C h a p t e r

12

Making Standard Note Blocks and Placing the Bracket in a Drawing Border

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

1. Making Standard Mechanical Notes
2. Using the Mtext Command to Place Text in the Drawing
3. Creating a Standards Note Drawing with the WBlock command
4. Moving from Model Space to Paper Space
5. Inserting a B-Size Drawing Border into the Paper Space Layout
6. Inserting a Standard Notes into the Paper Space Layout
7. Modifying Standard Notes in Paper Space

Making Standard Mechanical Notes

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Pick the New Drawing tool on the Standard toolbar and the Startup window will appear. You will begin the drawing by selecting the template you created in the previous assignment. In Figure 12.1, the window "Create New Drawing" is loaded by selecting the New tool on the Standard Toolbar. You can select the "Use the Template" (third button) at the top of the Create New Drawing window. A list of templates is loaded in the Select a Template list box. Scroll down the list and select Mechanical.dwt.
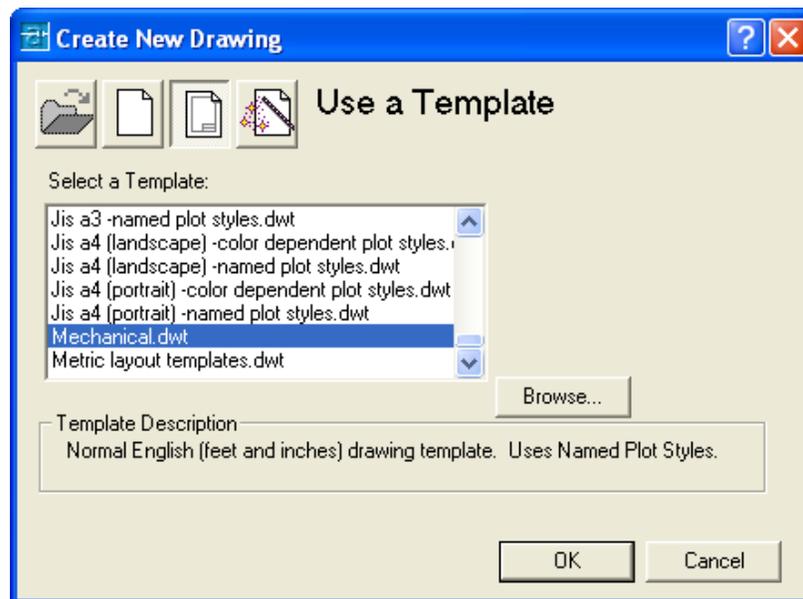
Figure 12.1 – Starting with the Mechanical Template

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. 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.

Using the Mtext Command to Place Text in the Drawing

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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 then the command line will prompt you to specify the opposite corner. Make the Mtext window at least a few inches wide.

The Multiline Text Editor window will appear in the center of the display. Remember in the Mechanical template the text height is already set at 0.125 and the font is already set 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.

Notes:
1. Material:
2. Remove all sharp edges and burrs.
3. Corrosion Protection:
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 button in the Multiline Text Editor. A menu will appear and select Plus/Minus to add the ± symbol before typing "0.06". 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".

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. You can Ignore the word or even Add the word to the dictionary by pressing the appropriate button in the Check Spelling window.
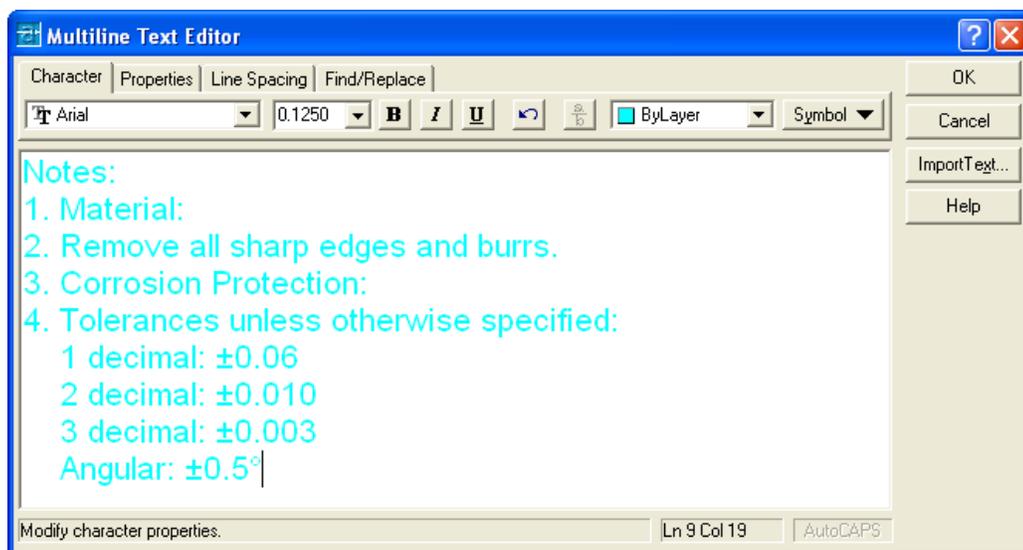
Figure 12.2 – Making a Set of Standard Notes

