

Placing Notes on the Circular Problem

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

- 1. Use the tools and toolbar for simple 2D Computer Aided Drafting (CAD)**
- 2. Use the Pan Realtime tool to change views in the drawing**
- 3. Changing layers to place text on the drawing**
- 4. Use the Mtext command to add text to the drawing**
- 5. Use special symbols when adding text to the drawing**
- 6. Checking the spelling of the notes using the Spell command**
- 7. Saving the drawing**

Using the Real Time Pan Tool to Change the View

The real time panning tool is a fantastic asset to the CAD drafter and designer when moving around a drawing without changing the zoom factor. Select the Pan Realtime tool on the Standard toolbar and pick the center of the graphic display. Holding the left mouse button down, move the mouse in a direction and the drawing will move with you. Remember, you are not changing the coordinates of the lines, circles and other entities that you have drawn, but just the view port onto which you are looking. Press the right mouse button when you are done moving the display screen and a small menu will appear to allow you to toggle to a zoom option, 3D orbit or exit the Pan command. Select Exit and proceed to change to the Text layer to add your notes the drawing.



Figure 7.1 – The PAN Realtime exit menu

Changing Layers to Place Text on the Text Layer

Next, you need to change the layer to Text Layer by clicking on the black arrow in the Layer Control List Box on the 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 is now showing in the Layer Control window and becomes the current layer in the drawing. Now, you are ready to add text to the drawing.

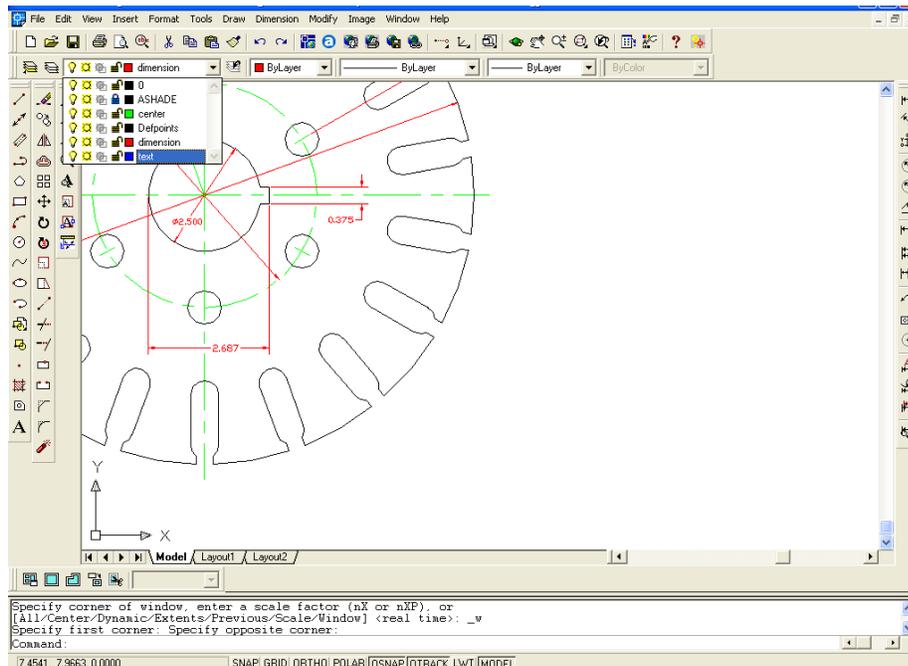


Figure 7.2 – Making the Mtext text window on the drawing

Using the Mtext Command to Place Text in the Drawing

To place text on the drawing, select the Mtext command on the draw toolbar. The command line will prompt you to select the first corner of the Mtext window. Select your first point directly below the vertical 4.00 dimension (Figure 7.3). The command line will prompt you to specify the opposite corner. Make the Mtext window as large as you can as in Figure 7.3.

