on the Multiline Text window to easily see your notes; 500% may be more appropriate. As you can see, the text size is already set to 0.125 because the Standard text style is selected on the Properties tab of the Multiline Text window.
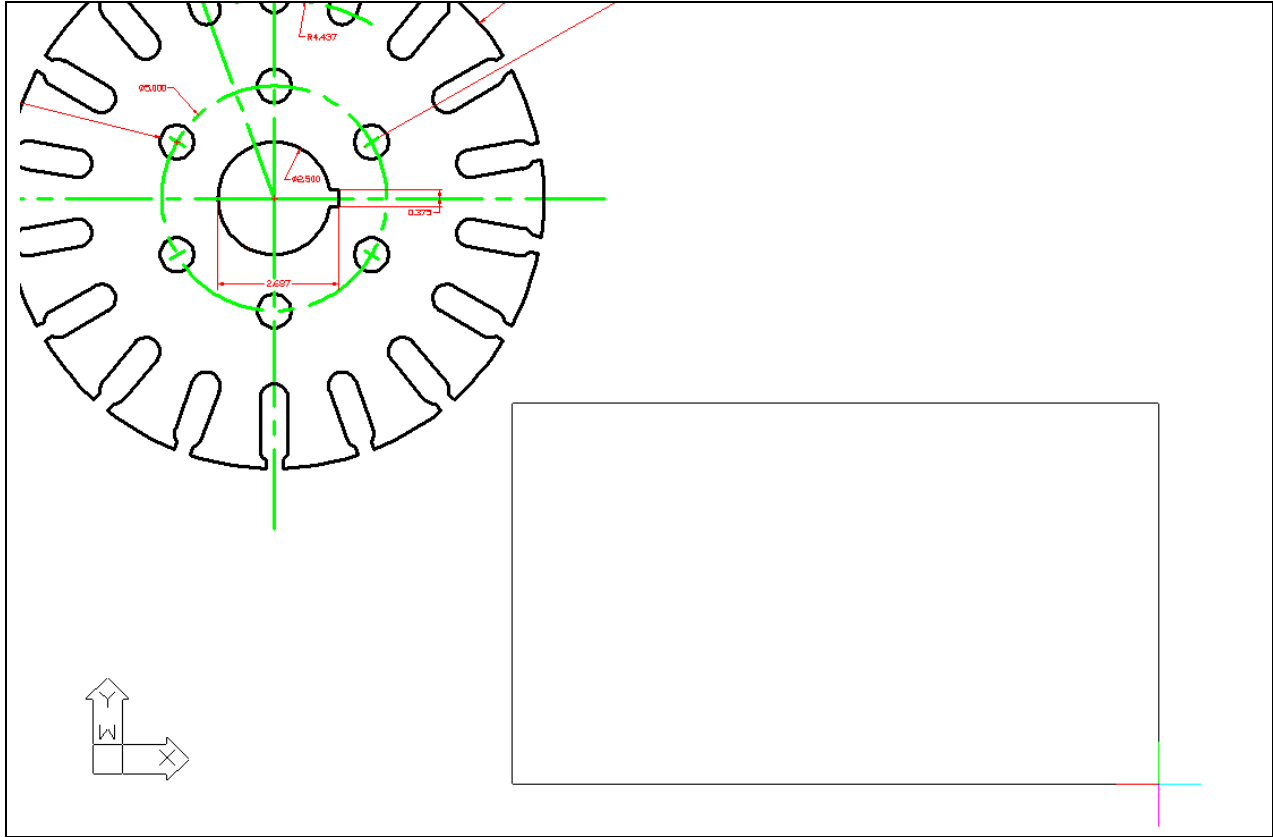
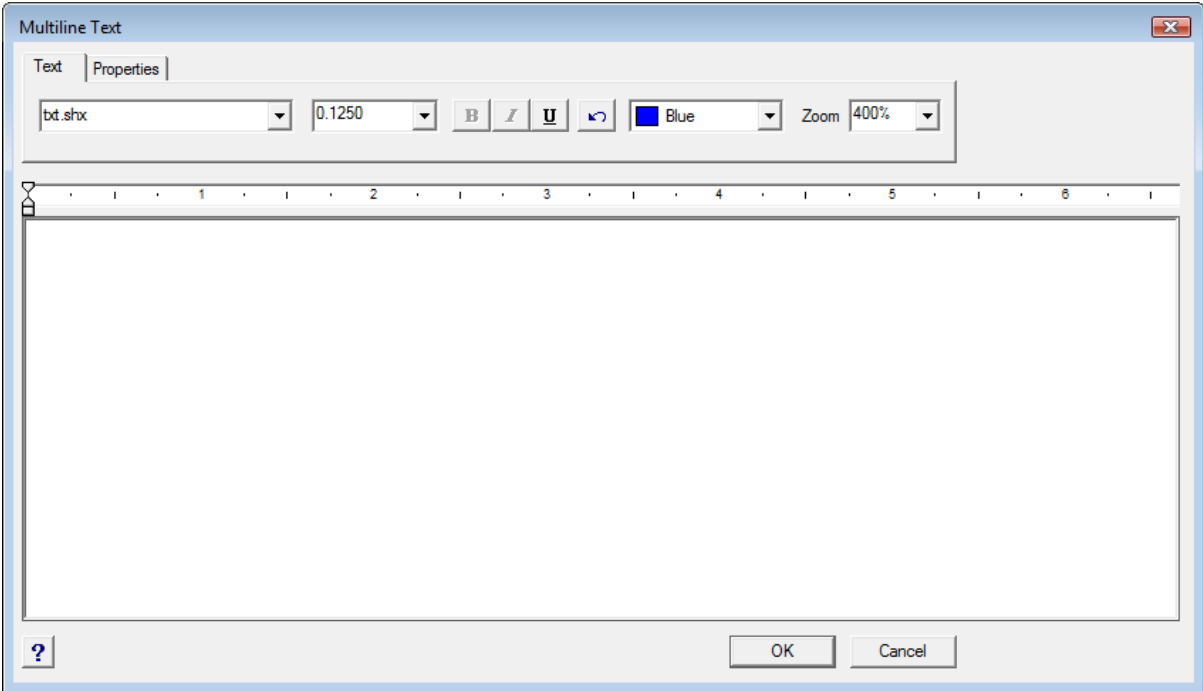**Figure 7.4 – Making the Mtext text window on the drawing**