Creating a Standards Note Drawing with the WBlock command
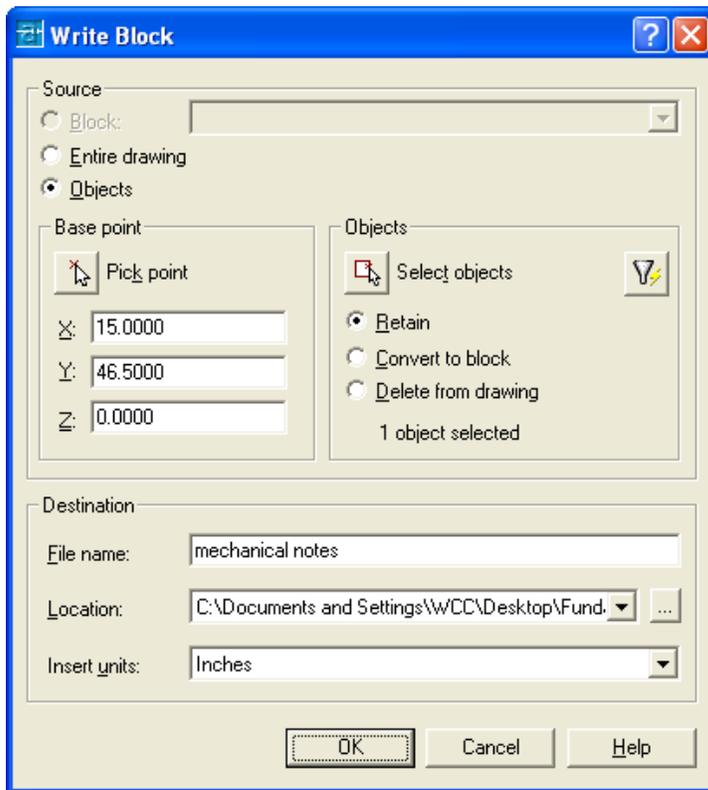
Figure 12.3 – Writing the Notes to a New Drawing

To save the drawing, type WBLOCK at the Command line and the Write Block window will appear in your graphical display as shown in Figure 12.3. To save the information in your Fundamentals of 2D Drawing Folder, type Mechanical Notes at the File Name textbox. Then select the button with the three dots to the right of the Location textbox and browse for the Fundamentals of 2D Drawing Folder. Choose the Pick Point for the Base Point option and select the entry node with the Insertion Object Snap as shown in Figure 12.4. Next pick the Select Objects button and pick the notes just typed. Now pick the OK button and a new drawing called Mechanical Notes is in your Fundamentals of 2D Drawing folder.

Figure 12.4 – Selecting the Insertion Point of the Text with the Insertion Object Snap

The Multiline Text Editor window will appear in the center of the display. Remember in the Mechanical template the text height is already set at 0.125 and the font is already set 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.

Notes:
    1.   Material:

2. Remove all sharp edges and burrs.
3. Corrosion Protection:
4. Tolerances unless otherwise specified:
     0 decimal: ±1.5
     1 decimal: ±0.25
     2 decimal: ±0.10
     Angular: ±0.5º
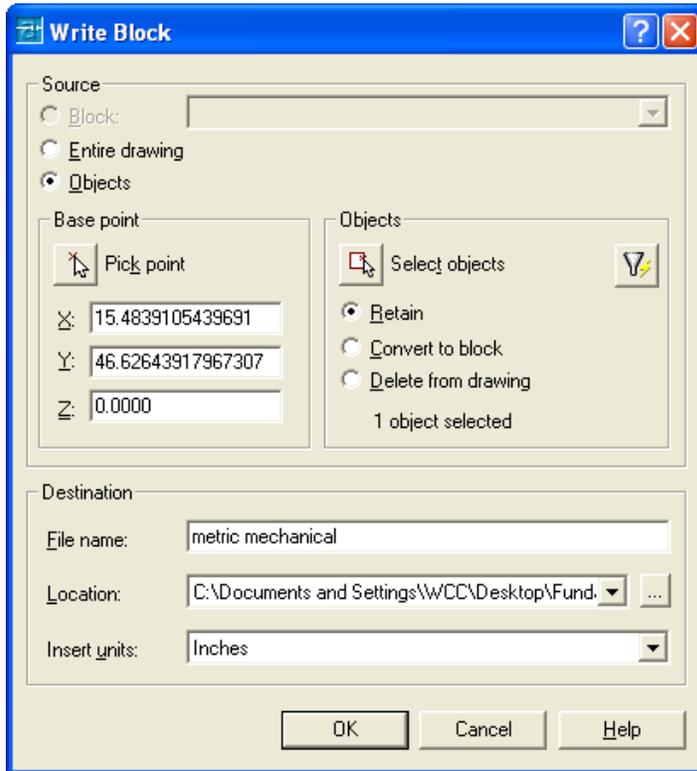
To save the drawing, type WBLOCK at the Command line and the Write Block window will appear in your graphical display as shown in Figure 12.3. To save the information in your Fundamentals of 2D Drawing Folder, type Metric Mechanical Notes at the File Name textbox. Then select the button with the three dots to the right of the Location textbox and browse for the Fundamentals of 2D Drawing Folder. Choose the Pick Point for the Base Point option and select the entry node with the Insertion Object Snap. Next pick the Select Objects button and pick the notes just typed. Now pick the OK button and a new drawing called Metric Mechanical Notes is in your Fundamentals of 2D Drawing folder.

