

# Chapter 16

## Creating a 2D Drawing in Paper Space

---

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

- **Converting 3D Solids to 2D Orthographic Views**
- **Opening a Solid Part Drawing**
- **Setting the Page Size in Paper Space**
- **Inserting the B Sized Border with Insert Block**
- **Creating a Paper Space Layer**
- **Making Four Windows into Model Space using MView**
- **Inserting an Isometric View**
- **Dimensioning the Drawing**
- **Completing the Title Block**
- **Inserting a Standard Notes into the Paper Space Layout**
- **Modifying Standard Notes in Paper Space**

## Converting 3D Solids to 2D Orthographic Views

---

In the bracket problem of the Fundamentals of 2D drawing textbook, you learned how to draw an orthographic representation of a 3D part using front and top views. Then you learned in the Language of Drawing textbook how to construct all six views, front, top, right, left, bottom and back. This process can take plenty of time when considering the number of lines, arcs and circles that we need to draw. One of the biggest advantages in drawing a part in three dimensions is that we can construct 2D orthographic views in a matter of seconds from the 3D solid model using the MView command.

In this chapter, we will learn the procedure for completing a multiview drawing using the MView command, our dimensioning skills, a predefined drawing border, and notes. We will use our sketch from chapter two that describes the dimensions for the project as a guideline to dimension the views. We will utilize the previously created borders, mechanical and metric notes files stored in our 2D Drawing folder to accomplish those tasks quickly. This chapter will teach you all of the techniques you need to know to turn your previously completed and pre-checked work into a drawing made efficiently and without error.

