

Chapter

4

Placing Notes on the Rectangular 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 command 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 Pan Command to Change the View

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

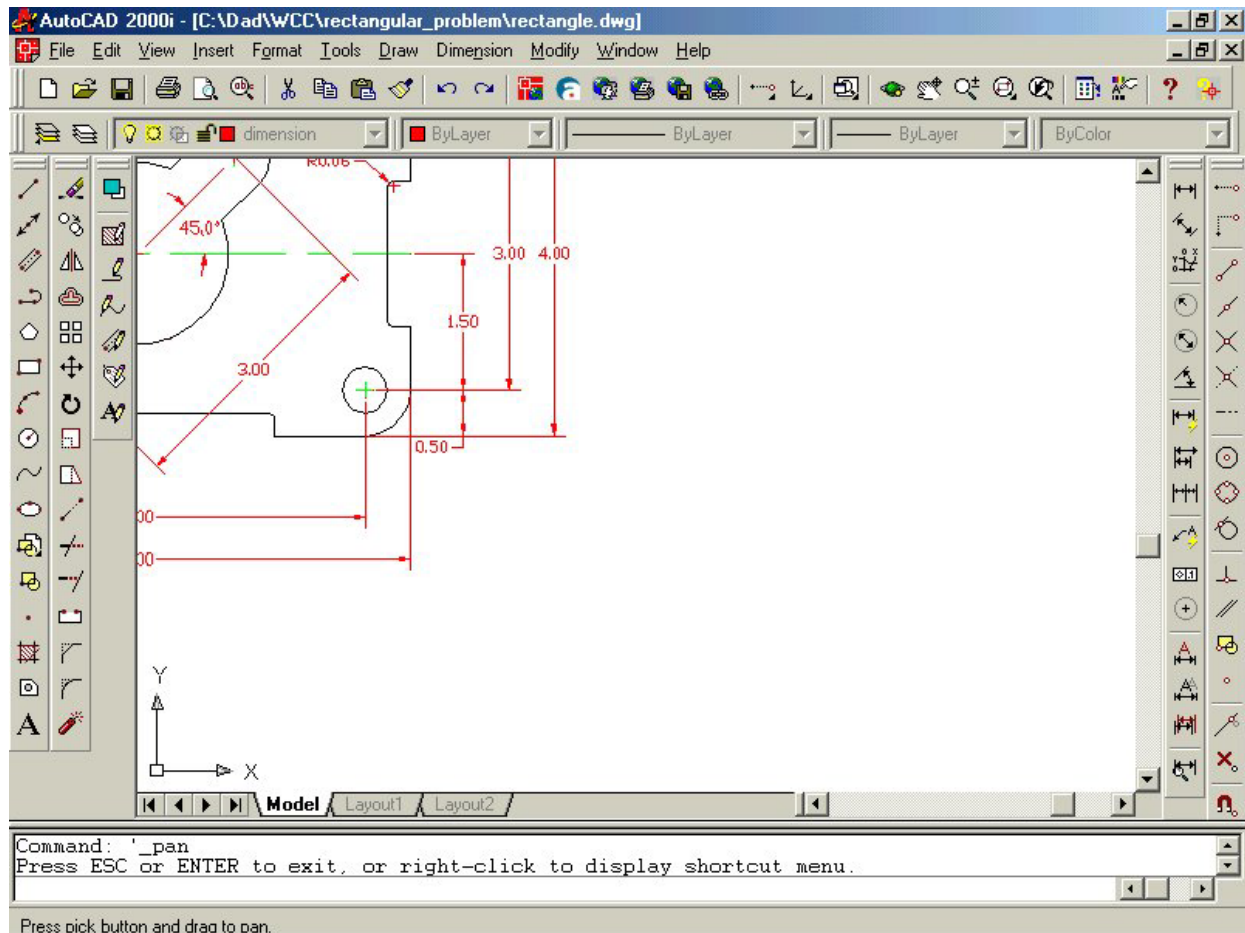


Figure 4.1 – Changing the view in the drawing using the Pan command

Changing Layers to Place Text on the Text Layer

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 4.2). 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.