Figure 12.5 – Writing the Metric Notes to a New Drawing

Moving from Model Space to Paper Space

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The drawing border and notes will be added to the drawing in Paper Space, which is an area of the drawing file that allows the CAD operator to place the drawing from Model Space into an A, B, C or D size border. You can open a single window or multiple windows from Paper Space scaling each view to whatever proportion that you desire, enabling you to show any detail in perfect clarity that is desired. The best application of the Paper Space technology is when an Engineering Designer draws a solid part in Model Space and then in the Paper Space Layout has the front, side and top views in a single drawing, when they open three different windows to show each perspective. Many Architectural companies will have their D-size border in Paper Space set to print easily to their printer or plotter, and the architect can easily scale the floor plan in the window. Where one business is making the printing an easy step without scaling the drawing border, the mechanical organization is only allowing the part to exist in Model Space so that the part is inserted into an assembly drawing without notes or

dimensions. When you learn the World Class CAD, Fundamentals of 3D Drafting techniques, the more advanced techniques of Model and Paper Space interaction will come clearly into understanding. Presently, in the Bracket problem, you will use Layout 1 tab of Paper Space to setup the drawing border for printing.

In Figure 12.6, you can see the Bracket problem with the Model Space tab set right above the status bar. To enter Layout 1, then just select the Layout 1 tab. The very first time you select the tab, the Page Setup - Layout 1 dialogue box will open. In the future, you can add these steps or procedures to your template file and the Page Setup – Layout 1 window will not appear since the choices you need to make all already made. Also, even though the Page Setup window appears during the initial jump into Paper Space when the template has not been set, after the printer, paper size and scale is chosen, any transitions from one mode to another will be immediate and the setup queries are no longer required.
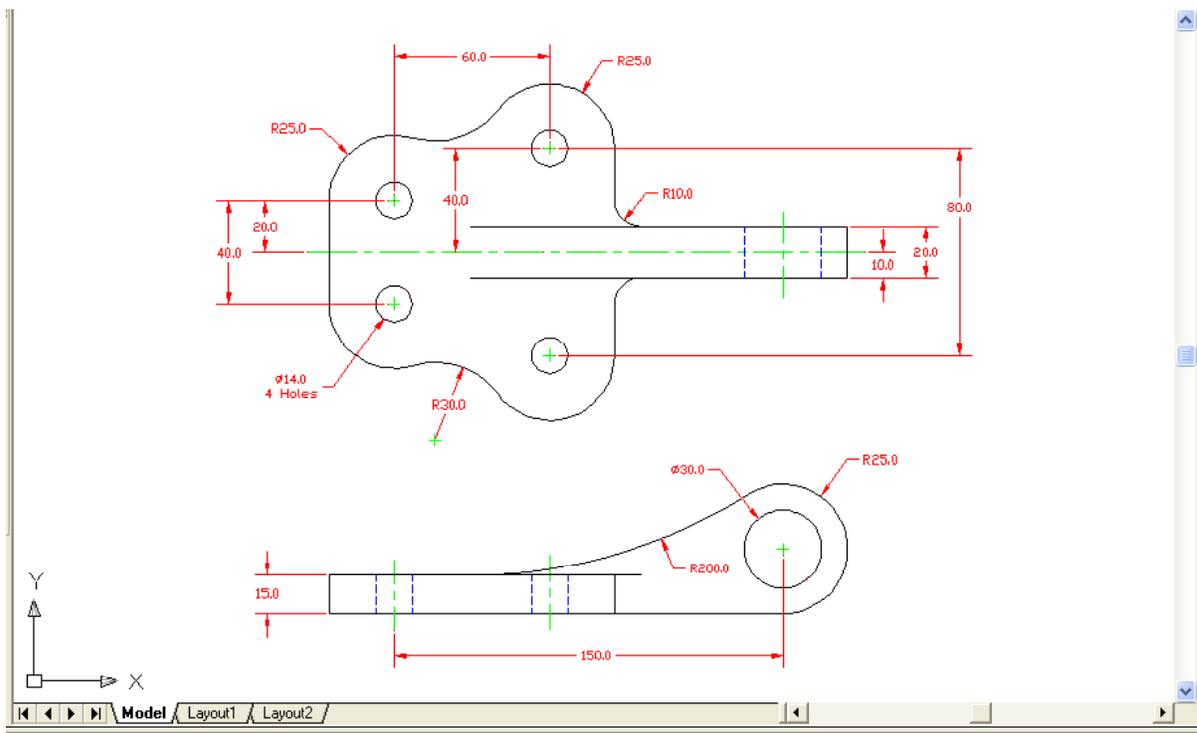
Figure 12.6 – Going to Select Layout 1 and Enter Paper Space