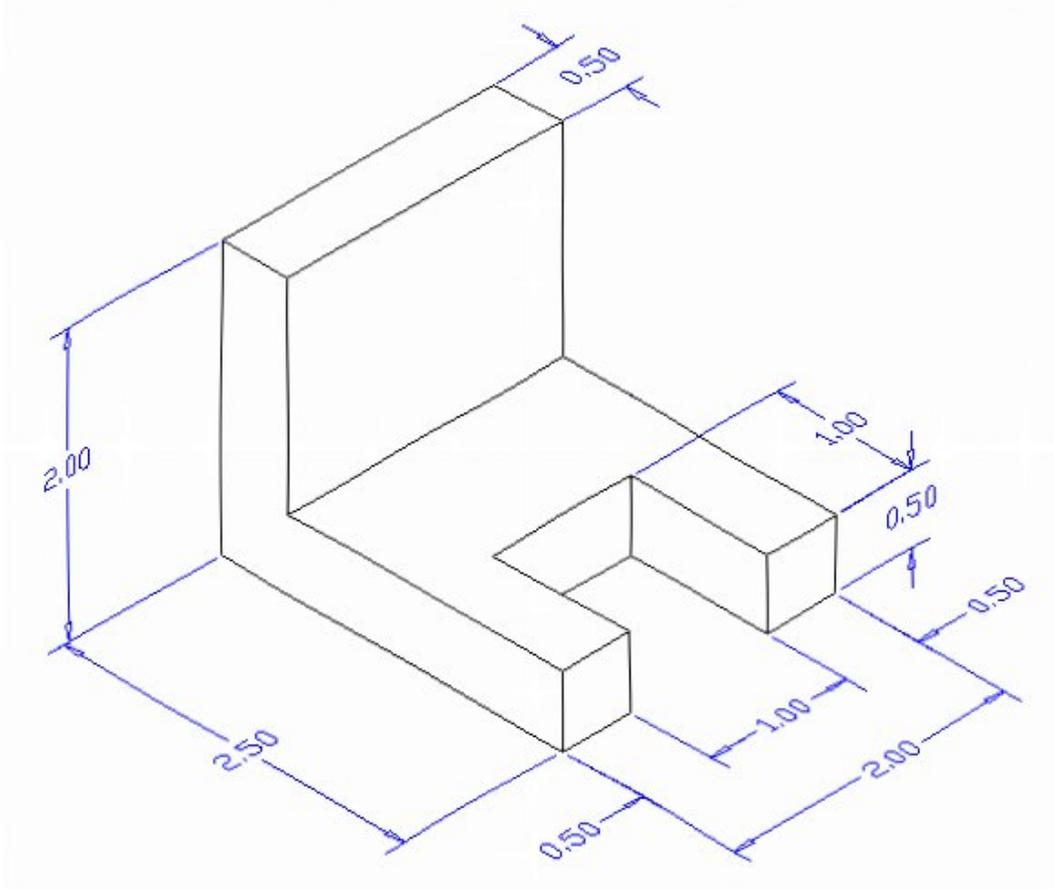


Figure 16.1 – Problem One Sketch

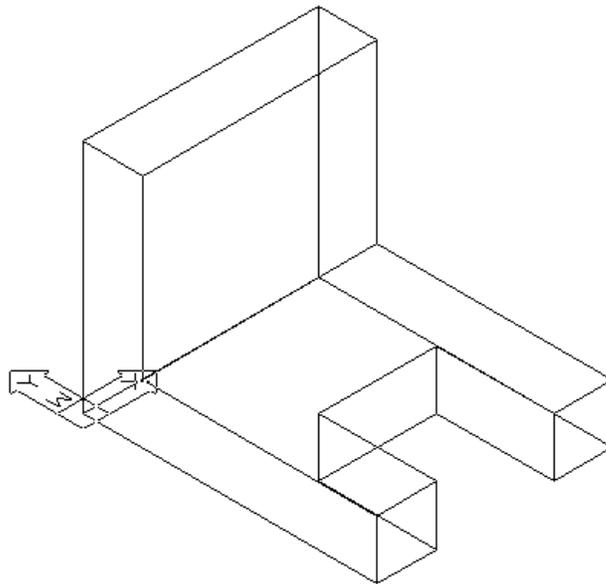
Remember, we will still need the sketch from problem 1 to aid us in dimensioning the part later. The dimensioning task will require the most work in this chapter. Using the sketch to assist in placing the dimensions on the part will remove valuable time from the process and improve your World-Class time. There are nine measurements on the sketch, so expect to see nine on the finished drawing. On a drawing of this shape, we will suppose that we need three views: front, top, and right or left side.

+

## Opening the Solids Part Drawing

---

Open Solids Problem 1 in progeCAD Professional and view the drawing. Type Hide at the command line to see the drawing as shown in Figure 16.2. We can use the Hide command to help us see the solid because the Hide function removes the rear tessellation lines.



**Figure 16.2 – Opening Solids Problem 1**

When you open the drawing, you will see that the solid resides in Model Space. The majority of the work we are about to do will be in Paper Space. We are now ready to leave Model Space and use the MView command to generate the orthographic and isometric views in seconds.

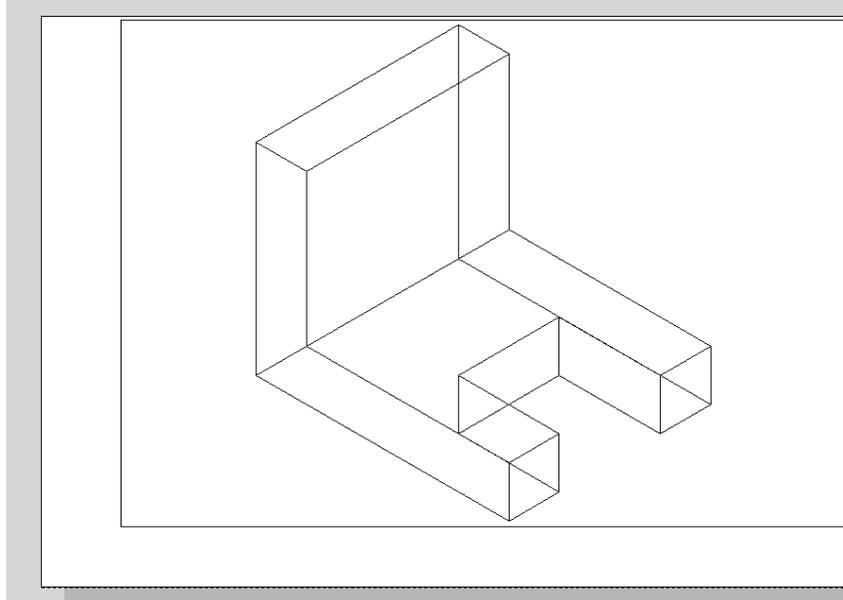
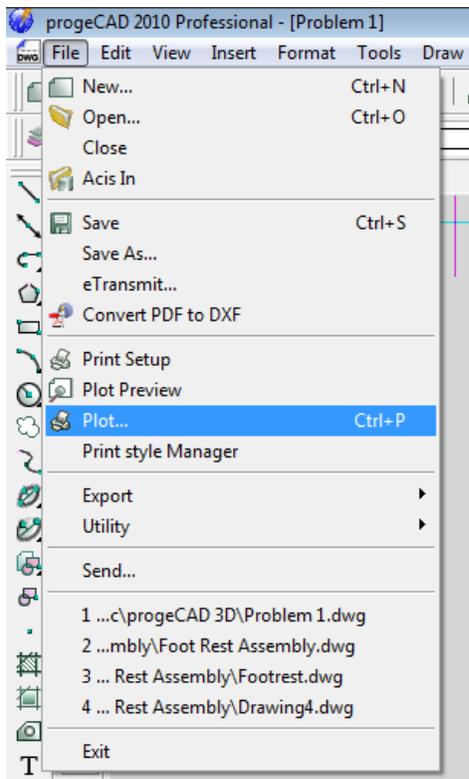