Figure 7.5 – The Multiline Text Window

The notes you need to type are:

NOTES:
1. MATERIAL: 18 GA. (0.046) C.R.S.
2. REMOVE ALL SHARP EDGES AND BURRS.
3. CADMIUM PLATE
4. TOLERANCES UNLESS OTHERWISE SPECIFIED:
   3 DECIMAL: ±0.010
   ANGULAR: ±0.1°

When adding the plus and minus symbol to the 1 decimal note, type the colon and add a space, and then type "**%%p**".   This control code is one of many that can be used to enter special characters in text. Now you can type "0.010" (Figure 7.6).
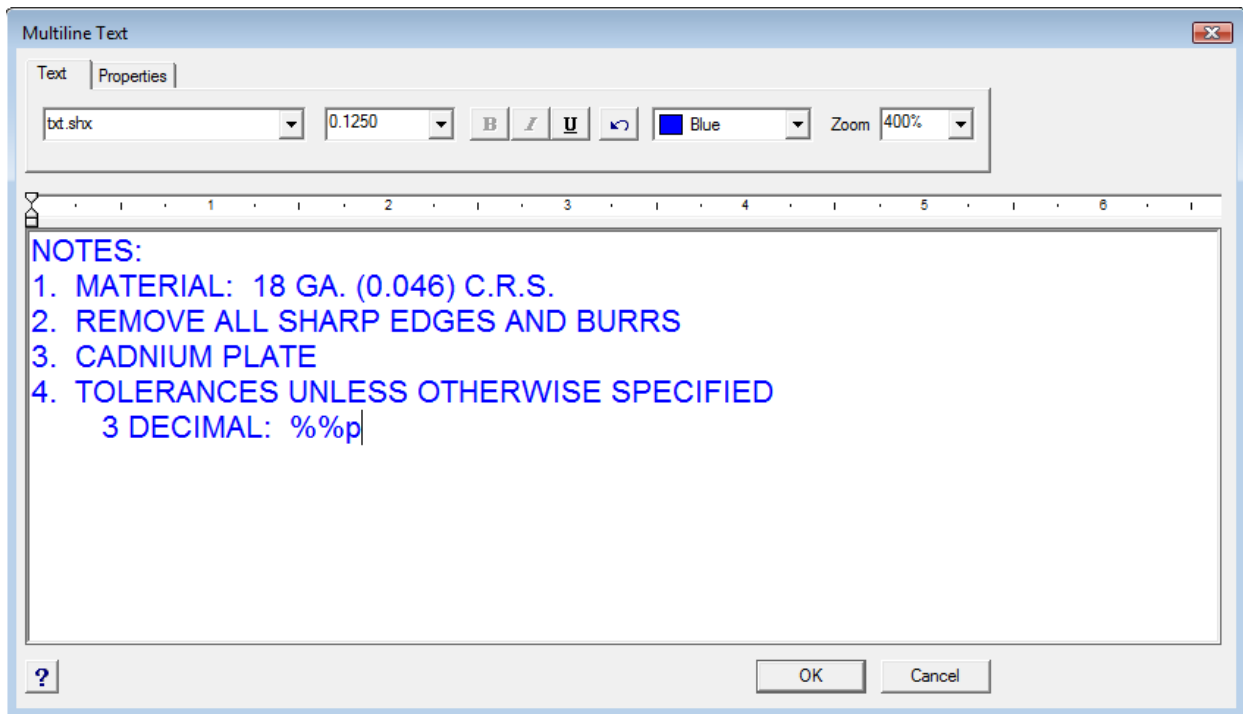