The first tab in the Layout window is to name the plot device. At home you probably just have a letter sized printer and a stock of 8.5 x 11or even longer legal size paper. By picking your printer in that case, you can run these smaller test plots using the A size paper for the A and C drawing borders and the legal cut paper for the wider B and D size borders. After checking the print outs, there are many companies that provide CAD printing services at very economical prices when larger layouts are needed for the customer. For those of you who are at an organization with the abilities to print any A, B, C or D size drawings, then you can have a 8.5 x 11 printer showing up in Name window that is intended for letters and you will notice one, two or maybe more printers and plotters available for the larger outputs. Always consult with the department manager to which printers are accessible for your use. In many professionals groups, there are black and white machines for check prints and color devices

for finished drawings. Some printing equipment have special paper that is costly, so again make sure that you pay attention to any orientation to special tools in your list of offered devices, and keep any special instructions posted in your work space, so not to waste resources or time. (See Figure 12.7)
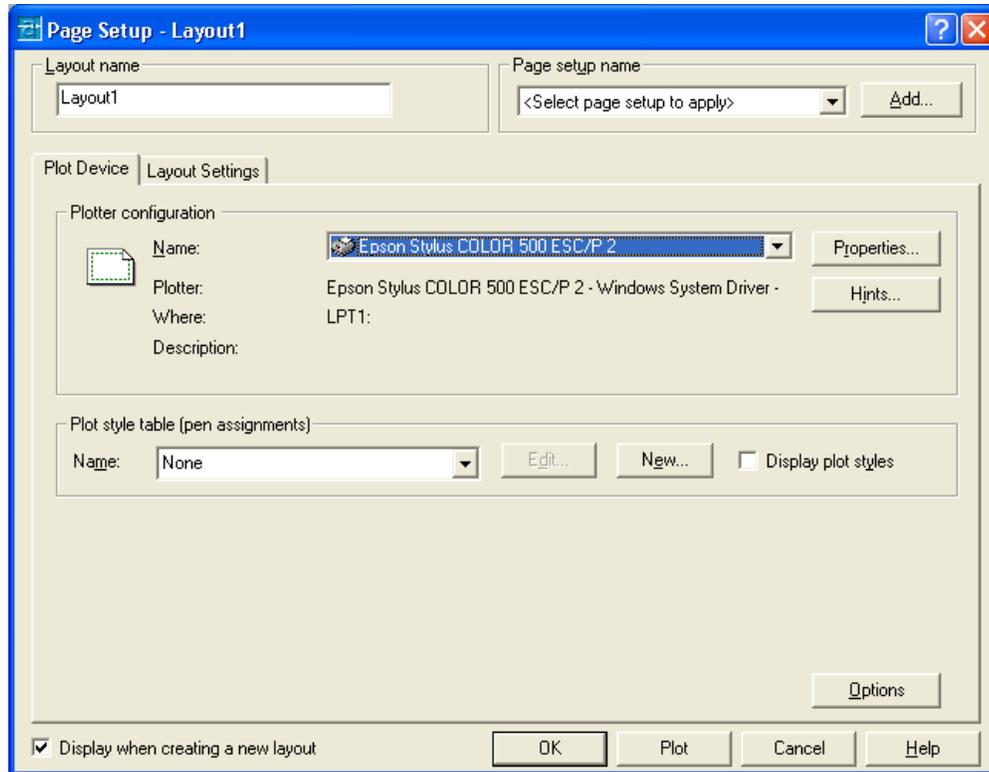
Figure 12.7 – Selecting a Printer to Plot Your Drawing

So you can choose your printer in the Name window and for the present you will ignore the Plot Style Table for pen assignments. Go ahead and select the Layout Settings tab and there is an unchangeable text label on this page. You will have to return to the Plot Device page to change the print device. (See Figure 12.8)

Next, since you will be printing the drawing on A size (8.5 x 11), legal or B size (11 x 17) paper choose you paper size in the Paper Size window list. Every document size your printer supports is listed in the list box.

Since the Bracket exercise is a metric drawing, select the **mm** option button and the printable area will report in millimeters instead of inches.

Continuing with the queries into output options, in the Drawing Orientation frame, select the **Landscape** option button to rotate the image within the wider paper size of the B-size border. Again, for this step, you must know your plotting or printing equipment or use the Print Preview to check the drawing's direction.

