4

Placing Notes on the Rectangular Problem

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

- **Using the tools and toolbar for simple 2D Computer Aided Drafting (CAD)**
- **Using the Pan command 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 Pan Command to Change the View

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Use the Pan tool on the bottom of the screen to change the view so you will have enough room to place text next to your dimensioned drawing. Pick in the lower right hand corner of the display, holding the left mouse button down, and drag the Pan symbol (white glove) to the upper left corner of the display. The display should match the image in Figure 4.1.
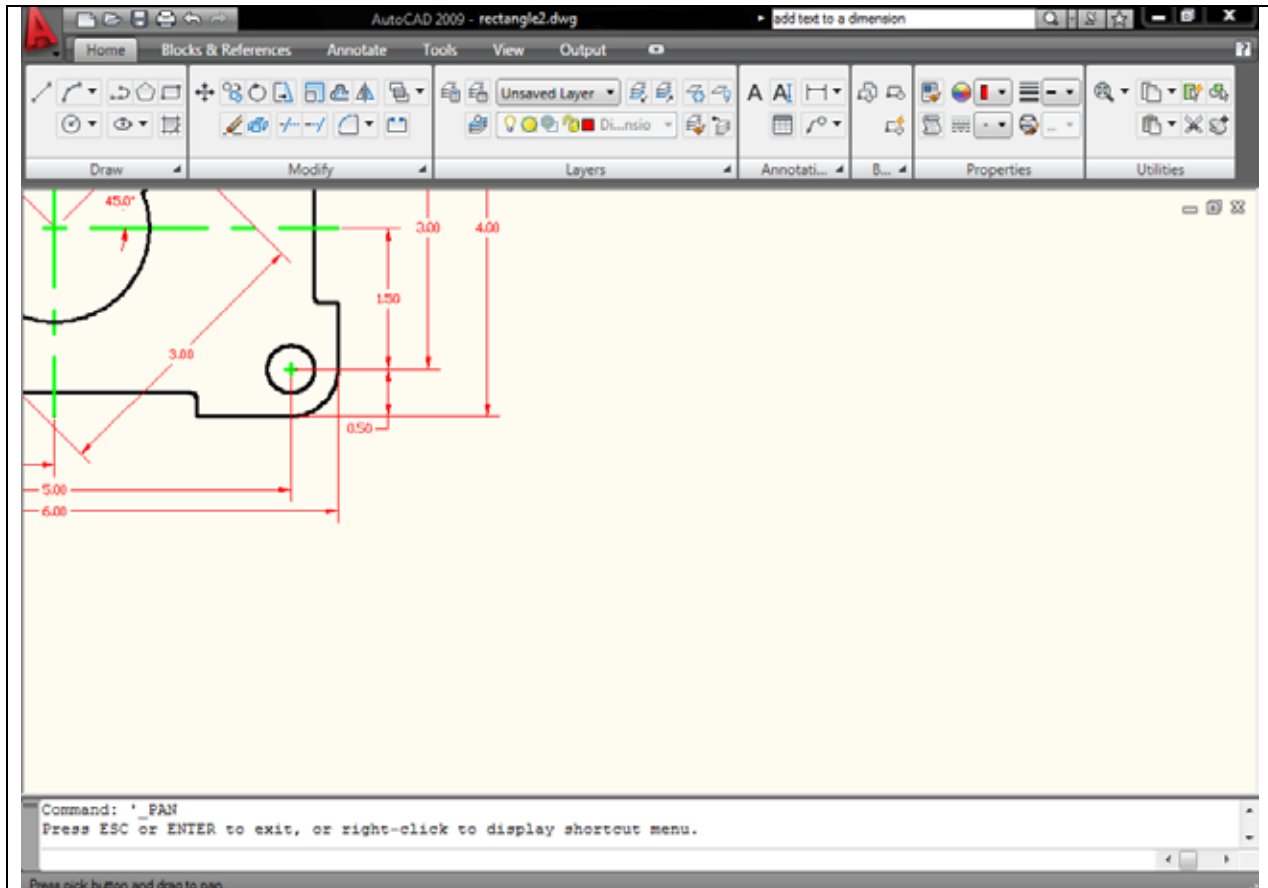