A

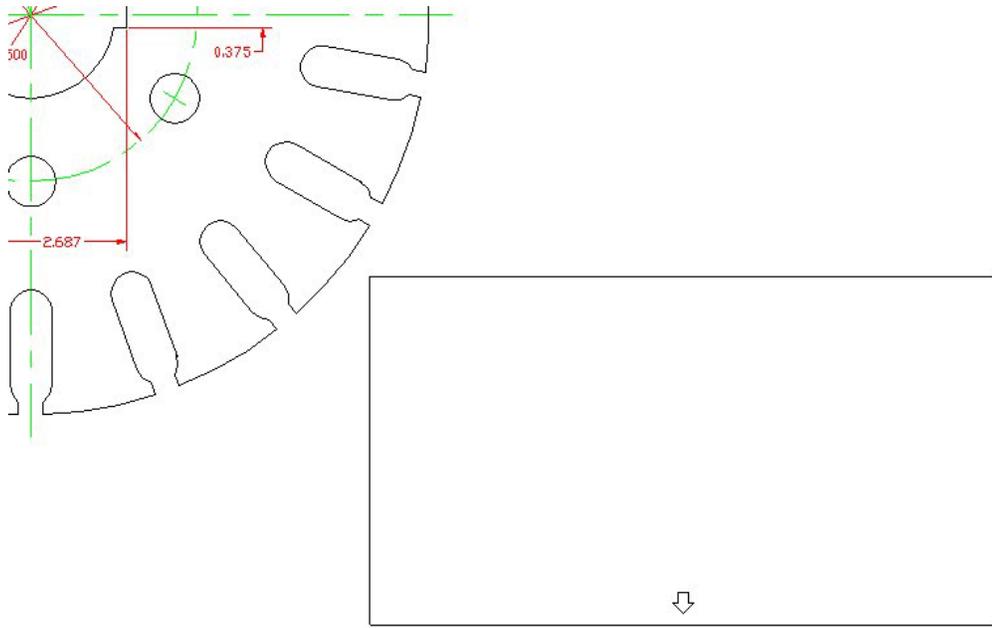


Figure 7.3 – Making the Mtext text window on the drawing

The Multiline Text Editor window will appear in the center of the display. Before you start typing, select the 0.2000 (Figure 7.4), the default text height on the Character tab and type 0.125 for the new text height. Next, select in the font window, where the “Txt” fonts is listed and replace the font with Arial. Now, select the typing window with your mouse to place the flashing typing bar. 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.

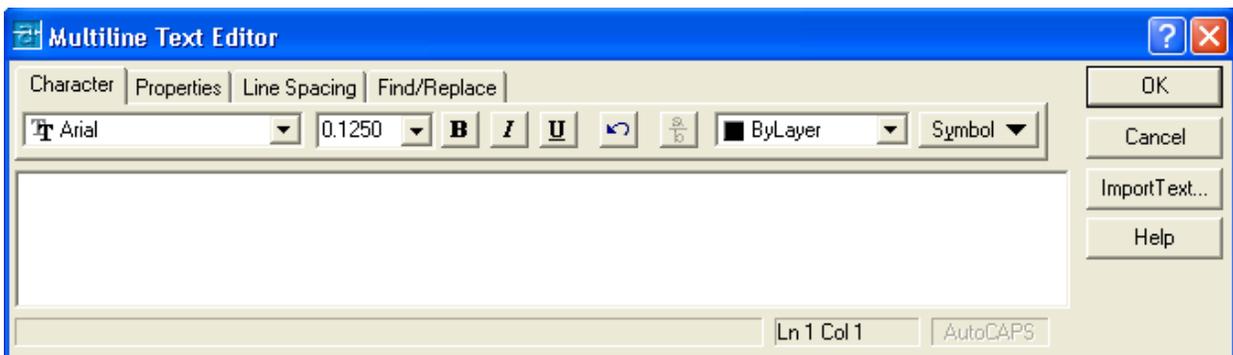


Figure 7.4 – The Multiline Text Editor window

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: $\pm 0.1^\circ$

When adding the plus and minus symbol to the 1 decimal note, type the colon and add a space, then select the Symbol button in the Multiline Text Editor. A menu will appear and select Plus/Minus to add the \pm symbol before typing “0.010” (Figure 7.5).