Subsequently, the Plot Area can remain at Layout. On the Plot Scale frame of the Page Setup window the scale for legal paper is 0.79 to 1. When entering special scales in the Custom textboxes, the scale is set to "Custom" automatically.
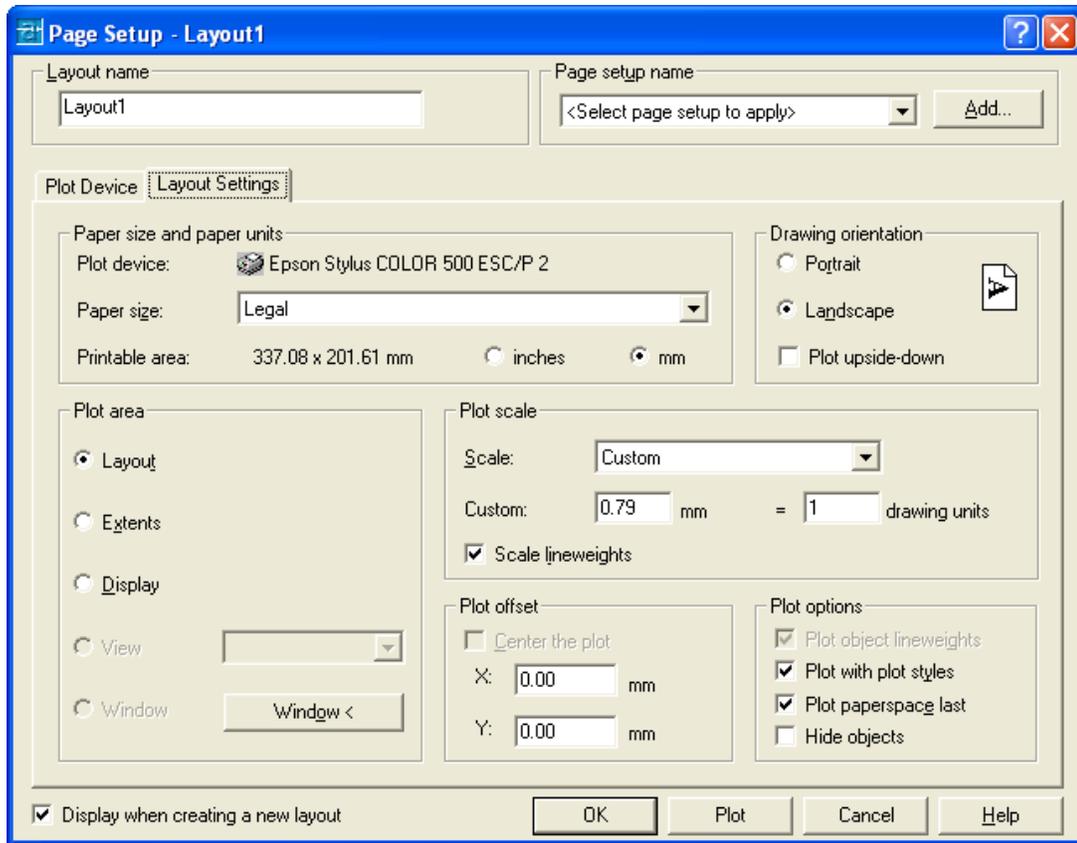
Figure 12.8 – Picking a Paper Size and Setting the Drawing Scale

To find the ratio for the custom scale, the y-axis of the printable area is 201.61 mm and the B-size drawing border of 10 inches or 254 mm will not fit into the smaller height. By using your calculator you will find that 201.61 divided by 254 is 0.79, when rounded to two decimal places. When using an 11 x 17 paper, your scale should be 1 to 1. You probably do not wish to use a B-size border with 8 x 11 paper, since the proportions of the two do not coincide. If you choose the A-size document to print on, the output width is 260.88 mm, so since the 16 inch B-size border is 406.4 mm wide, take 260.88 and divide by 406.4 to get 0.64, when rounded to two decimal places. The custom scale is 0.64 mm = 1 drawing unit. After you make your choice, press the OK button and you will enter Paper Space for the first time with the entire Bracket and dimensions appearing in a window looking into Model Space. (See Figure 12.9)

So how can you determine that you are in a different dimension of the AutoCAD software? First, after selecting the paper size, you will see a representation of the white document sitting on a grey background. Secondly, a dotted rectangle covers the white area which displays the printable area of the document. Third, the UCS symbol is no longer the arrows pointing towards the x and y-axis, but is now a triangle. Fourth, the entire content in Model Space is inside another rectangle which is the window looking from the Layout 1's two dimensional area into Model Space.
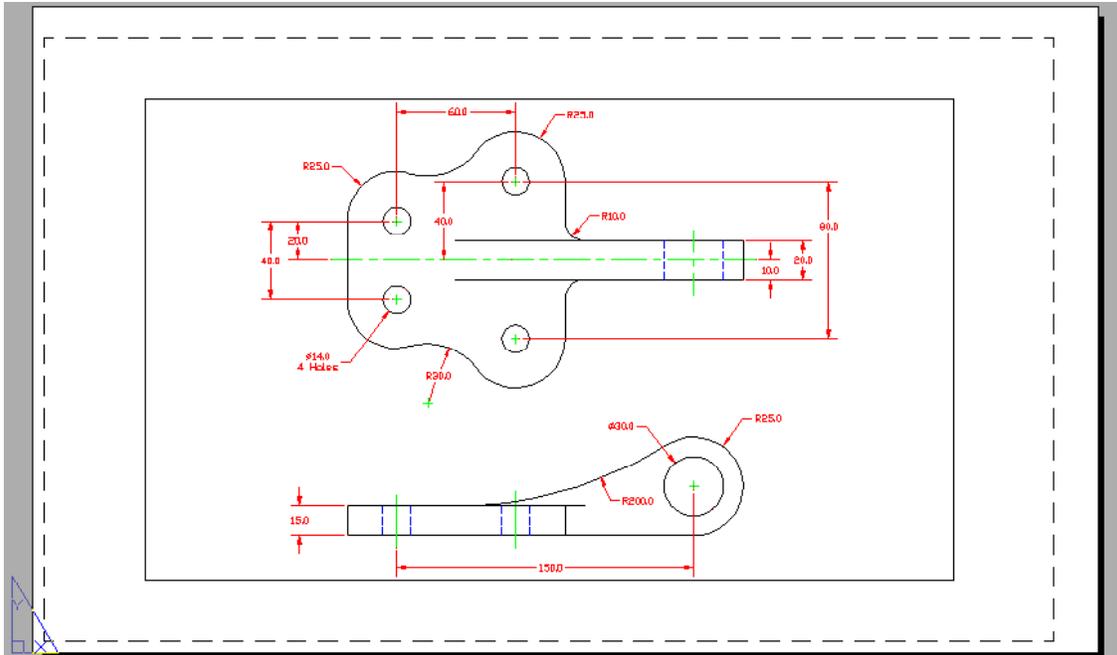
Figure 12.9 – Looking at Paper Space for the First Time

