

Placing Notes on the Circular Problem

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

- **The tools and panels for simple 2D Computer Aided Drafting (CAD)**
- **Using the Pan Realtime tool to change views in the drawing**
- **Changing layers to place text on the drawing**
- **Using the Mtext command to add text to the drawing**
- **Using special symbols when adding text to the drawing**
- **Checking the spelling of the notes using the Spell command**
- **Saving the drawing**

Using the Real Time Pan Tool to change the view

The Pan tool is a fantastic asset to the CAD drafter and designer that wants to move a drawing without changing the zoom factor. Select the Pan tool from the Navigation bar on the right side of the screen and pick the center of the graphic display. Holding down the left mouse button, move the mouse in any direction and the drawing will move with you. In this case, you need to move the drawing to the top left corner of the screen so that you have room to place notes, just like in the Rectangle problem. Remember, you are not changing the coordinates of the lines, circles and other entities that you have drawn, but rather the viewpoint from which you are viewing them. 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.

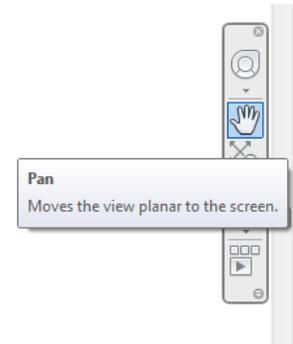


Figure 7.1 – The Navigation bar

Changing Layers to place text on the Text layer

You need to switch to the Text layer selecting Text from the Layer Control List Box on the Layers panel (Figure 7.2). Once you left click on the Text Layer, the Text layer will show in the Layer panel and become the current layer in the drawing. Now you are ready to add text to the drawing.