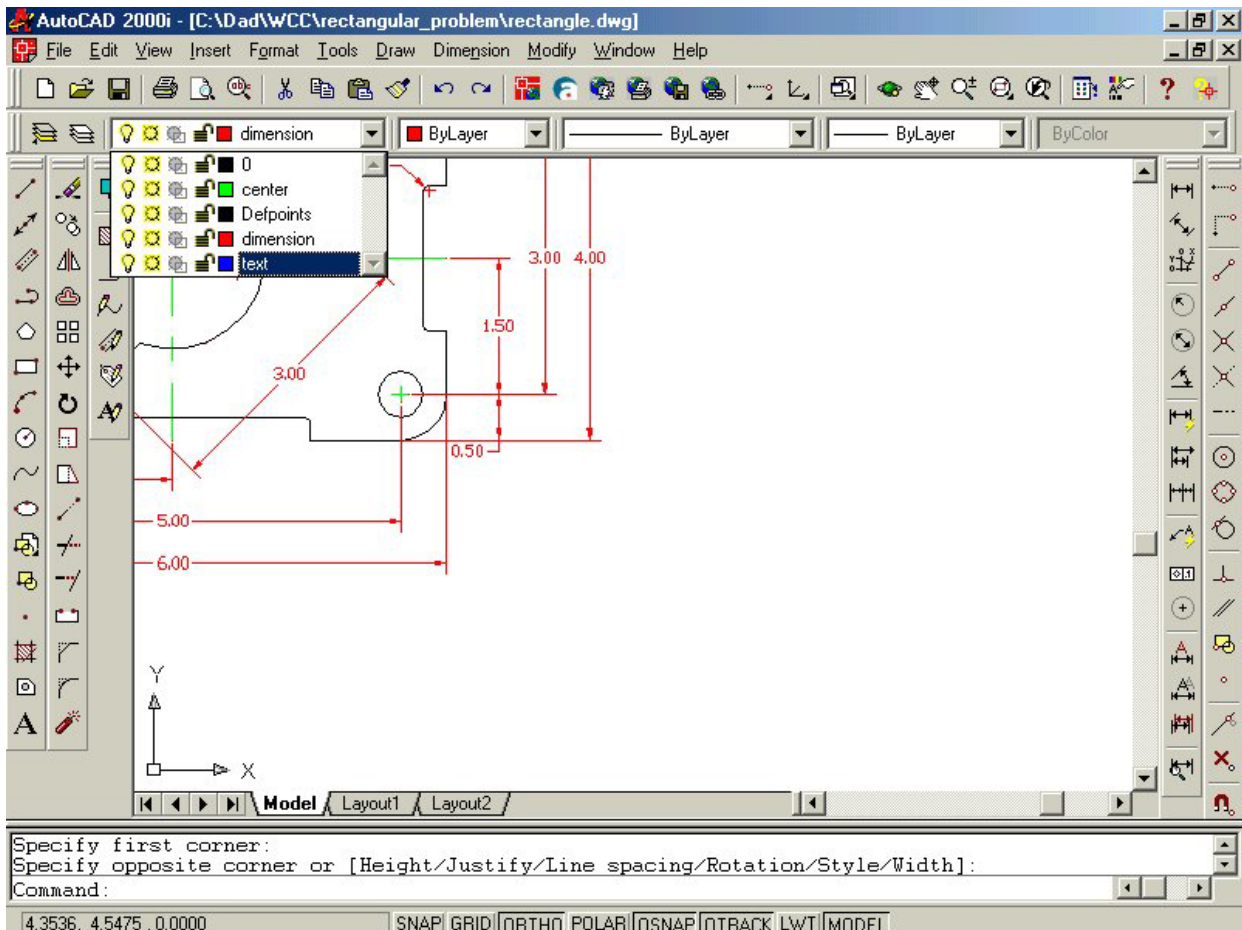


Figure 4.2 – Making the Mtext text window on the drawing

Using the Mtext Command to Place Text in the Drawing

Most drawings you work on 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, 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 speed along 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 aesthetics 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

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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 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 4.3). The command line will prompt you to specify the opposite corner. Make the Mtext window as large as you can as in Figure 4.3.