Inserting a B-Size Drawing Border into the Paper Space Layout

To insert the block containing the B-Size Border into Layout 1, select the Insert tool on the Draw toolbar and the Insert window (Figure 12.10) will appear in the graphical display. Use the Browse button to find the "b size border" in your Fundamentals of 2D Drawing folder. Keep the Specify On-screen checkbox checked and also indicate that the Uniform Scale by checking the box. Type 25.4 in the X: textbox since you need to increase the scale for metric because the border was drawn in inches.
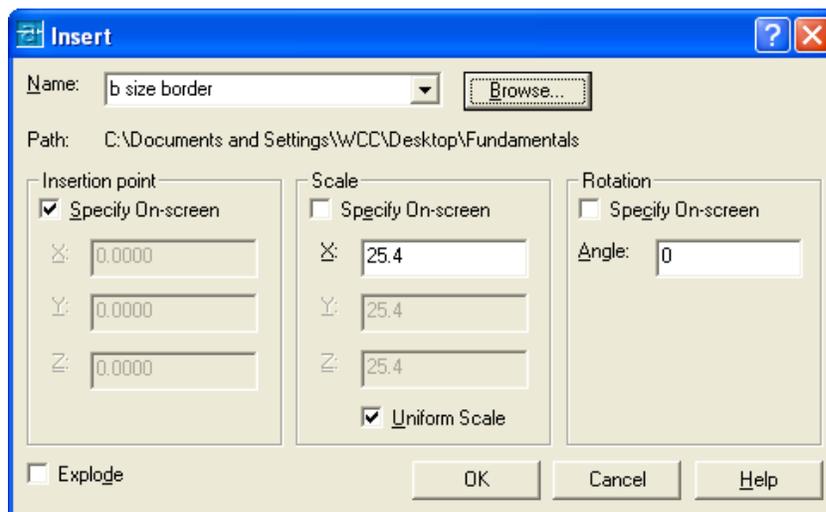
Figure 12.10 – Inserting the B Size Border

After picking the OK button on the Insert window, pick the insertion point for the border, placing the entire contents inside the dotted printable area. You will see in Figure 12.11 that the border is justified to the right side of the dotted rectangle. This leaves a gutter on the right side of a group same sized drawings for stapling.
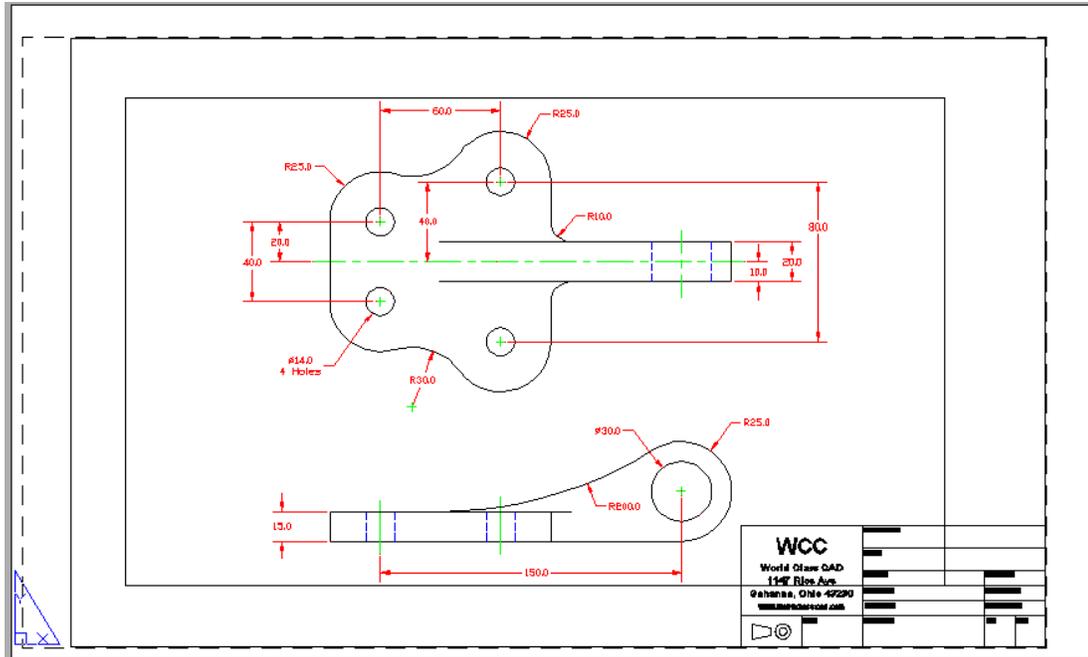
Figure 12.11 – The B Size Border in Paper Space