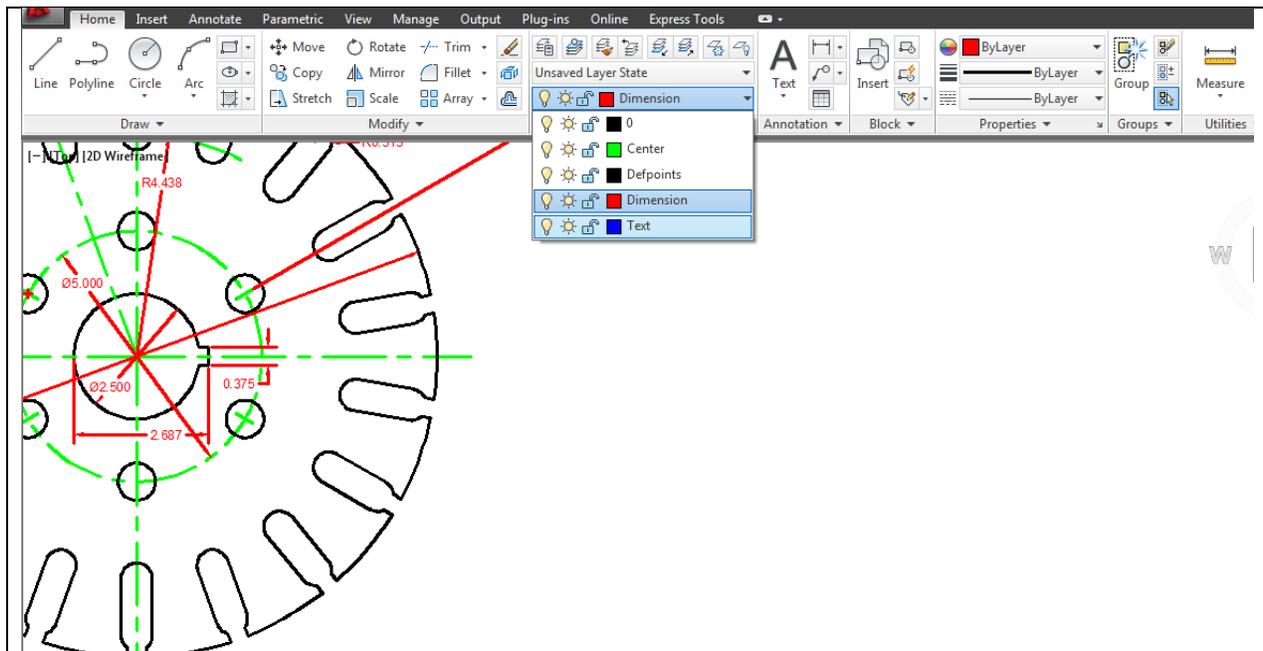


Figure 7.2 – Selecting the Text layer

Using the Mtext Command to place text in the drawing

To place text on the drawing, select the Multiline Text tool from the Text panel under the Annotate tab. The command line will prompt you to select the first corner of the Mtext window. Select your first point near the bottom right of the circle (Figure 7.3). The command line will then prompt you to specify the opposite corner. Make the Mtext window as large as you can, as shown in Figure 7.3.

A
Multiline
Text

The Text Editor tab will automatically appear. Before you start typing, select 0.2000 in the Style panel (Figure 7.4), the default text height, and type 0.125 for the new text height. Also, change the font to “Txt” to create a professional drafting look with your text. Select the typing window with your mouse to place the flashing cursor.

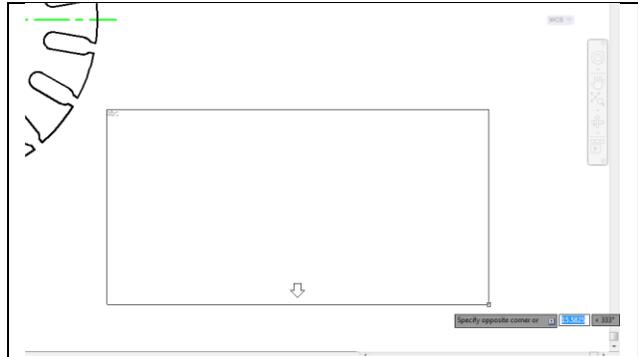


Figure 7.3 – Making the Mtext text window on the drawing

Now you are ready to type your notes, hitting ENTER when you need to start a new line. The Multiline Text Editor is very much like a word processing program. Feel free to zoom in if the text appears too small on the screen.