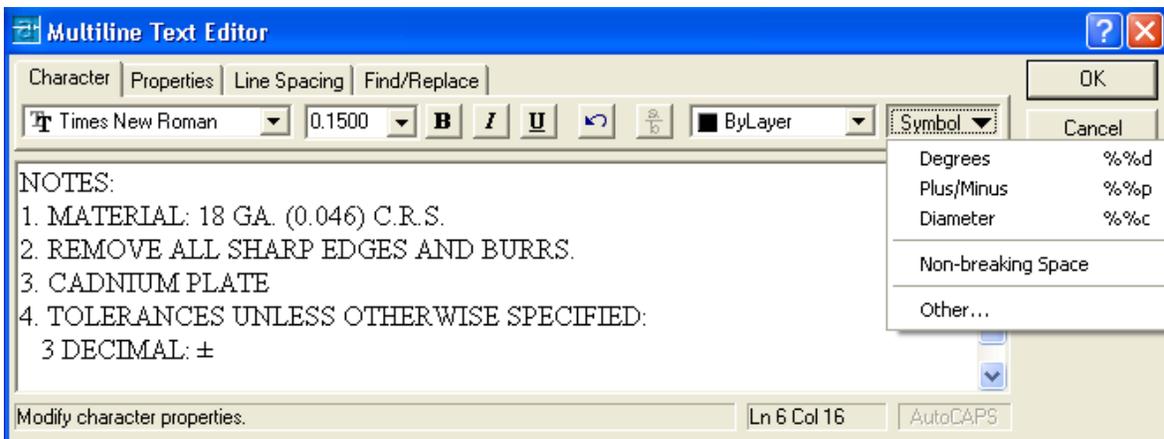


Figure 7.5 – Adding the plus or minus symbol in the Mtext window

When adding the degrees symbol after the Angular tolerance note, type the colon and add a space, then select the Symbol button in the Multiline Text Editor. A menu will appear and select Degrees to add the $^\circ$ symbol after typing “0.1”.

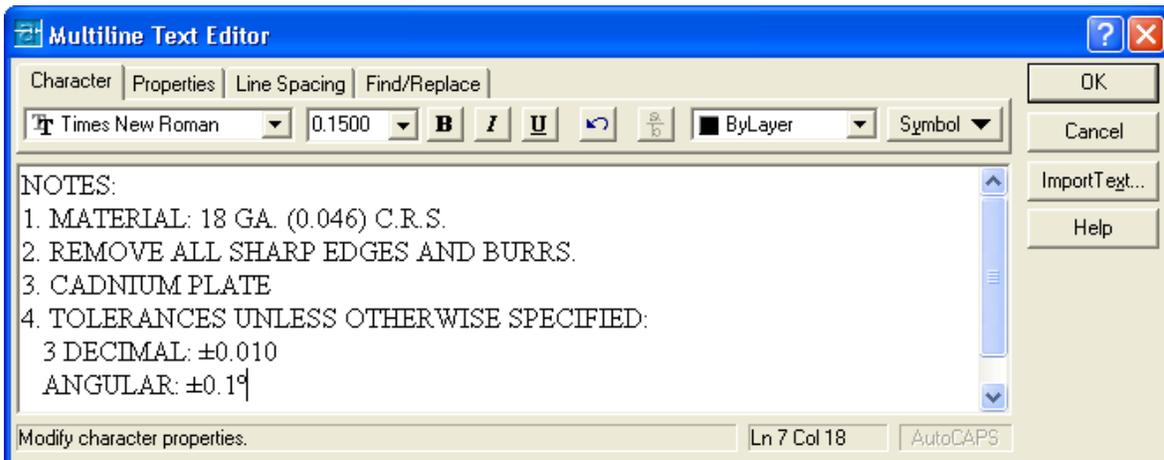


Figure 7.6 – Discovering a misspelled word in the notes

Checking the Spelling of the Notes Using the Spell Command

You will see in Figure 7.6, the spelling of “cadmium” (CADNIUM) is incorrect. To correct spelling errors in the drawing, type “Spell” and ENTER at the command line. When you type Spell at the command line, there will be one of two windows that appear in the display, which will be an AutoCAD Message stating that Spelling Check Complete or the Check Spelling window with a misspelled word. The Spelling check Complete message indicates you may not have a problem with your notes, which is fantastic and you will continue to finish and save the drawing. The Check Spelling window will have suggestions for the misspelled word. You may select the suggested correction; pick another suggestion from the list box, or type a correction in the suggestions window yourself. As you can see, you can Ignore the word or even Add the word to the dictionary by pressing the appropriate button in the Check Spelling window. In this example, the word “cadmium” was typed in the window and the Change button was pressed to correct the notes (Figure 7.7).



Figure 7.7 – Fixing the misspelled word using the Check Spelling window

Saving your Drawing

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 7.8).