Now us the Move tool on the Modify toolbar to shift the window containing the Bracket problem to the left as shown in Figure 12.12.
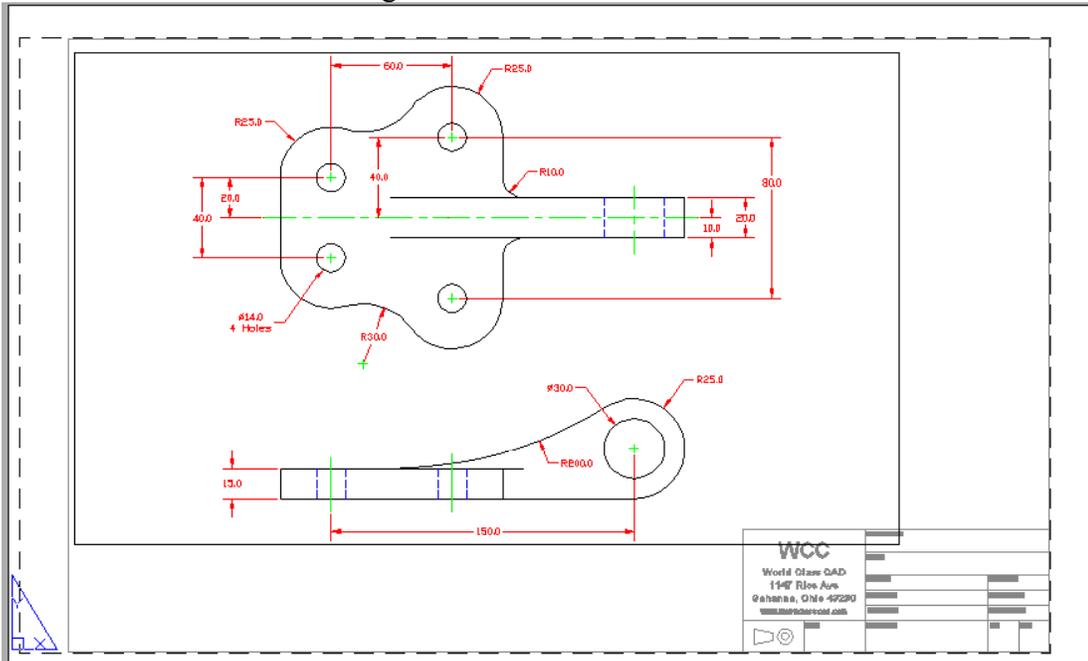
Figure 12.12 – Moving the Model Space Window to the Left

Inserting a Standard Notes into the Paper Space Layout

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To insert the block containing the standard notes into Layout 1, select the Insert tool on the Draw toolbar and the Insert window will appear in the graphical display. Use the Browse button to find the "metric mechanical notes" in your Fundamentals of 2D Drawing folder (See Figure 12.13).
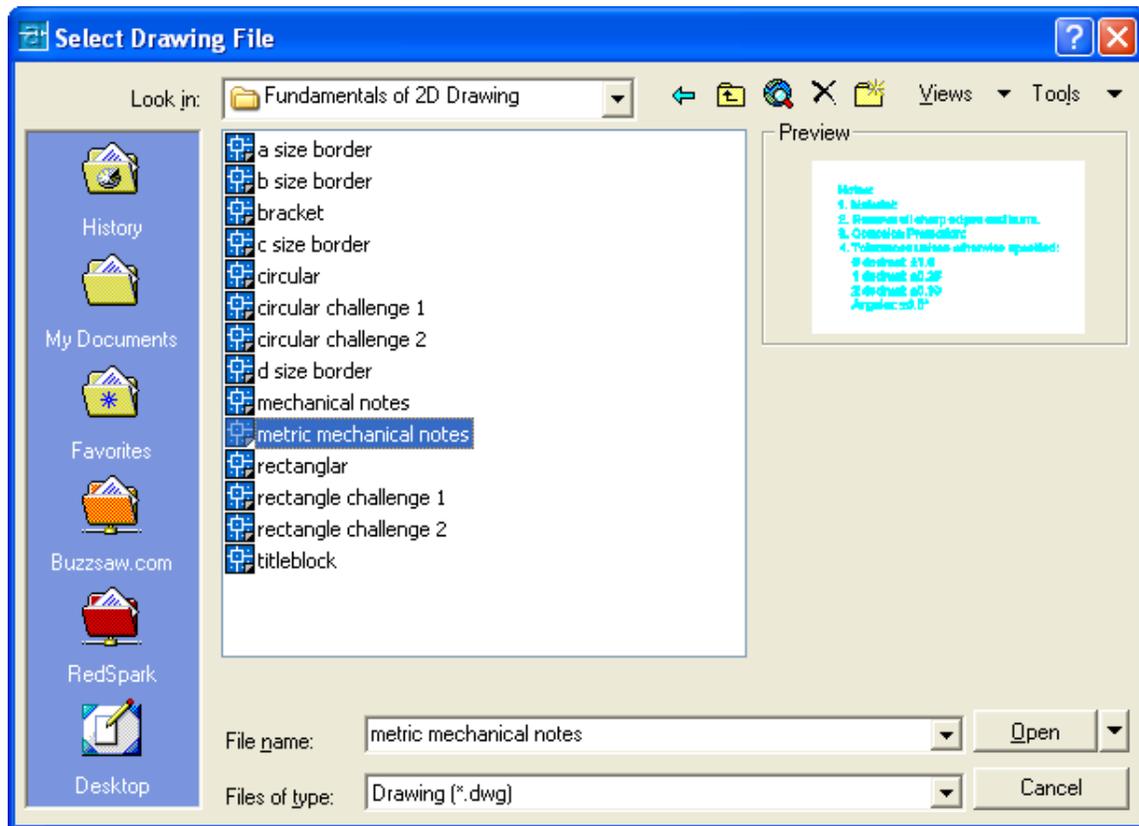
Figure 12.13 – Inserting the Metric Mechanical Notes