Figure 7.6 – Adding the plus or minus symbol in the Mtext window

When adding the degrees symbol after the Angular tolerance note, type the colon and add a space, and then type "**%%d**".   This is the control code that adds a degree symbol (Figure 7.7).
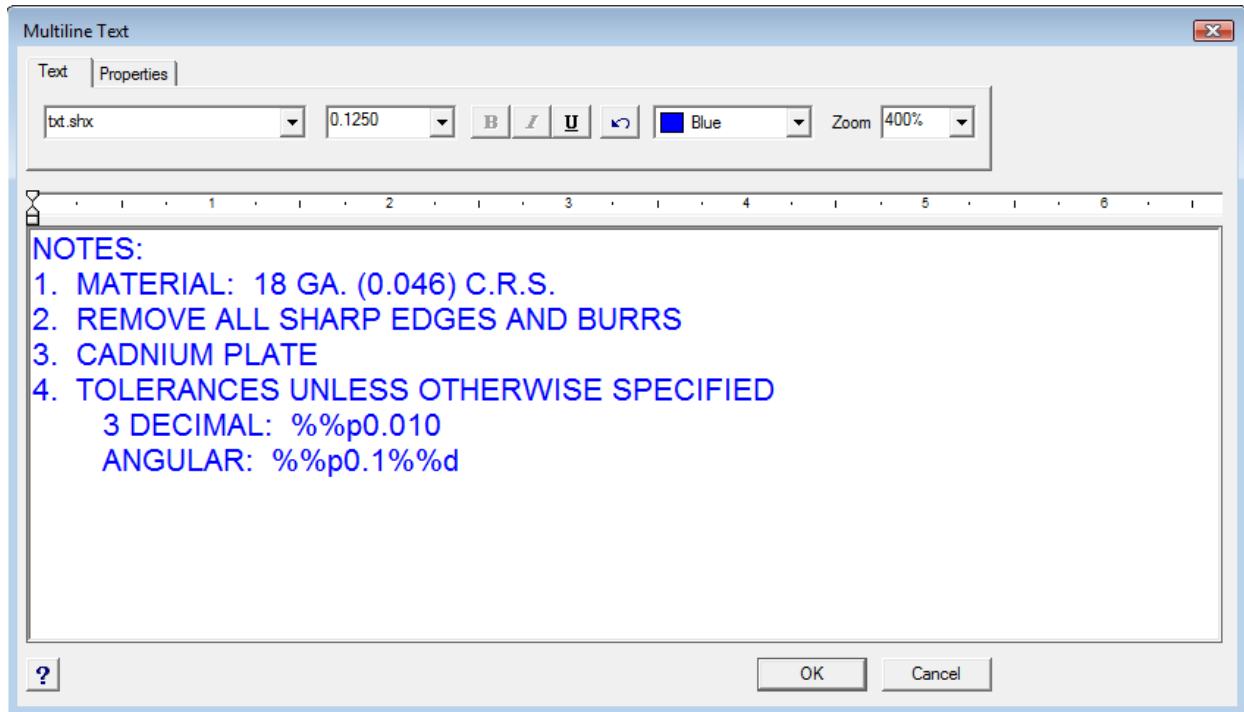
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Figure 7.7 – Adding the degree symbol control code

Checking the Spelling of the Notes

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You will see in Figure 7.7, the spelling of "cadmium" (CADNIUM) is incorrect. Unfortunately progeCAD Smart does not