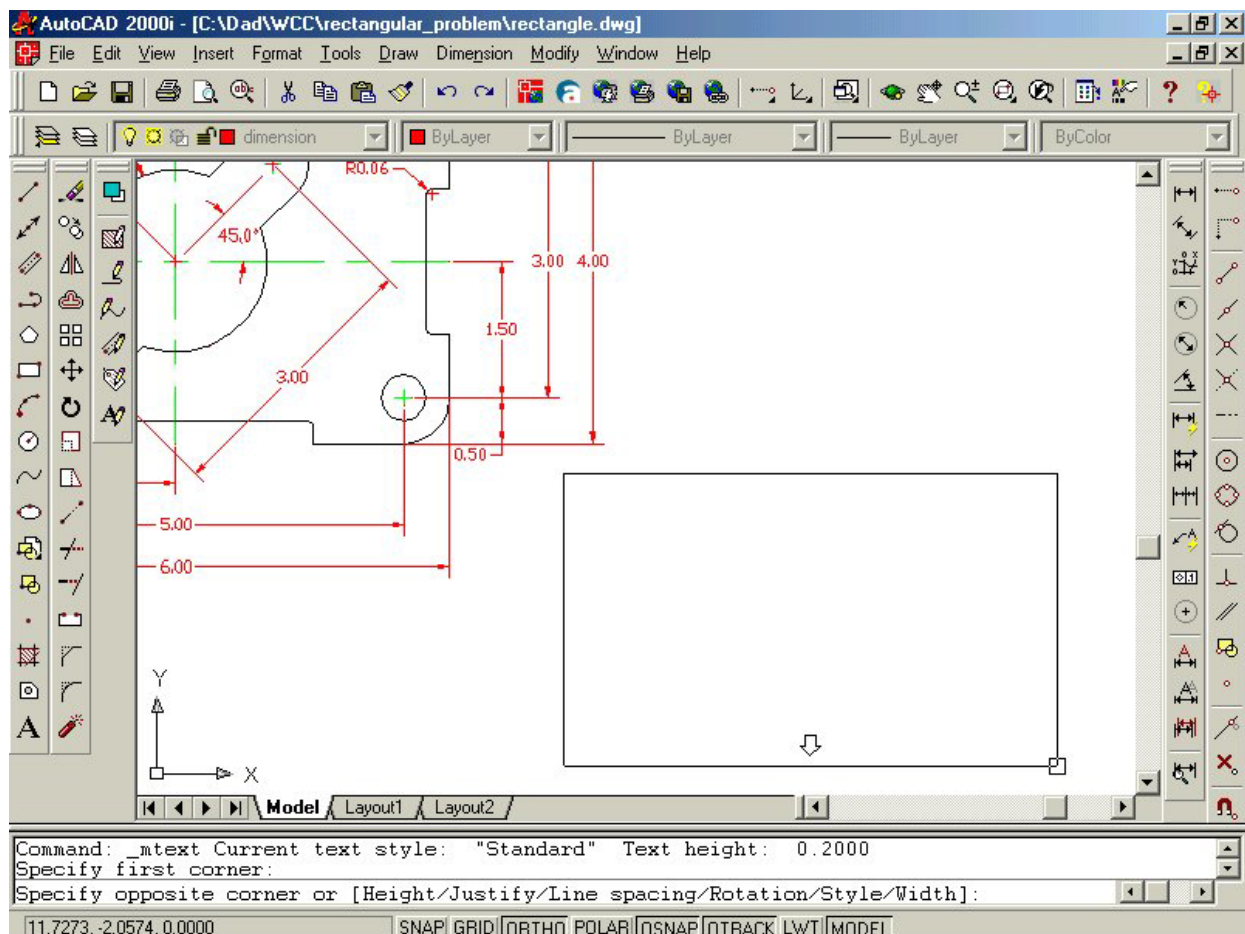


Figure 4.3 – Making the Mtext text window on the drawing

The Multiline Text Editor window will appear in the center of the display. Before you start typing, select the 0.2000 (Figure 4.4), the default text height on the Character tab and type 0.125 for the new text height. Next, 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.