The metric notes are already scaled appropriately, so keep the Specify On-screen checkbox checked, and the X: scale to be 1. Check the Explode checkbox so you can modify the standard notes after inserting them. Exploded text will allow you to modify each sentence. You have to rotate text very few instances, so the majority of time the Rotation Angle will remain at zero.
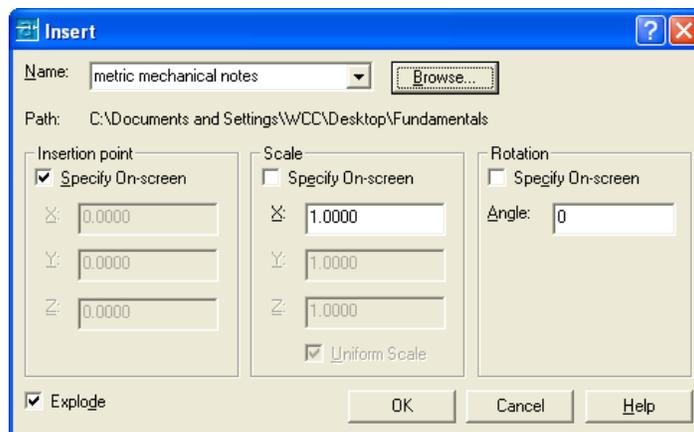
Figure 12.14 – Check Explode in the Insert Window

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After picking the OK button on the Insert window, pick a point to the right of the Bracket problem to place the notes. Next, change the text using the Edit Text tool.

Modifying Standard Notes in Paper Space

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To add text to the standard notes, select the Edit Text tool on the Modify II toolbar. The Multiline Text Editor window will appear in the graphical screen. As you type changes in a Word Processing program make the following changes to notes one and three, while leaving notes two and four alone.

1. Material: 25 mm cold rolled steel

 And

3. Corrosion Protection: zinc plate

Select "OK" to close the window. (See Figure 12.15)
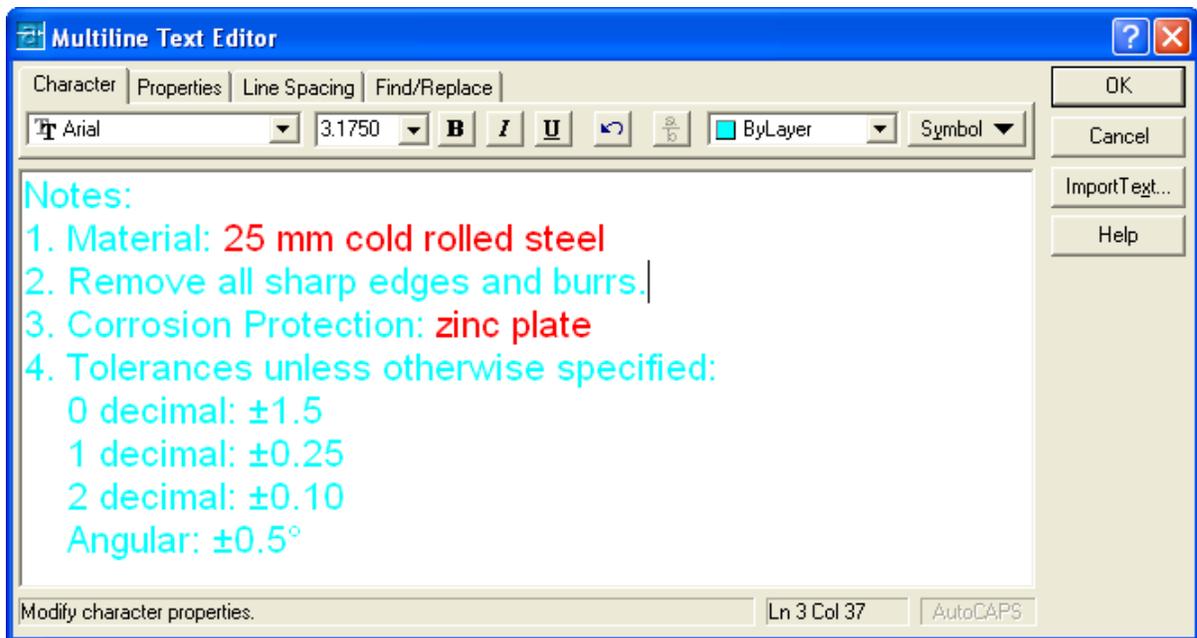
Figure 12.15 – Modifying the Notes by Adding Material and Corrosion Details

Now that you have your notes in the drawing, pick the rectangle that surrounds the Bracket problem and then pick the PSpace layer. Then freeze the layer so the rectangle does not show up. The rectangle is now not visible and the Bracket cannot be moved in Layout 1 (See Figure 12.16).

In later lessons, you will learn how to add attributes to the Titleblock, so each block will contain data after answering each question.

Figure 12.16 – The Finished Bracket Drawing

* World Class CAD Challenge * - Close this drawing file. Create a New file and draw the Bracket problem, dimension, and place the border and notes in less than 15 minutes. Continue this drill four times, each time completing the drawing under 15 minutes to maintain your World Class ranking.