have a spell check feature so reviewing your notes for spelling errors is very important. There have been cases where jobs have been halted because of simple spelling errors on a drawing's notes. In order to communicate exactly what we want to say on a drawing, we must have correct spelling. To edit the text in the notes, just double-click on the notes and the Multiline Text window will automatically appear. Make the necessary changes and then hit OK to finish the notes (Figure 7.8).
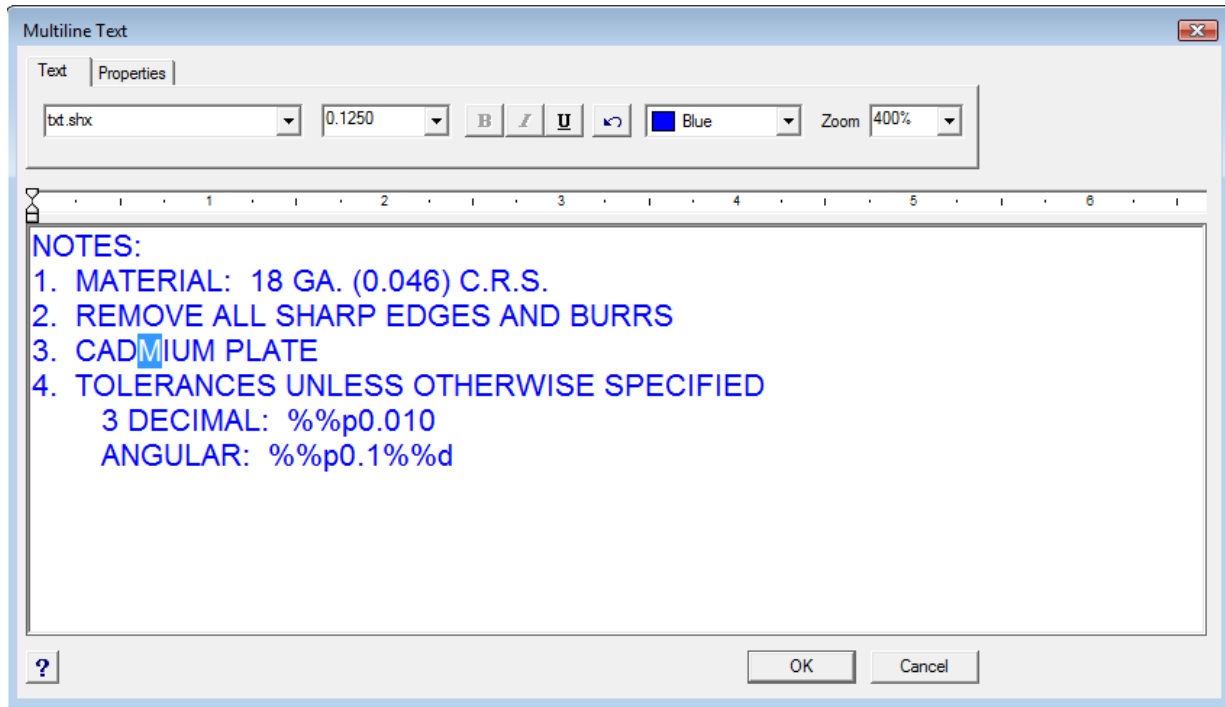
Figure 7.8 – Fixing the misspelled word using the Multiline text Window