Figure 4.1 – Changing the view in the drawing using the Pan command

Changing Layers to Place Text on the Text Layer

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Next, you need to change the active layer to Text by clicking on the dropdown arrow in the Layer Control List Box on the Layer Panel. Scroll down the list box and highlight the Text Layer (Figure 4.2). Once you click on the Text Layer, the Text layer will show in the Layer Control window and will become the current layer in the drawing. Now, you are ready to add text to the drawing.
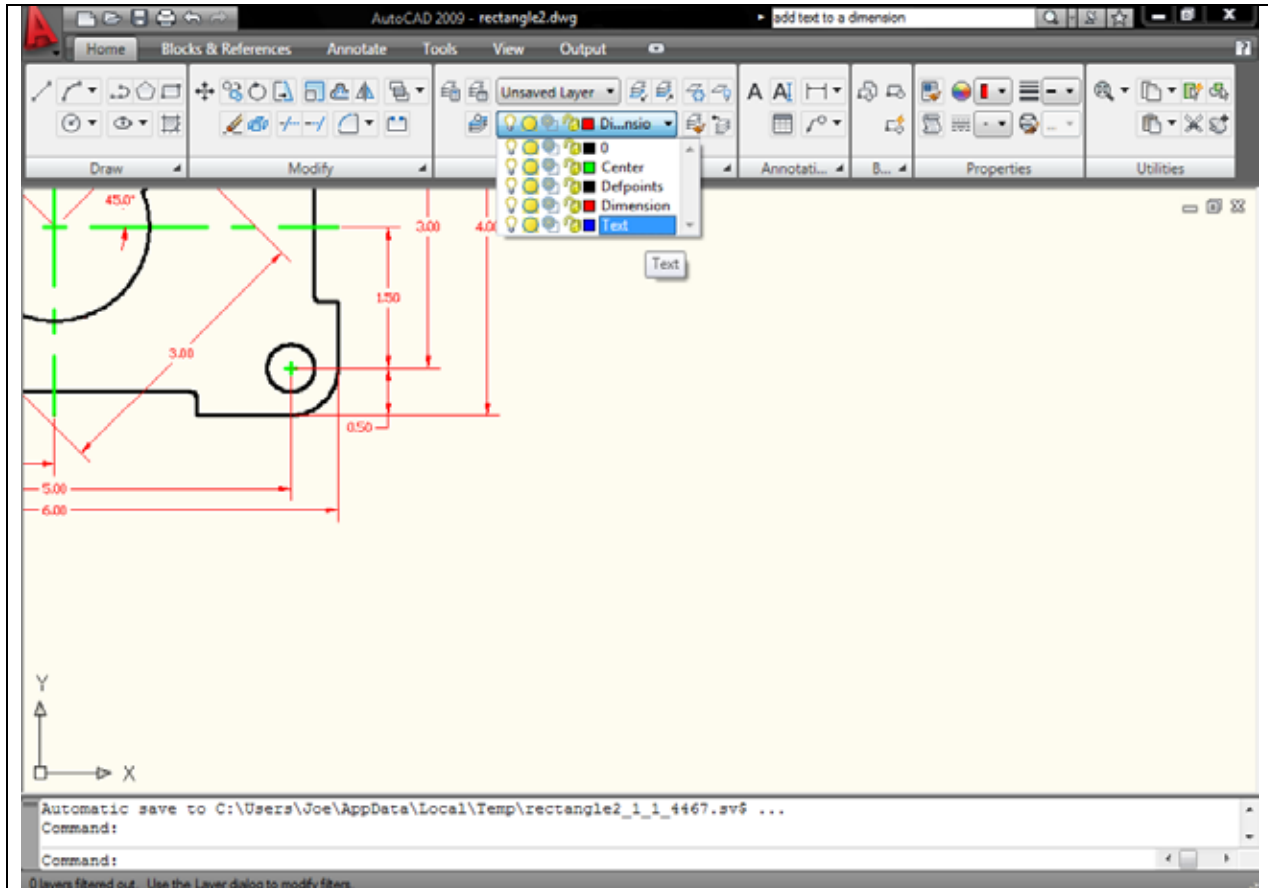
Figure 4.2 – Selecting Text as the active layer

Using the Mtext Command to Place Text in the Drawing
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Most drawings in your career will have text that describes the material, finish, coating, and tolerance. Notes on drawings will come from many sources. Architects, Designers, and Engineers, all who have the experience to select different material shapes and thicknesses based on physical calculations, will select material. Today, calculations are easier to do because Architectural and Engineering groups have created accurate handbooks and tables to expedite the material selection process. The finish of the material should be as smooth and clean as possible. Whether the part is an injected molded plastic or stamped aluminum, flash material that remains from manufacturing processing needs to be removed for handling safety and the coating process. This step of manufacturing costs money and is not done automatically, so you need to request finishing on your drawing. The coating process is done either for aesthetic reasons or to keep the part from decaying due to environmental impact, like rust from oxygen on aluminum or steel. Finally, you need to add tolerance control to the drawing. No component or part is the same even if it is made with the most accurate LASAR tool. Each part will be different and the tolerances on the drawing will control whether the component is able to fit and function in the assembly. You will experience many more

types of notes in your drawings, but these are the only ones required for your first problem. The efficiency of this command is directly affected by the speed and accuracy of your typing.