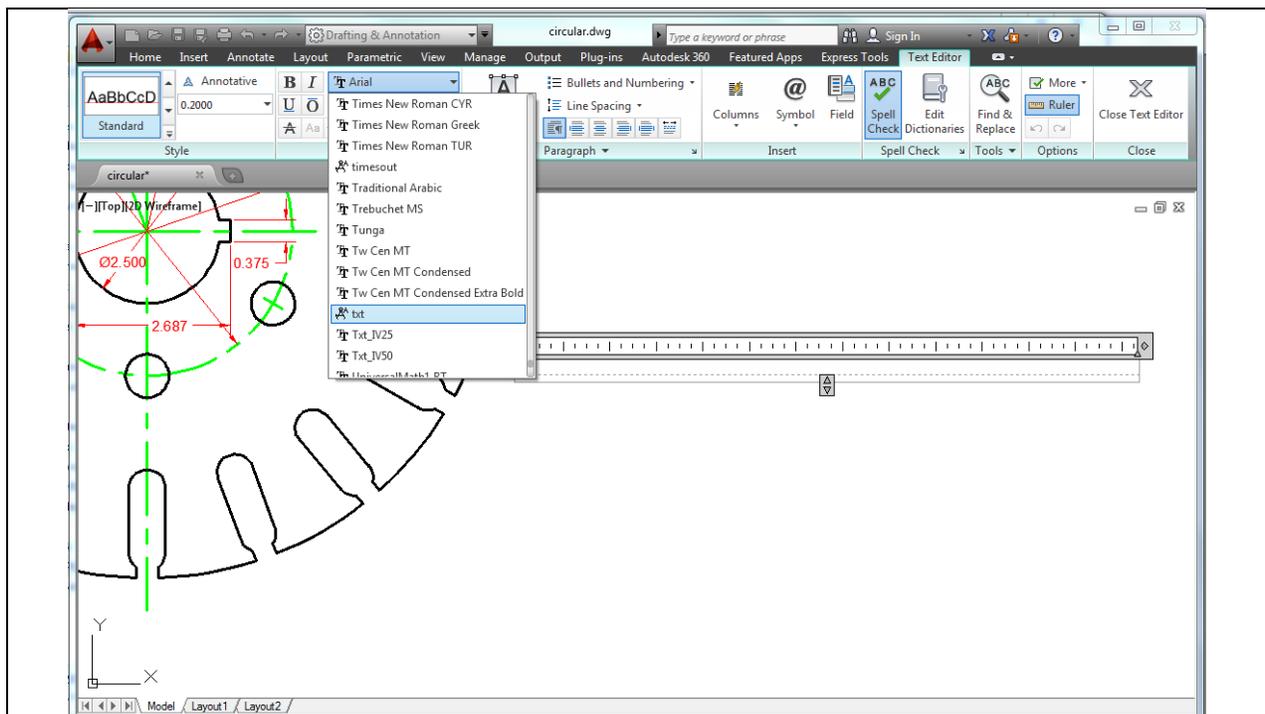


Figure 7.4 – The Multiline Text Editor window

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/minus symbol to the 1 decimal note, type the colon and add a space, then select the Symbol icon from the Insert panel. Select Plus/Minus symbol from the menu and the \pm symbol will appear. Now you can type “0.010” (Figure 7.5).

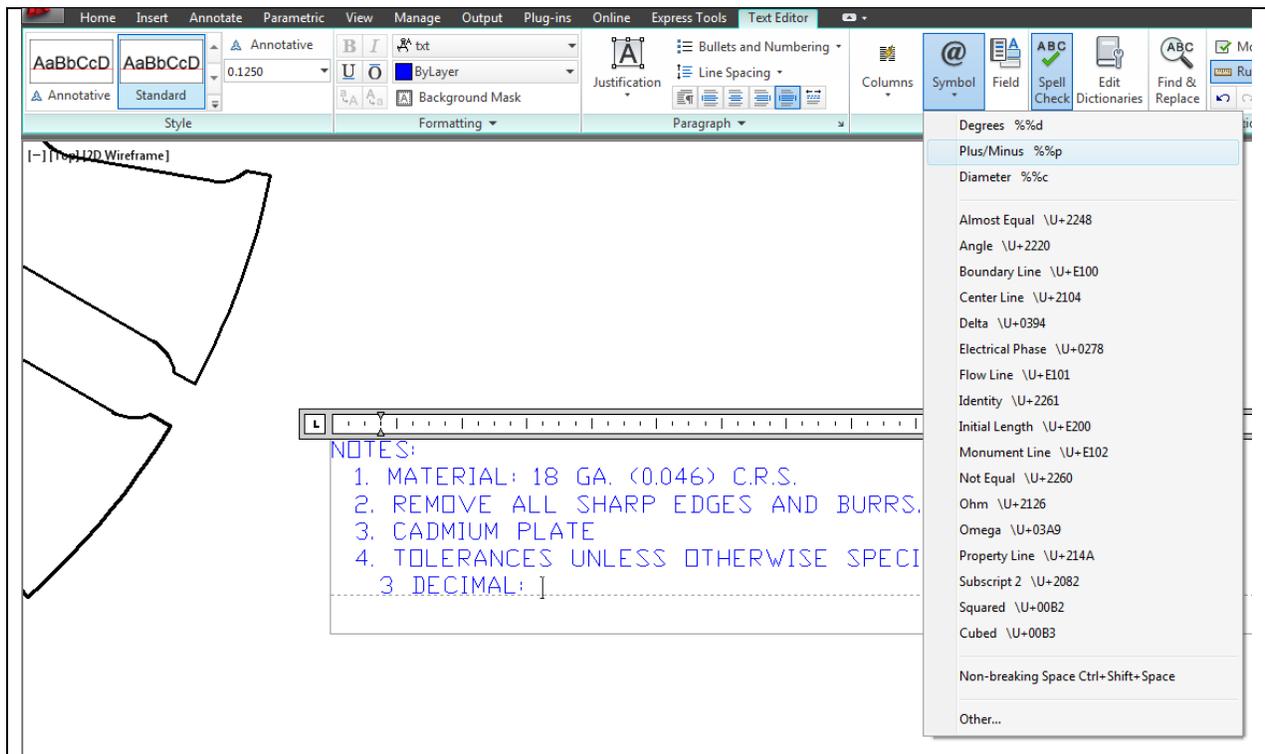


Figure 7.5 – Adding the plus or minus symbol in the Mtext window

When adding the degrees symbol after the Angular tolerance note, type the number value and then select the Symbol icon from the Insert panel. Select Degrees from the menu and the $^\circ$ symbol will appear.