Figure 16.3 – Problem 1 in Layout1 (Paper Space)

## Setting the Page Size in Paper Space



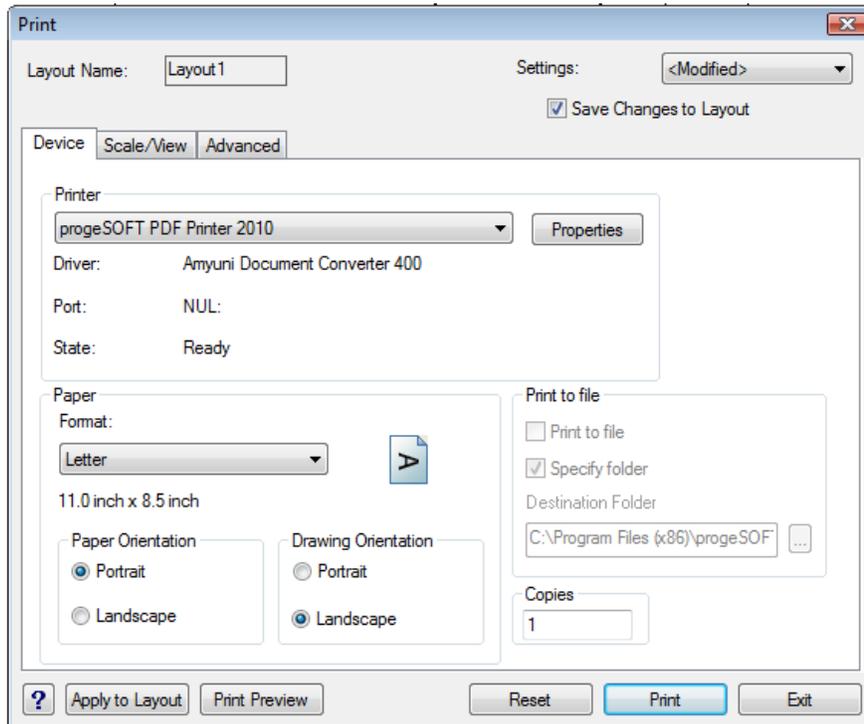
Select File and then Plot as shown in Figure 16.4. The first Tab in the Print window deals with the printing device. You may only have a small printer and a stock of 8.5 x 11 or even longer legal sized paper. By using your own printer, you can run smaller test plots using the A size paper for the A and C drawing borders and the legal cut paper for the wider B and D size borders. After checking printouts at home, designers can send their drawings to any number of companies that provide CAD printing services at very economical prices for the larger layouts that are needed by the customer. For those of us who are at an organization with printers that have the abilities to print any A, B, C or D size drawings, you can select that printer in the Printer section of the Plot window. Always consult with your department manager about which printers are accessible for your educational use. Many professionals group have black and white machines for check prints and color devices for finished drawings.

Figure 16.4 – Plot

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 to save resources or time.

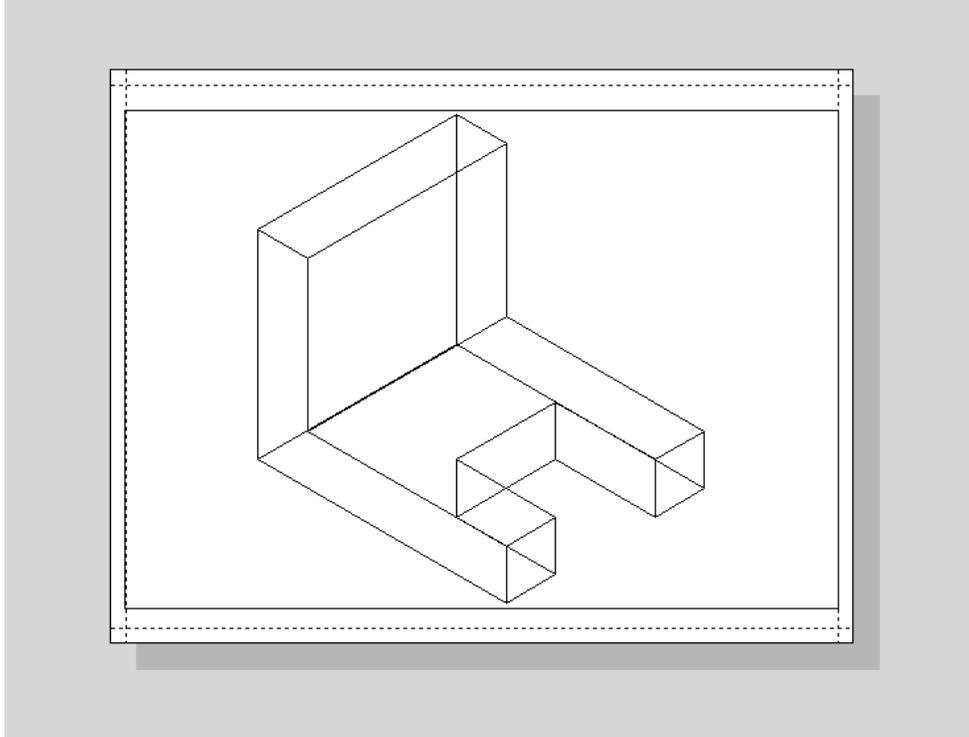
Choose your printer in the Name window. Next, since you will be printing the drawing on A size (8.5 x 11) paper, choose the correct paper size in the Paper Size window list. Every document size our printer supports is listed in the list box.