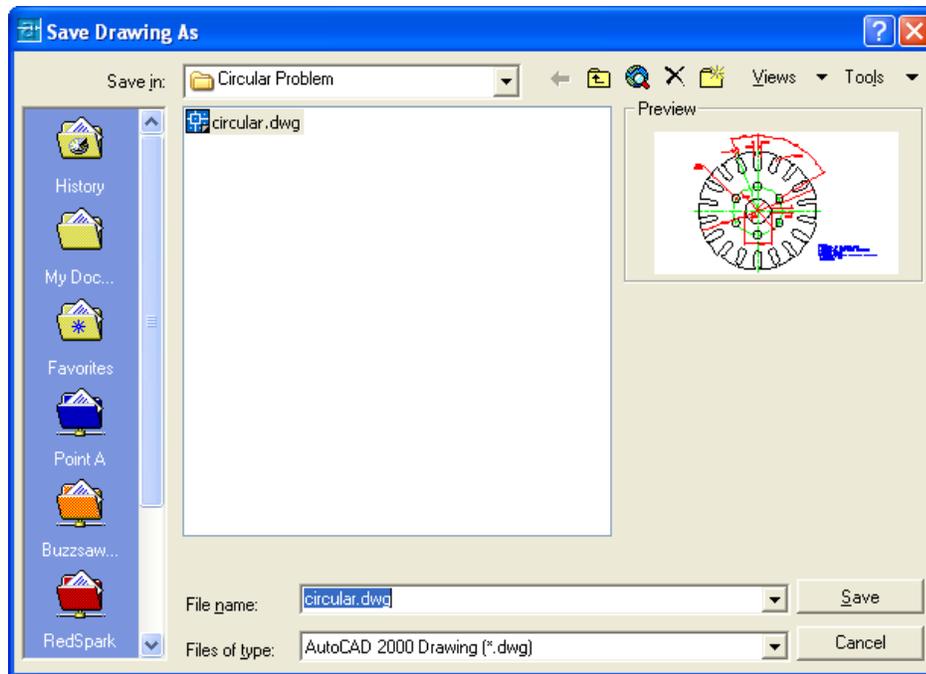


Figure 7.8 – Saving the drawing as circular.dwg

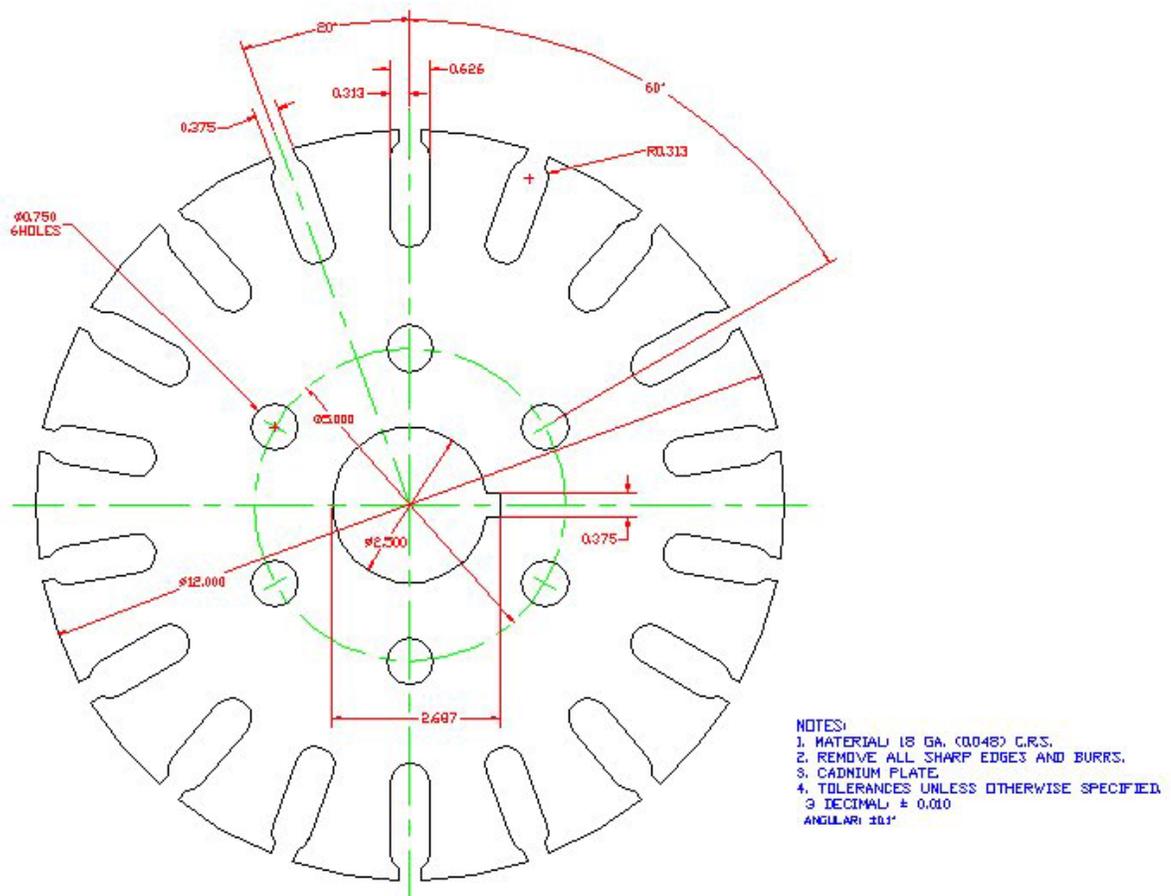


Figure 7.9 – The finished Circular Problem

*** World Class CAD Challenge * - 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, the Border Problem.**

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.

www.worldclasscad.com