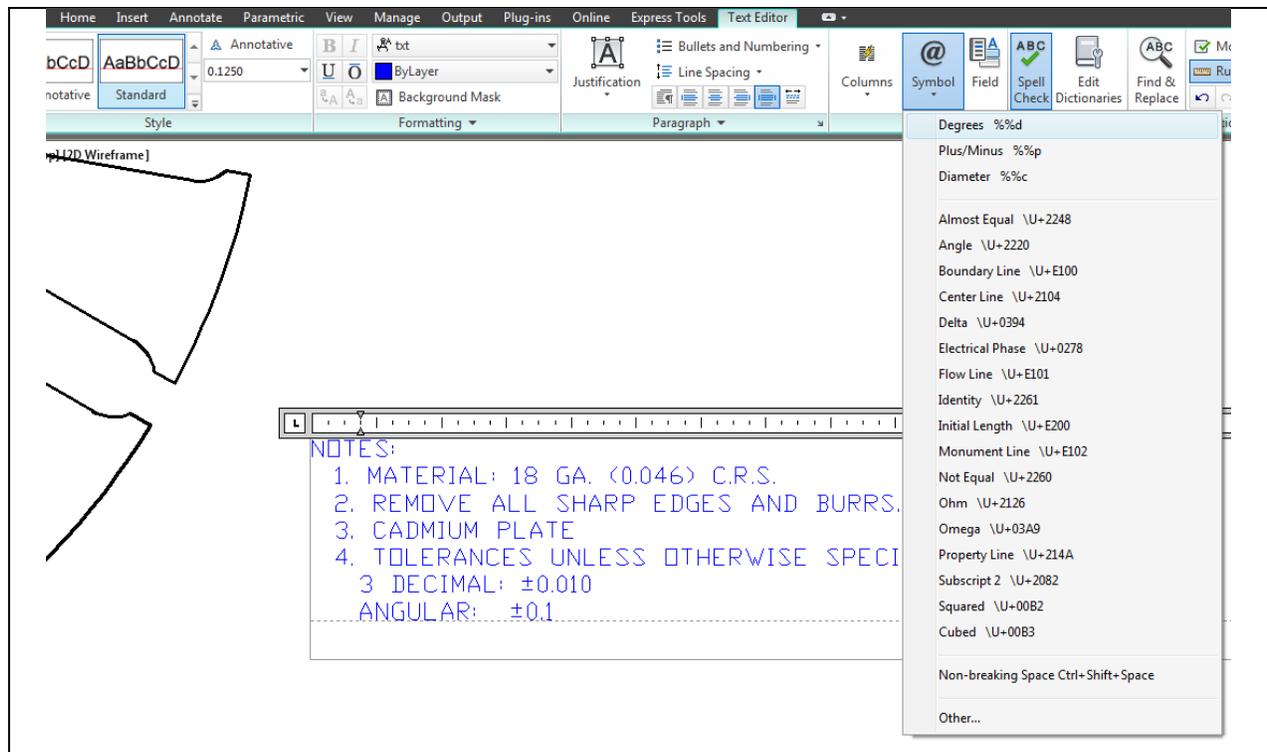


Figure 7.6 – Adding the degrees symbol

Checking the spelling of the notes using the Spell Command

Let us pretend the spelling of “cadmium” (CADNIUM) is incorrect in your notes. First, make sure that the Spell Check icon is toggled on in the Spell Check panel. If it is, any misspelled words you have typed will be underlined by a dashed red line, as shown in Figure 7.7. To quickly correct the mistake, right-click on the misspelled word and select the correct spelling from the top of the menu that appears (Figure 7.7). You also have the options to Ignore the mistake or to Add the word the dictionary. If none of the suggested words in the menu are the correct spelling, just exit the menu and correct the mistake manually. When there are no longer any underlined words, you have completed checking your spelling. Close the Text Editor tab by selecting the Close panel when you are finished editing the notes.