Continuing with customizing your Plot Options, select Portrait for Paper Orientation and Landscape for Drawing Orientation to rotate the image within the wider paper size of the A-size border. Again, for this step, we must know our plotting or printing equipment or use the Print Preview to check the drawing paper's direction.



**Figure 16.5 – Print Window**

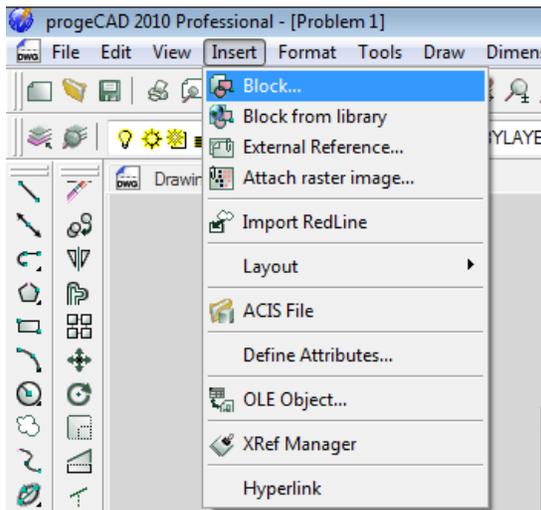
The default settings should be fine for our purposes in this chapter. After making all of these changes, select the Apply to Layout, and then hit Exit. The B-size drawing sheet is shown.



**Figure 16.6 – B Size Sheet in Layout View**

Before inserting the B sized border, pick the Erase tool on the Modify toolbar and select the rectangle box around the model and hit ENTER. The window displaying Model Space will disappear.

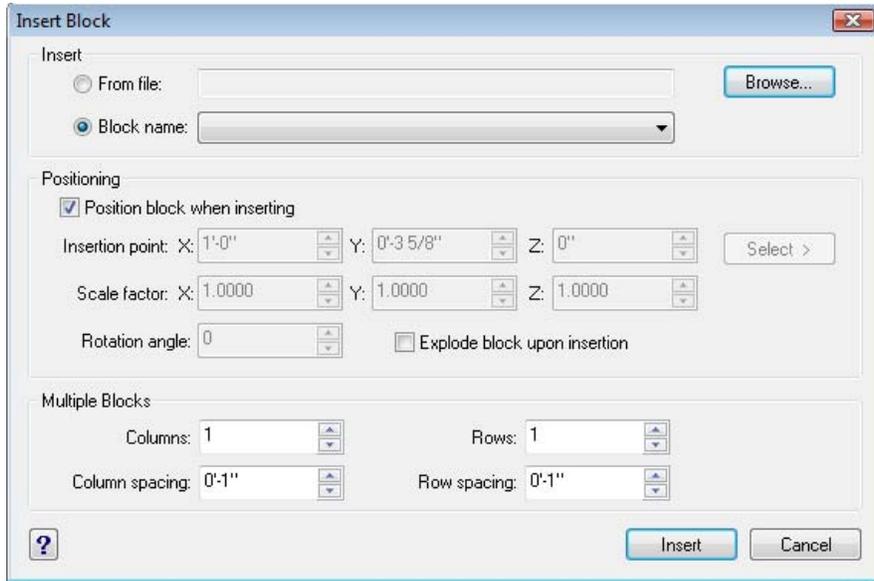
## Insert the B Sized Border with Insert Block