To place text on the drawing, select the Multiline Text command 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 directly below the vertical 4.00 dimension (Figure 4.3). The command line will prompt you to specify the opposite corner. Make the window as large as you can, as in Figure 4.3.
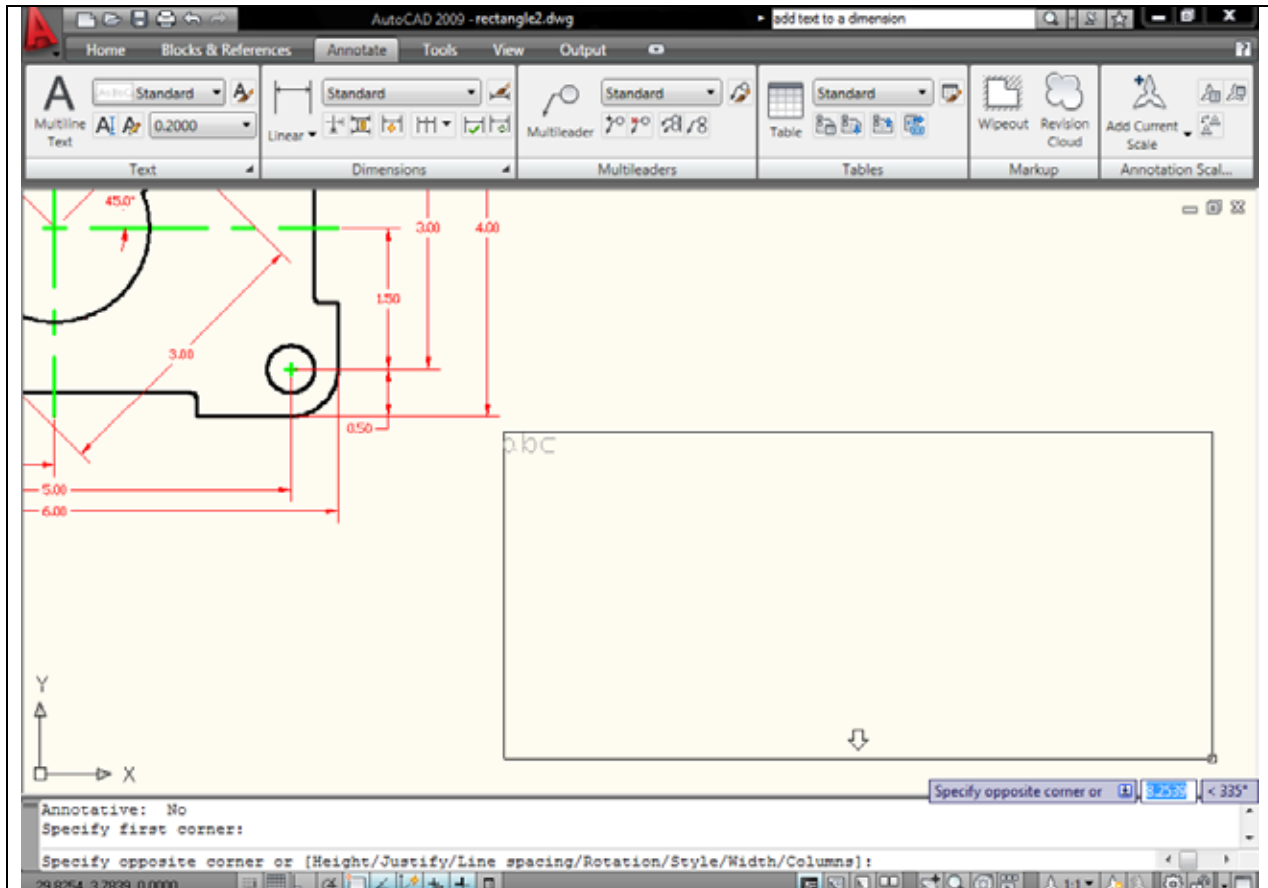