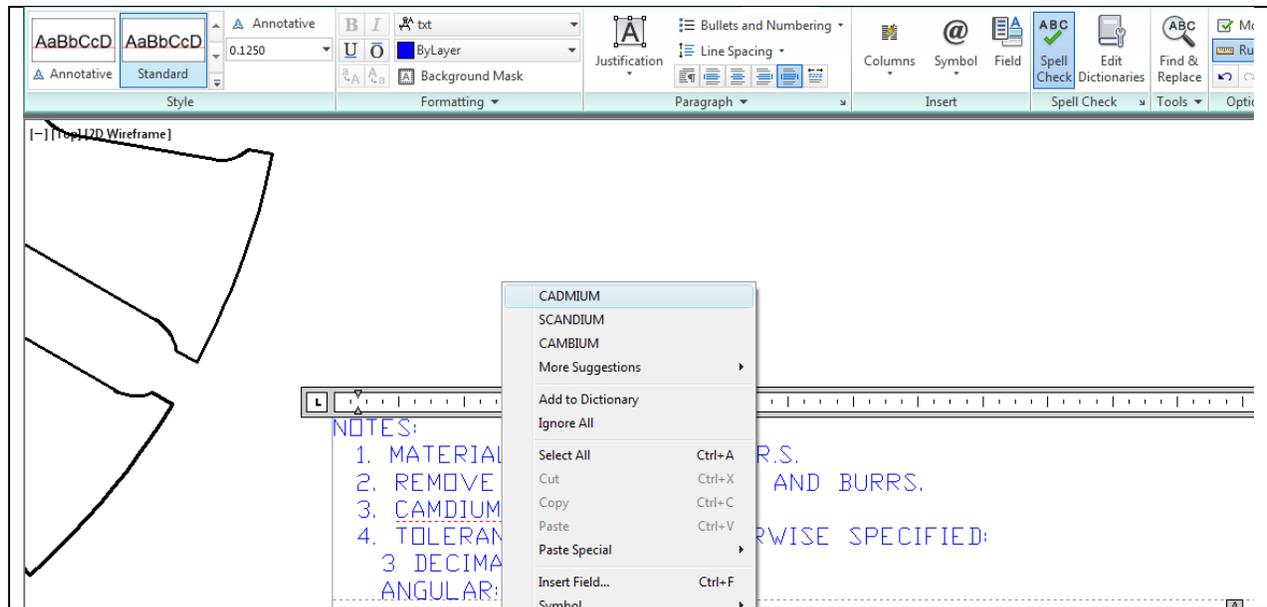


Figure 7.7 – Fixing the misspelled word by right-clicking

Saving your Drawing

To save the drawing, select the Save tool on the Quick Access toolbar in the top left corner of the screen. 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 “Circle” in the File Name text box. The “.dwg” suffix is automatically added to the name when you save the drawing (Figure 7.8).