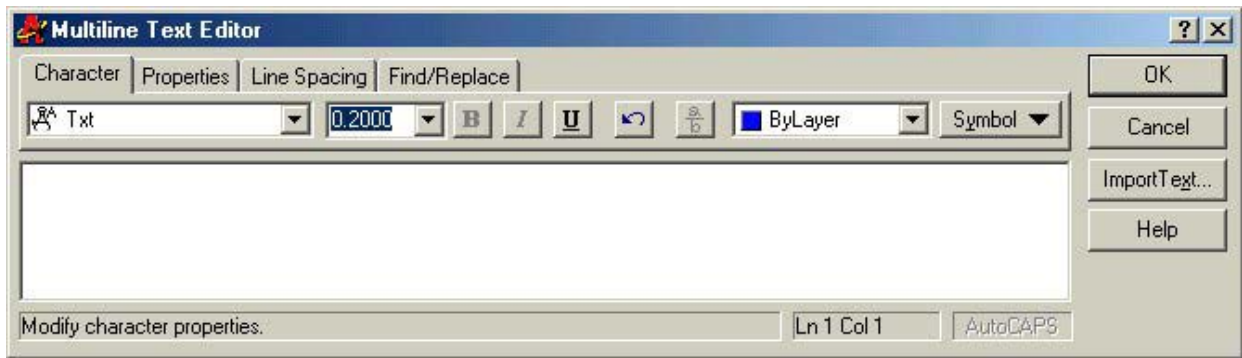


Figure 4.4 – The Multiline Text Editor window

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: $\pm 0.5^\circ$

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 “0.06” (Figure 4.5).

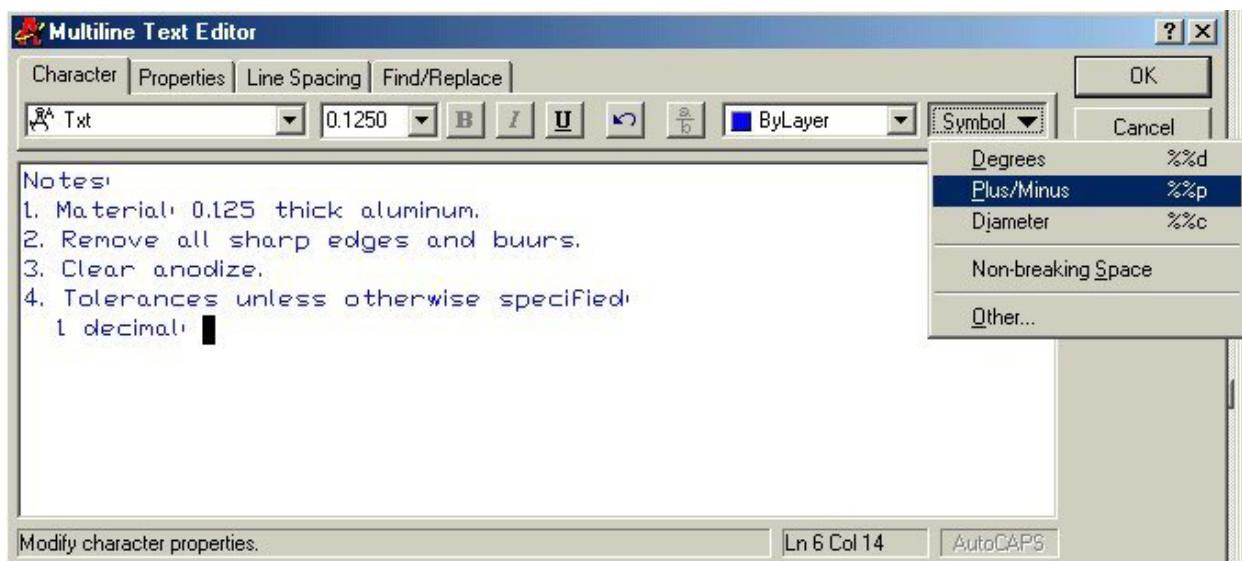


Figure 4.5 – Adding the plus or minus symbol in the Mtext window

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 ° symbol after typing “0.5” (Figure 4.6).