Figure 4.3 – Making the text window on the drawing

The Multiline Text tab will automatically appear after you draw the text box. Before you start typing, select the 0.2000 in the Style panel (Figure 4.4), the default text height, and type 0.125 for the new text height. Next, select the text box with your mouse to place the flashing cursor. 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.
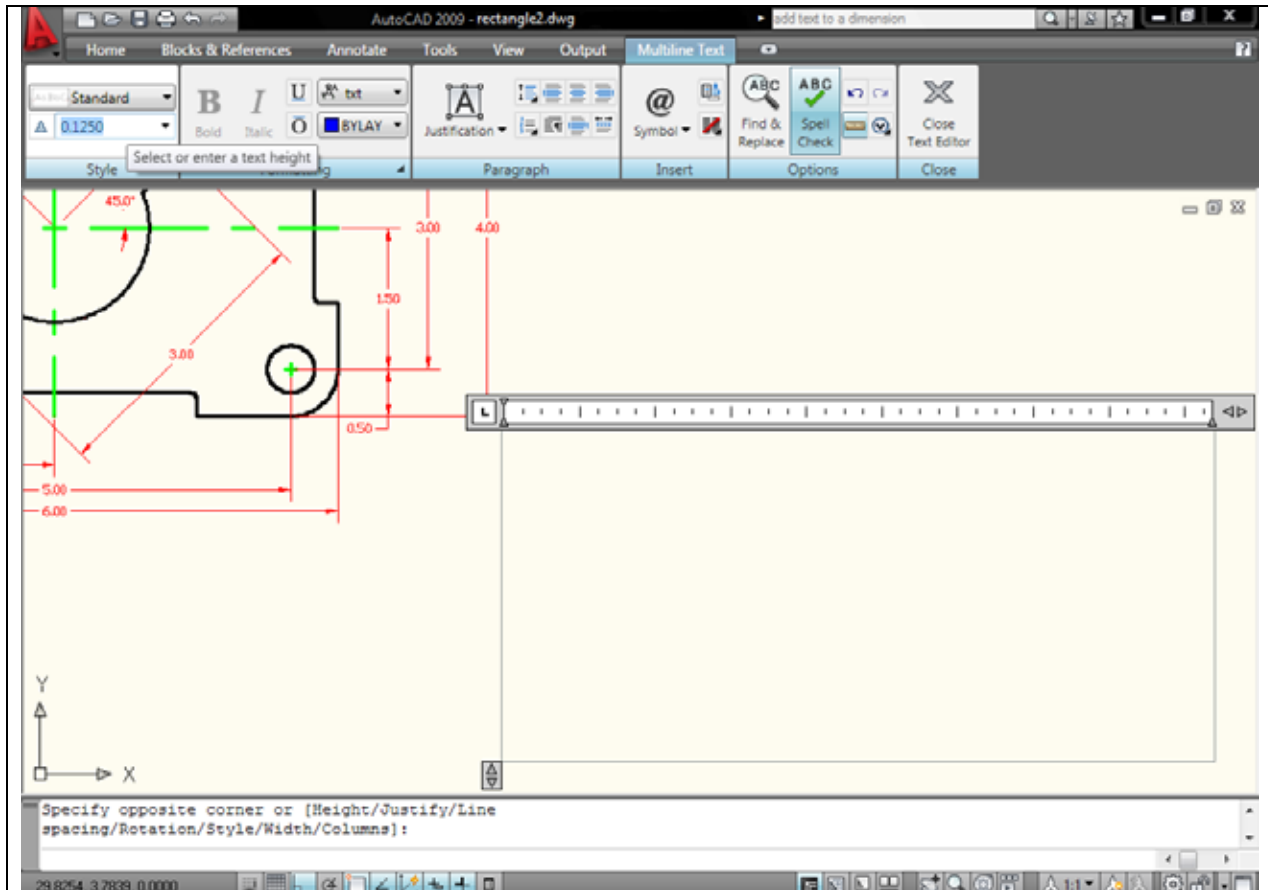
Figure 4.4 – The Multiline Text Editor window

The notes you need to type are:

> Notes:
> 1. Material: 0.125 thick aluminum.
> 2. Remove all sharp edges and burrs.
> 3. Clear anodize.
> 4. Tolerances unless otherwise specified:
>   1 decimal: ±0.06
>   2 decimal: ±0.010
>   3 decimal: ±0.003
>   Angular: ±0.5°