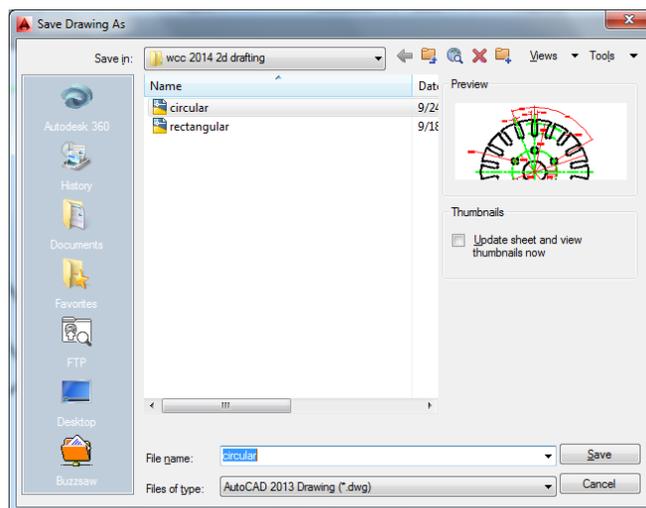


Figure 7.8 – Saving the drawing as circle.dwg

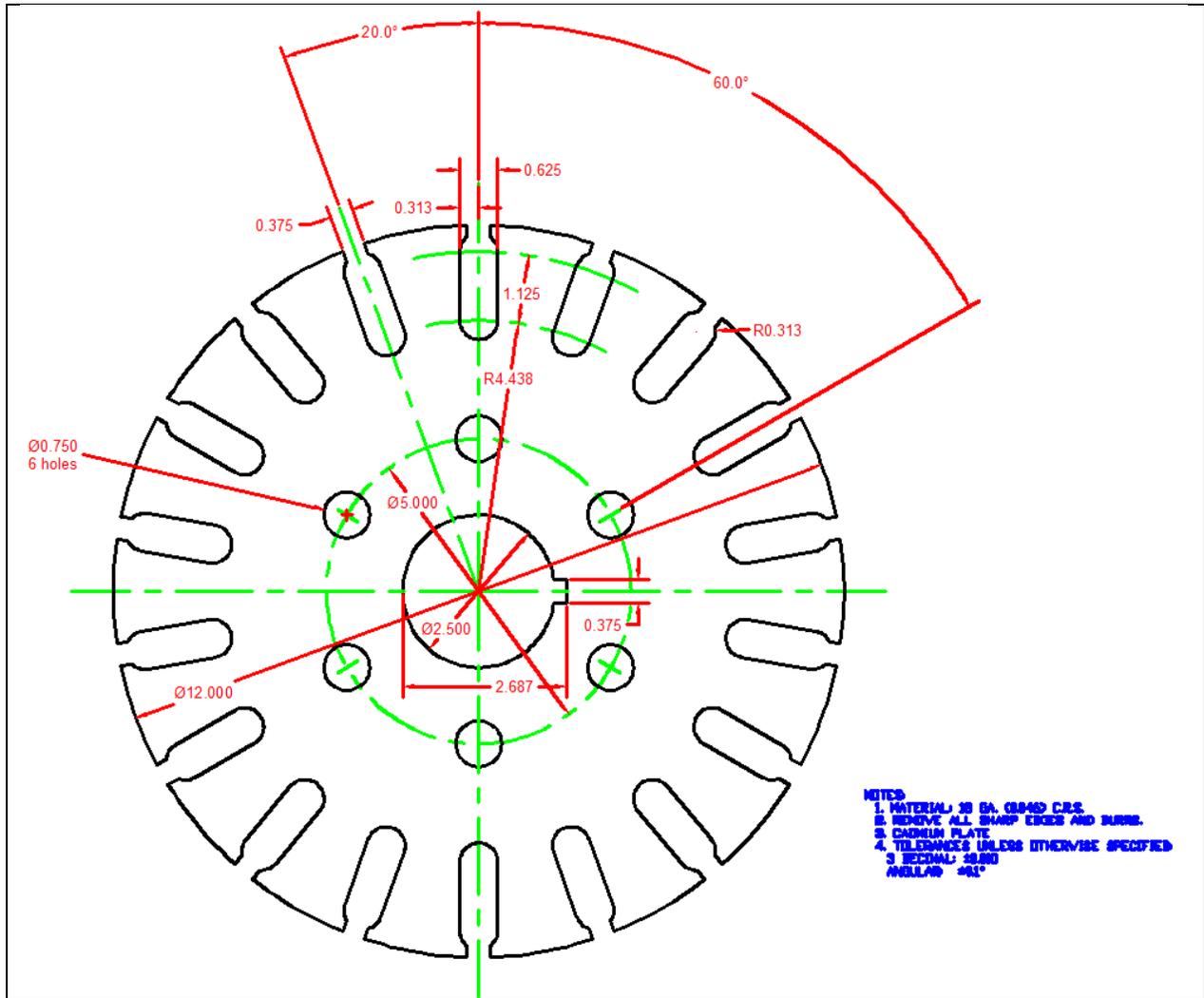


Figure 7.9 – The finished Circular Problem

* World Class CAD Challenge 02-16* - 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 in less than 15 minutes to maintain your World Class ranking. The world record for the completed 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.
www.worldclasscad.com