Saving your Drawing

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To save the drawing, select the Save tool on the Standard toolbar. The Save Drawing As window will appear in the display.

You can make your own directory or you can place the drawing in the My Documents folder. Ask your Network Administrator or supervisor to show you where you should store your finished drawings. You should type the name of the drawing "Circular" in the File Name text box. The ".dwg" suffix is automatically added to the name when you save the drawing (Figure 7.9).
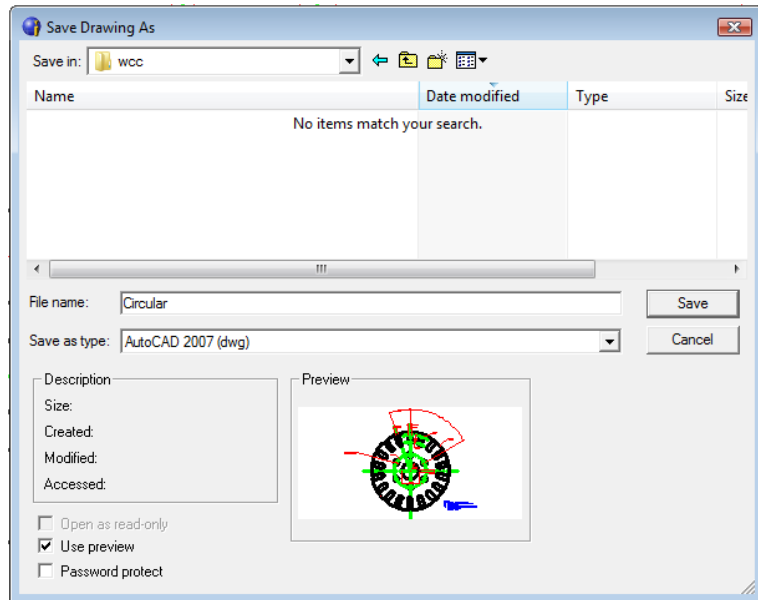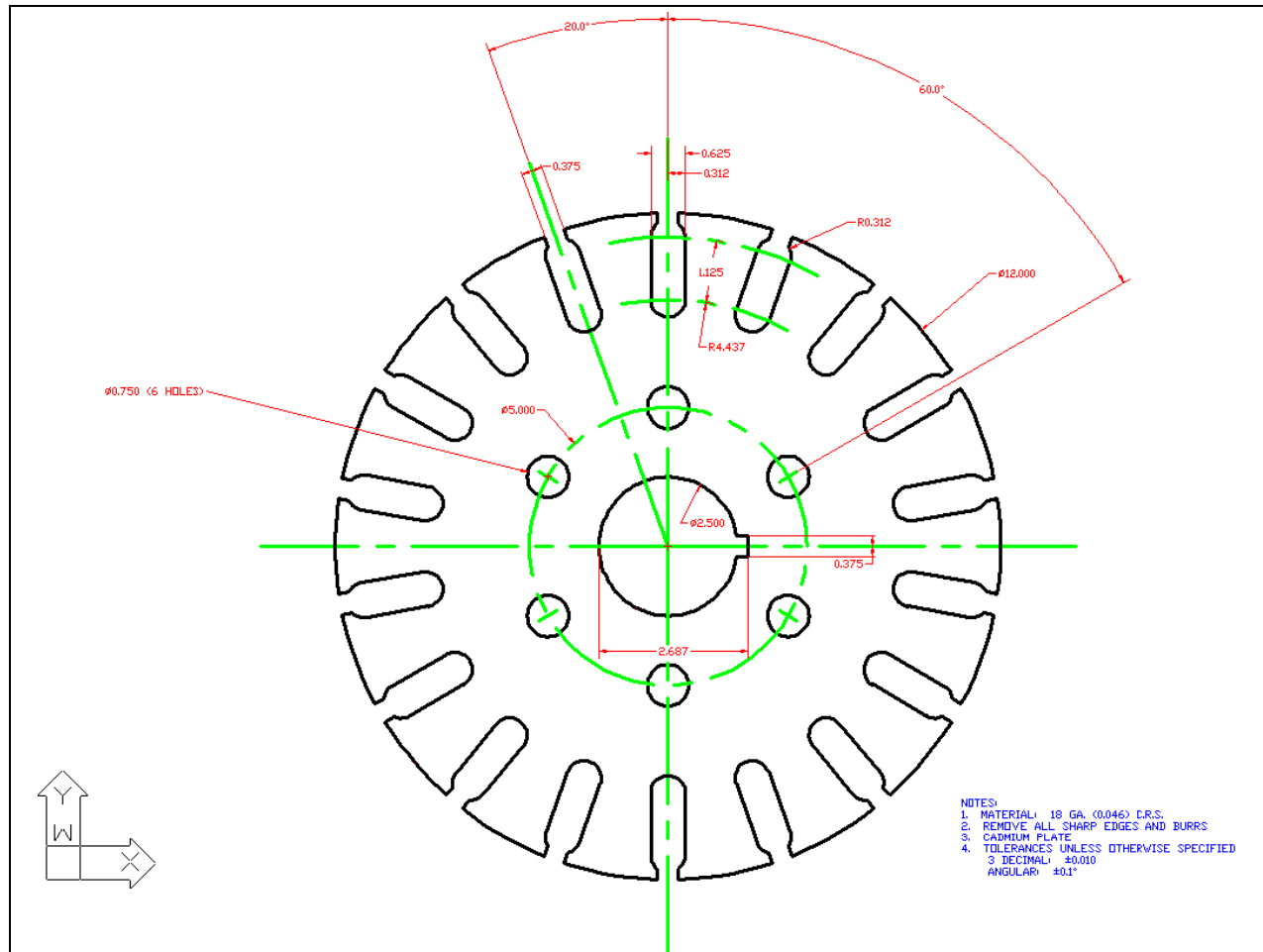

Figure 7.9 – Saving the drawing as circular.dwg

Figure 7.10 – The finished Circular Problem

* World Class CAD Challenge 100-16* - Close this drawing file. Create a New file and draw the entire circular problem on proper layers, using proper dimensions and finally placing the notes on the drawing. Continue this drill four times, each time completing the drawing under 15 minutes to maintain your World Class ranking. The world record for the complete drawing is under 5 minutes. See if you can match that effort, but if you are under 15 minutes, you may proceed to the next chapter, making a drawing template.

Send your best time and a copy of your drawing for verification to the authors of these problems to have your name, location and time posted. See the web site for instructions.
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