When adding the plus and minus symbol to the 1 decimal note, type the colon and add a space, then select the Symbol icon on the Insert panel. A menu will appear; select Plus/Minus to add the ± symbol before typing "0.06" (Figure 4.5). Select Symbol, then Degrees to add the ° symbol after typing "0.5" (Figure 4.6).
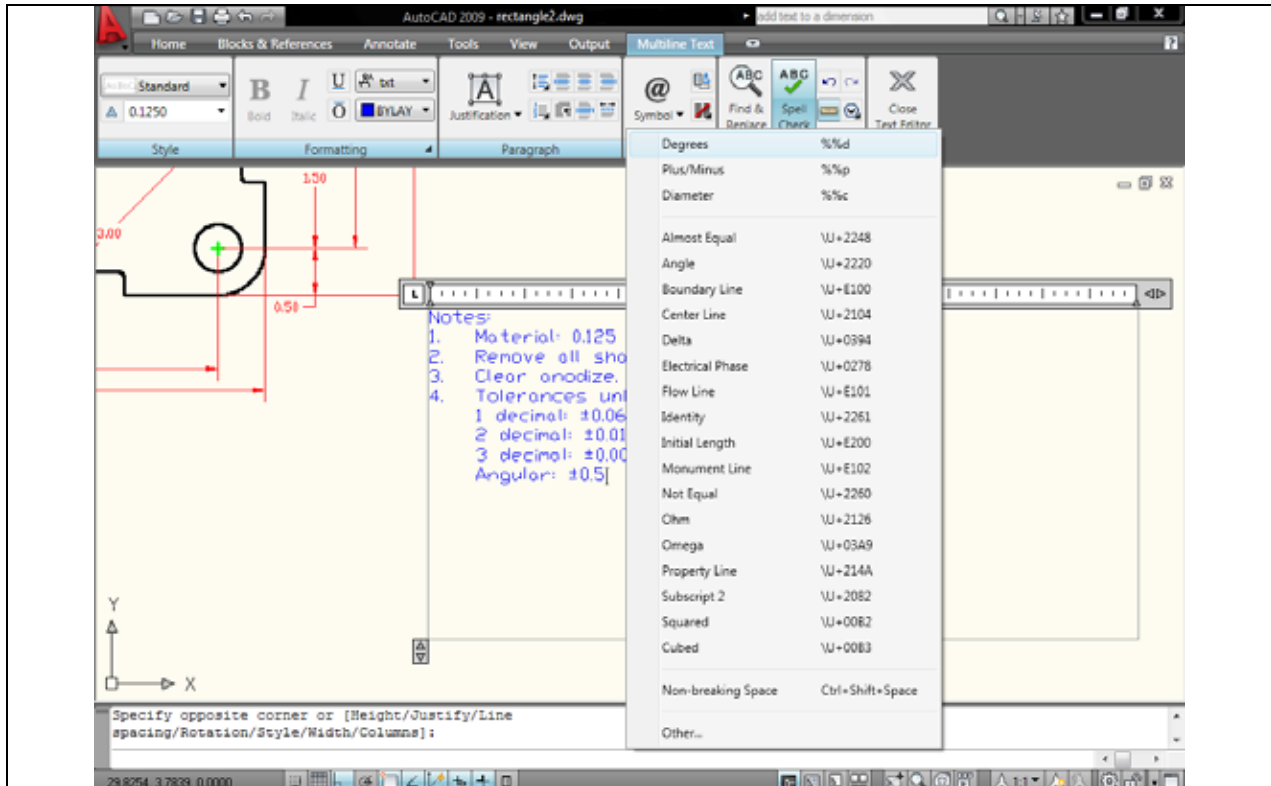
Figure 4.5 – Adding the plus or minus symbol in the Mtext window