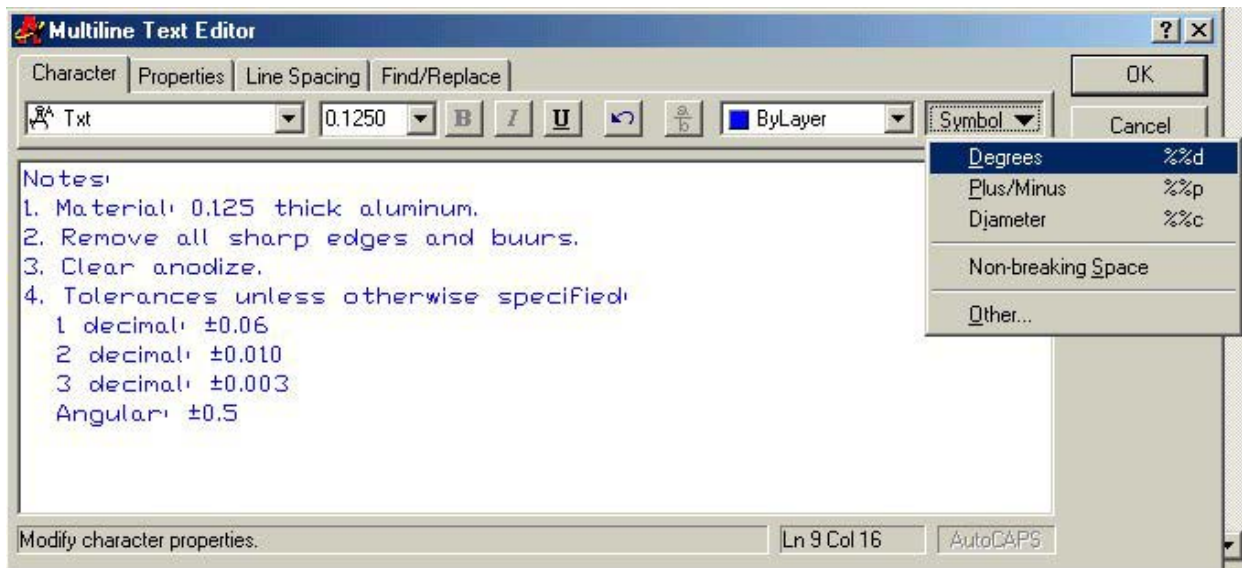


Figure 4.6 – Adding the degree symbol in the Mtext window

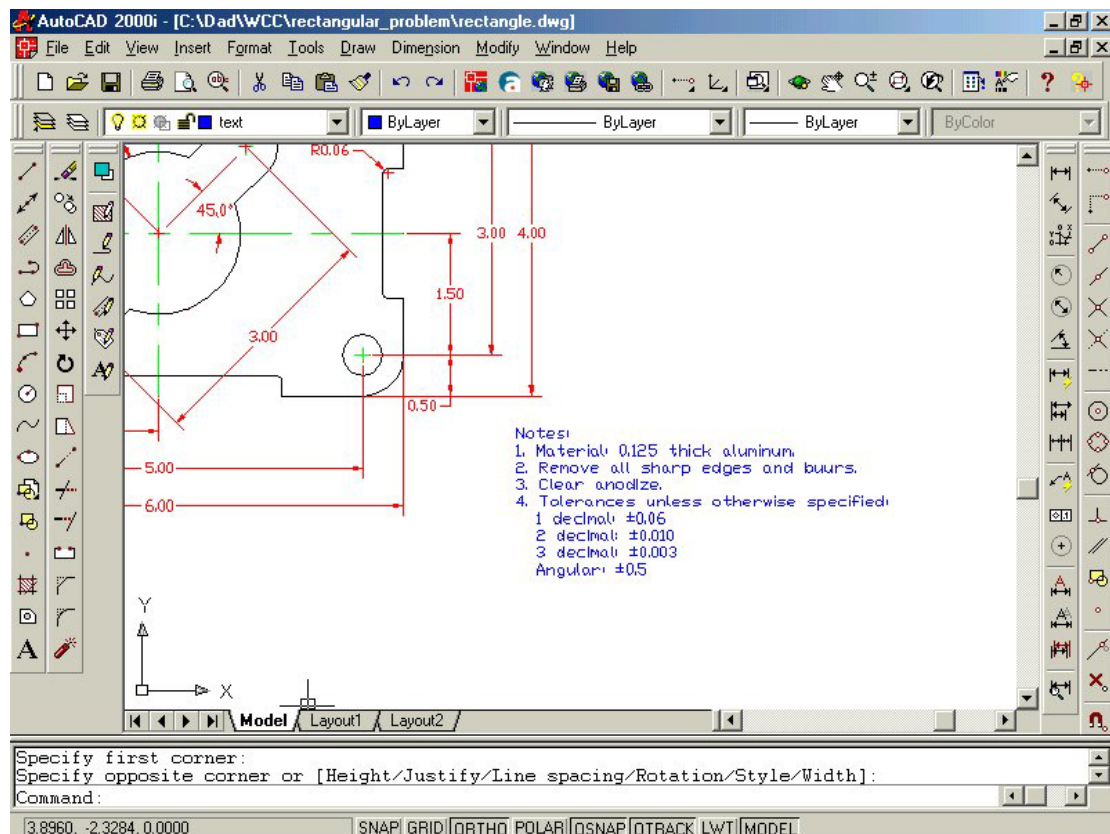


Figure 4.7 – Burrs is misspelled in the notes

Checking the Spelling of the Notes Using the Spell Command

You will see in Figure 4.7, the spelling of “burrs” (BUURS) is incorrect. To correct spelling errors in the drawing, type “Spell” and ENTER at the command line. This is the first time you will start a