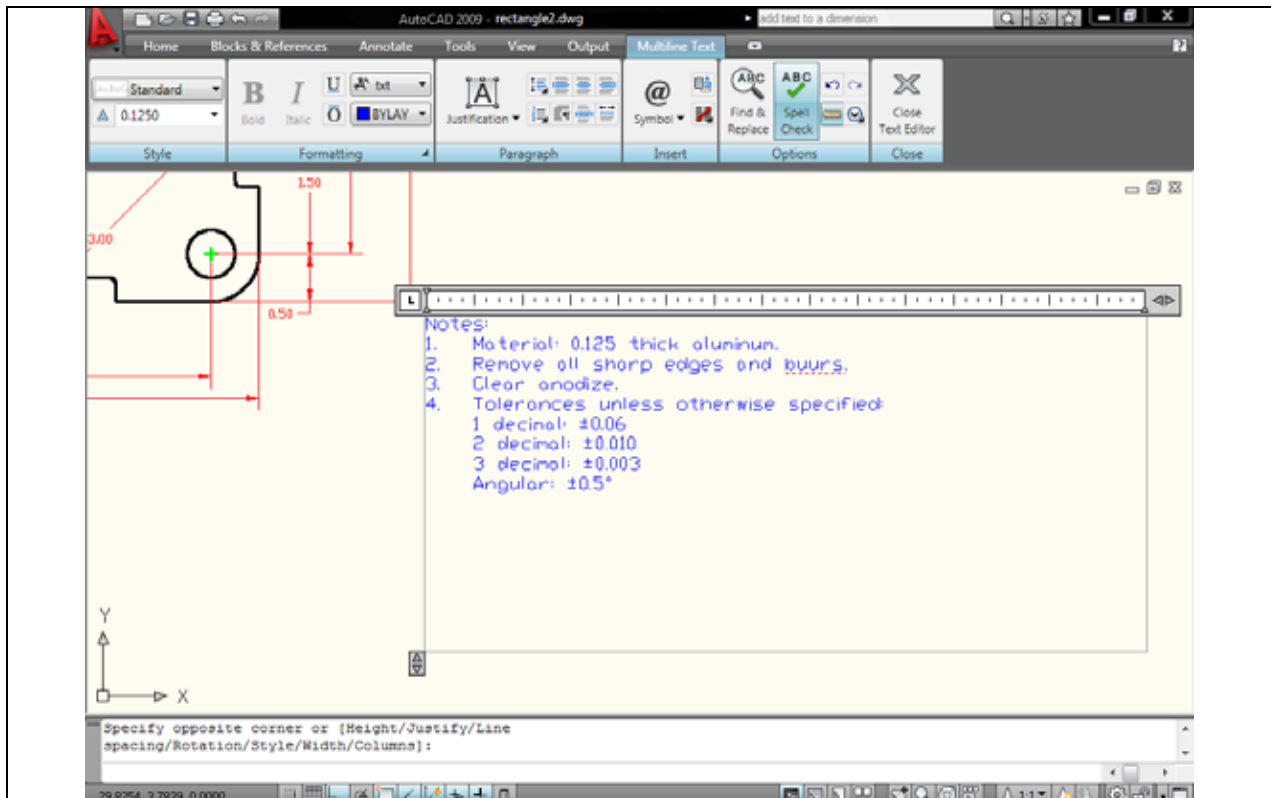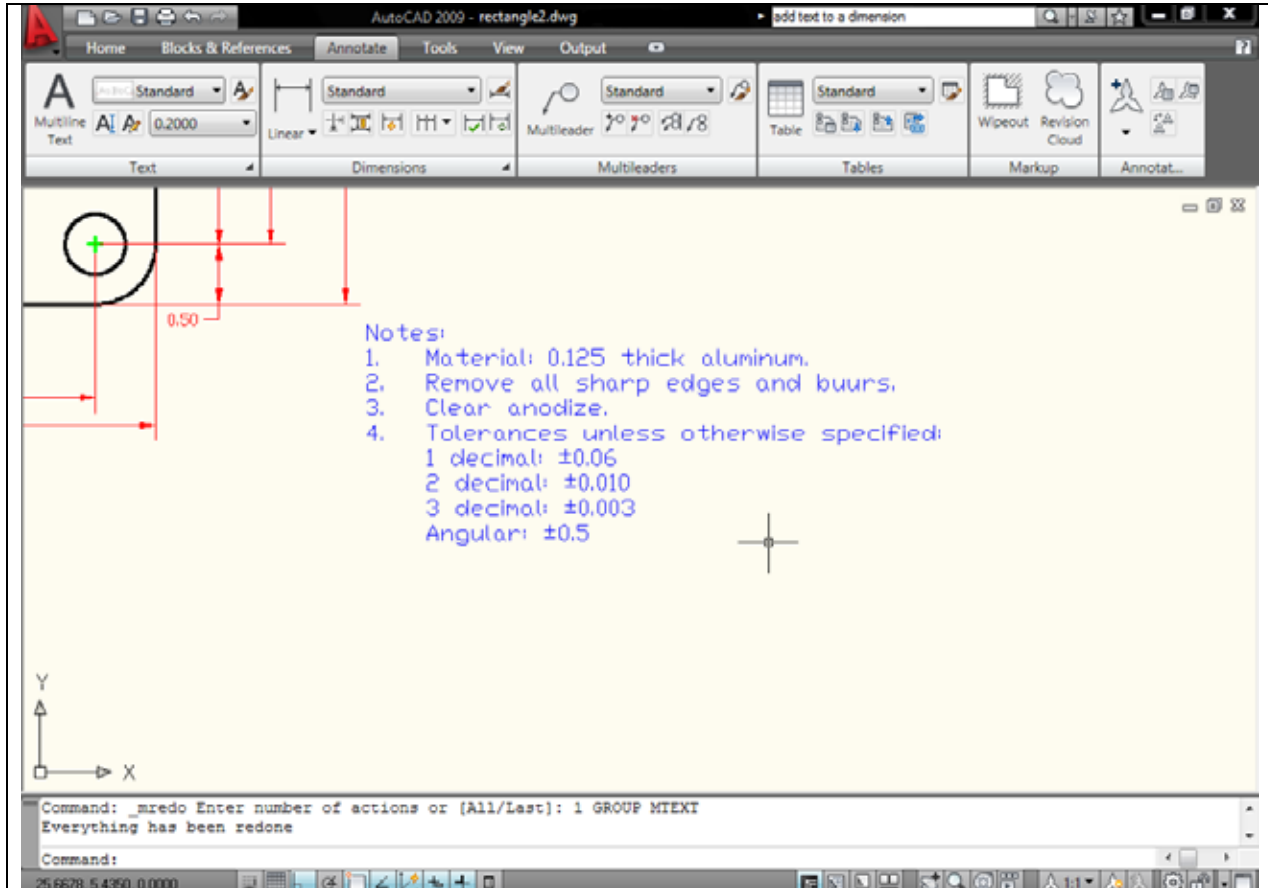
Figure 4.6 – The finished text

Figure 4.7 – Burrs is misspelled in the notes

Checking the Spelling of the Notes Using the Spell Command
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You will see in Figure 4.7, the spelling of "burrs" (BUURS) is incorrect. To correct spelling errors in the drawing, type "Spell" at the command line and hit ENTER. This is the first time you will start a function at the command line. (you can also select the Spell Check tool from the Options panel under the Multiline Text tool or the Text panel under the Annotate tab.) When you type Spell at the command line, you will be prompted to decide where to check spelling; select Entire Drawing and hit Start. 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; 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. (Figure 4.8).
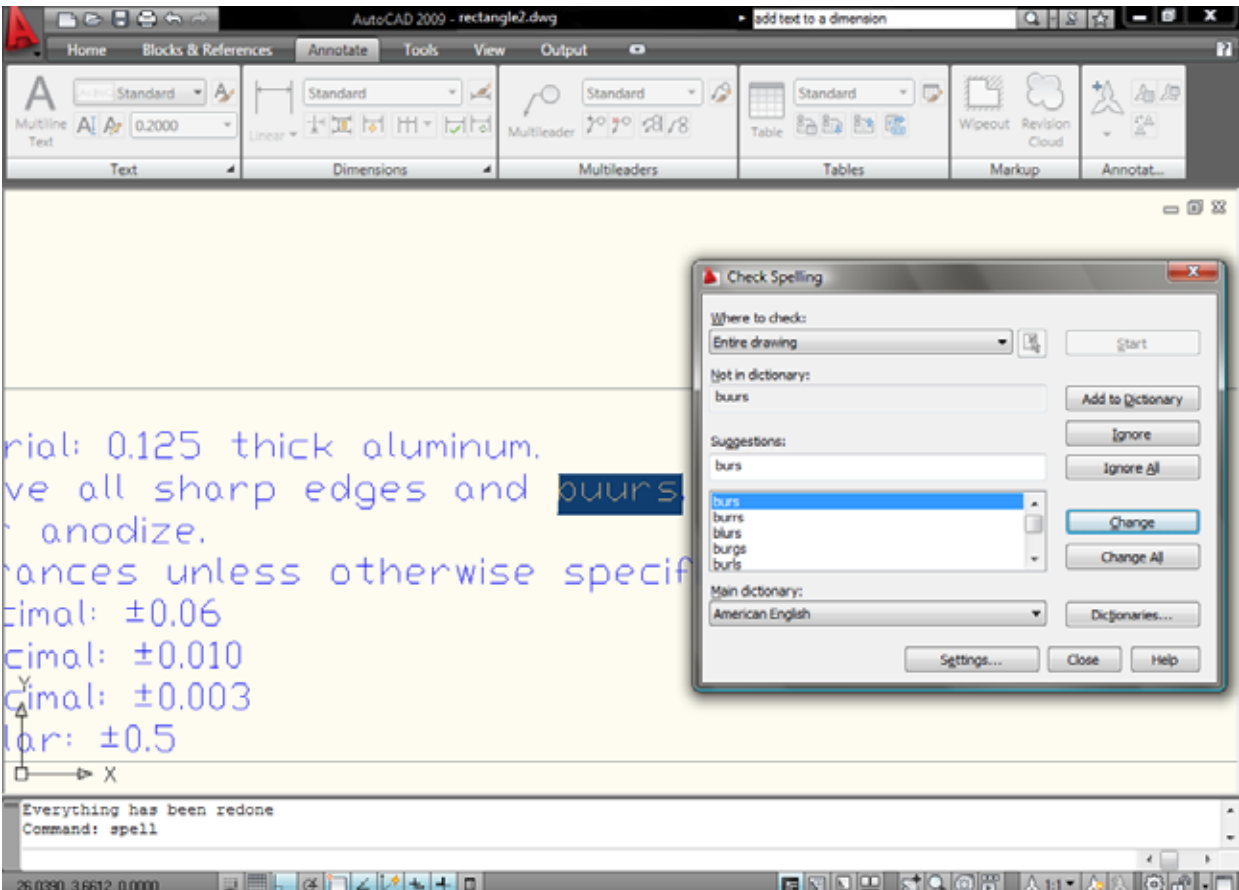
Figure 4.8 – Fixing the misspelled word using the Check Spelling window