function 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 “burrs” was typed in the window and the Change button was pressed to correct the notes (Figure 4.8).

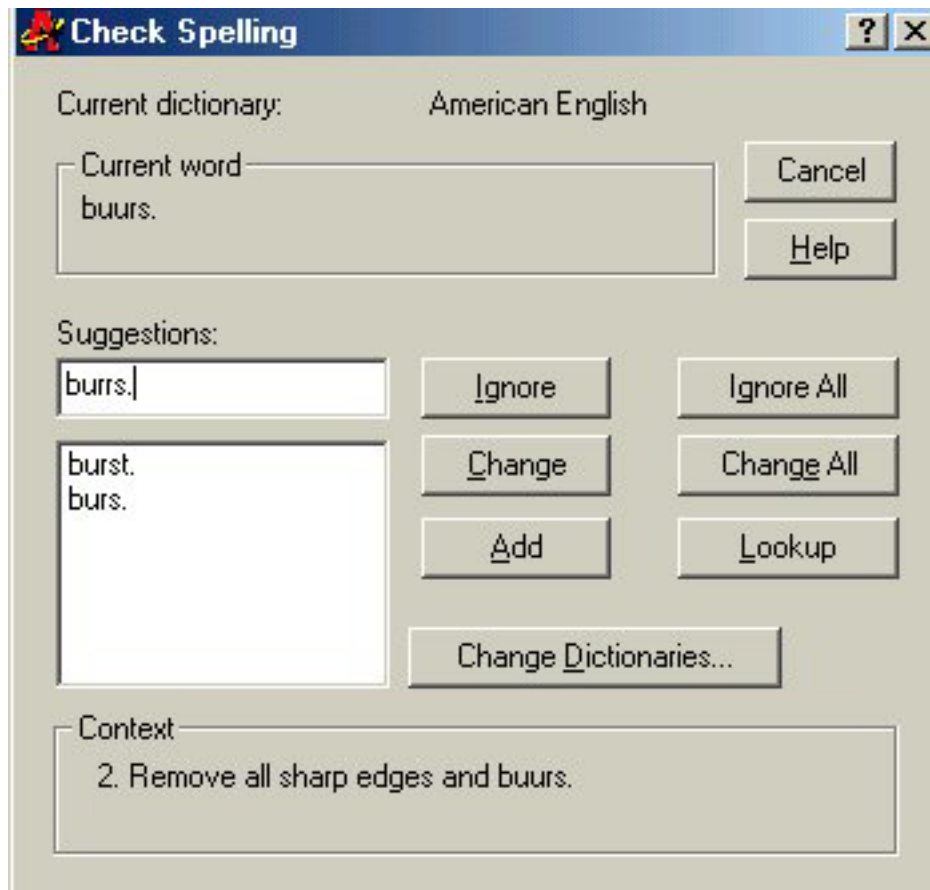


Figure 4.8 – Fixing the misspelled word using the Check Spelling window

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 “Rectangle” in the File Name text box. The “.dwg” suffix is automatically added to the name when you save the drawing.

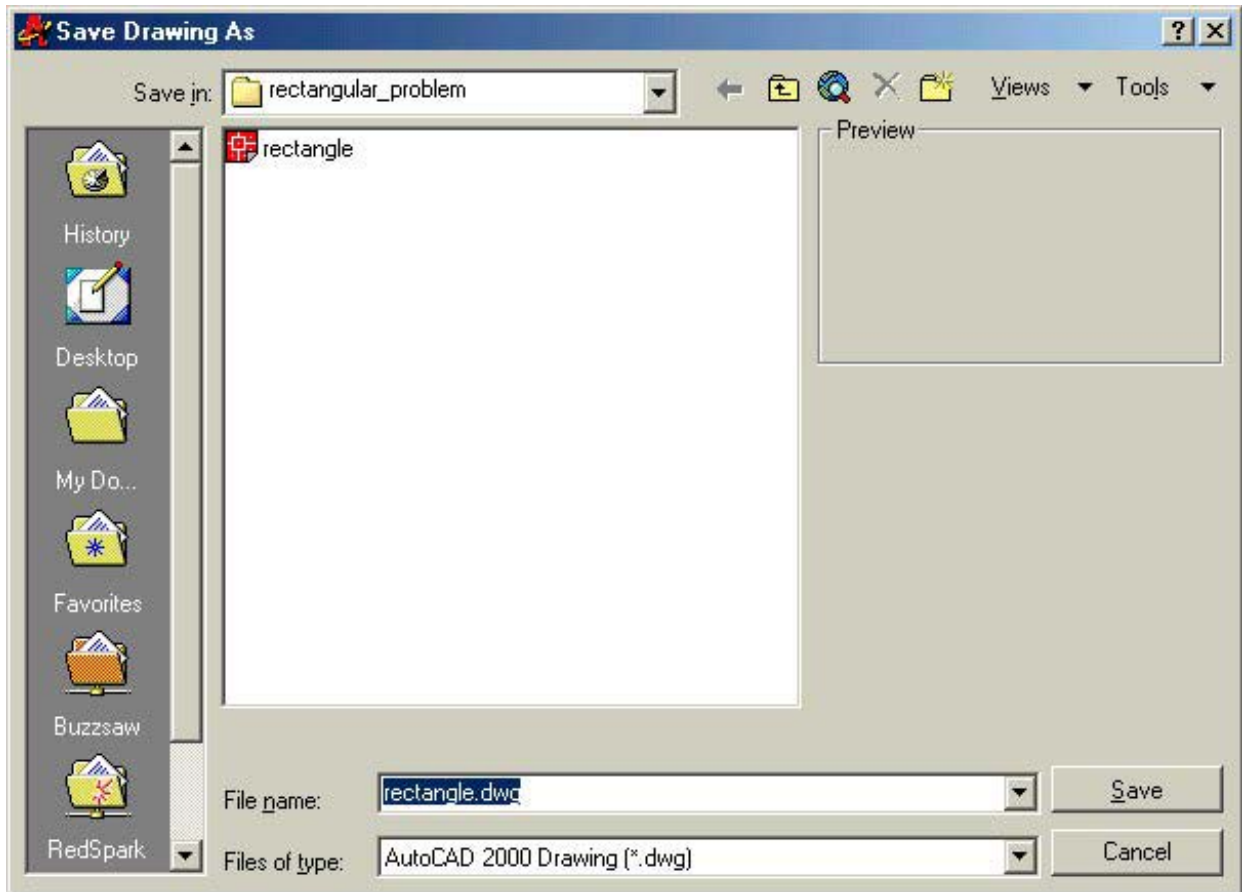


Figure 4.9 – Saving the drawing as rectangle.dwg

*** World Class CAD Challenge * - 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