Saving your Drawing

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To save the drawing, select the Save As tool from the Menu Browser. 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 "Rectangle" in the File Name text box. The ".dwg" suffix is automatically added to the name when you save the drawing.
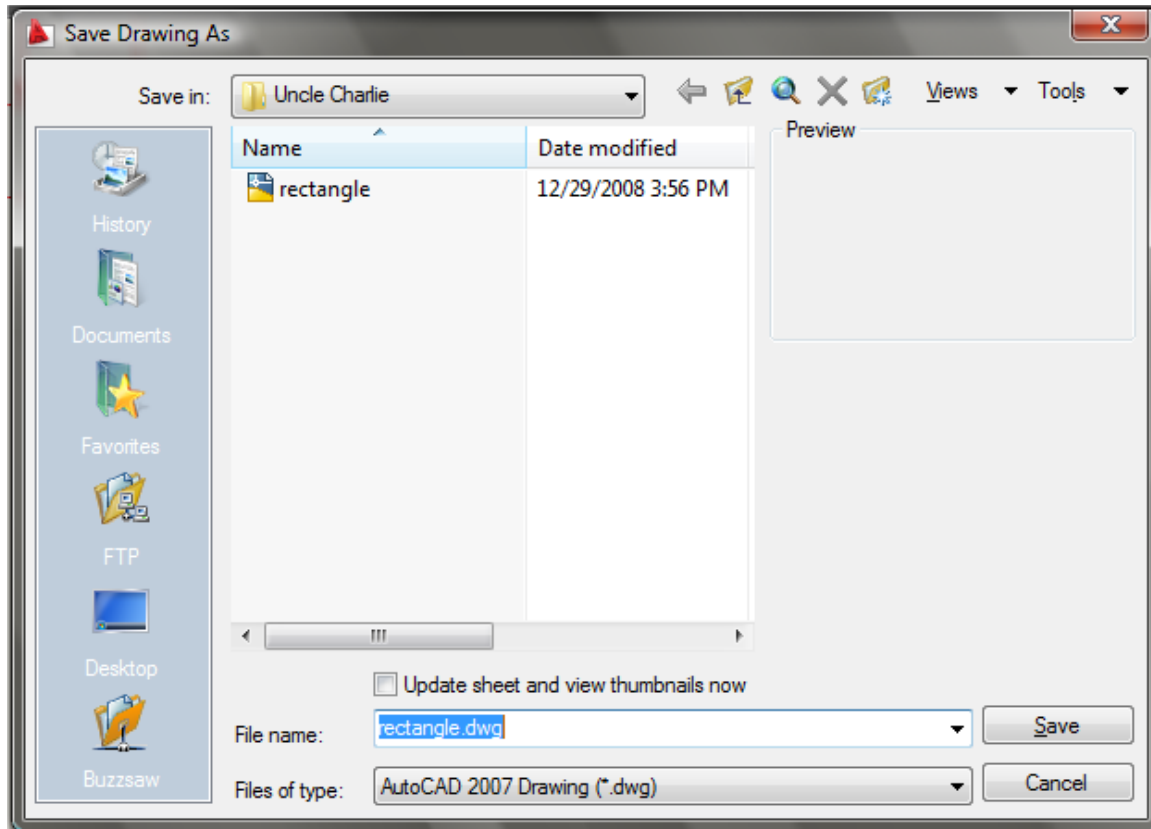
Figure 4.9 – Saving the drawing as rectangle.dwg

* World Class CAD Challenge 02-09 * - Close this drawing file. Create a New file and draw the entire rectangular 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 Circular 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