**Figure 16.7 – Insert Block**

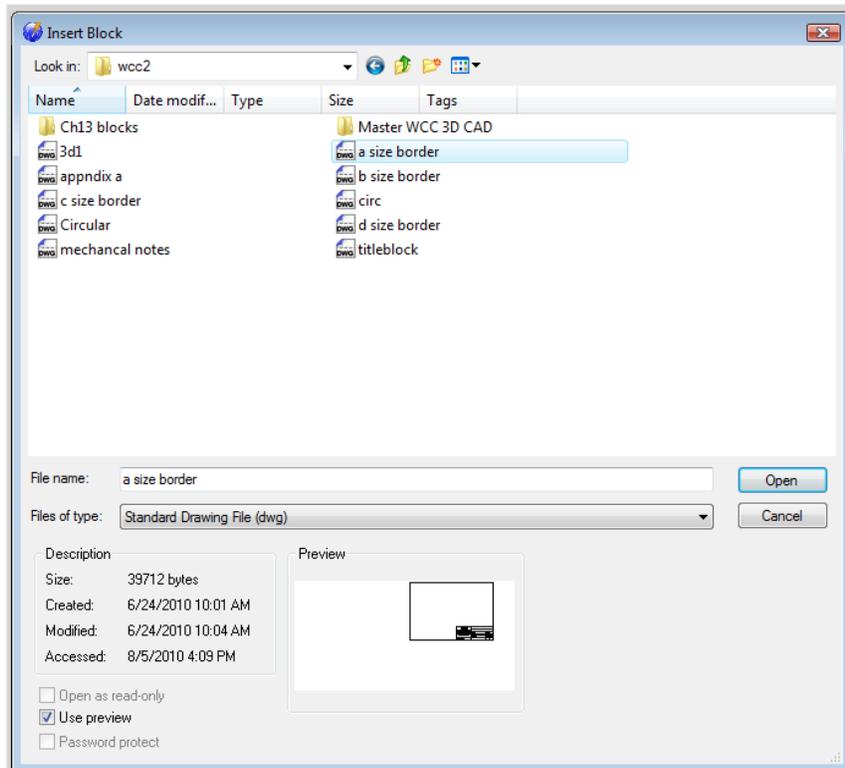
To insert the block containing the A-Size Border into Layout 1, select the Insert Block tool on the Draw toolbar or select Insert, then Block on the Menu as shown in Figure 16.7. The Insert Block window (Figure 16.8) will appear in the graphical display. Use the Browse button to find the “b size mechanical” border in your 2D Drawing folder (Figure 16.9). Keep the Position Block When Inserting checkbox checked (Figure 16.10).





**Figure 16.8 – Insert Block Window**

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 (Figure 16.11). We can grab and move the drawing frame to reposition it inside the border. Move the frame to the top left corner of the border as shown in Figure 16.13.



**Figure 16.9 – Select Drawing File Window**

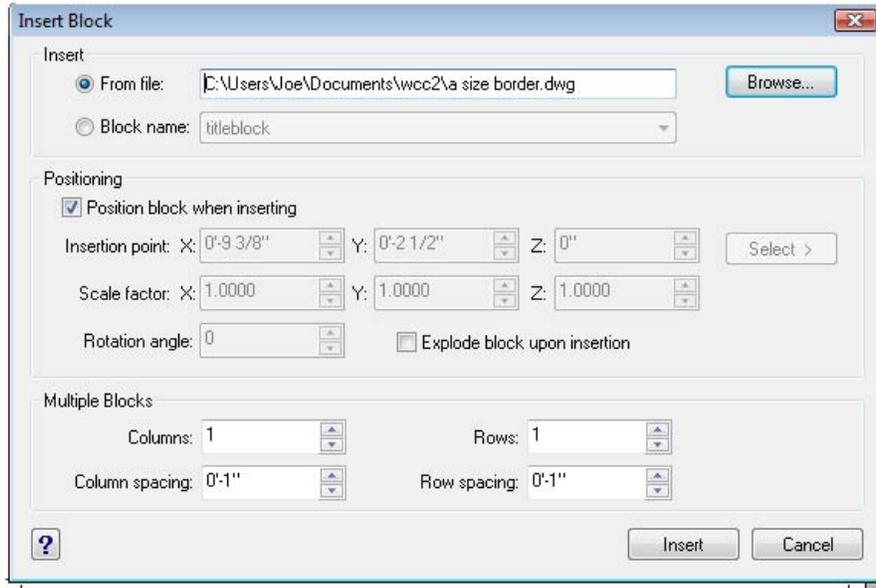


Figure 16.10 – B Size Mechanical

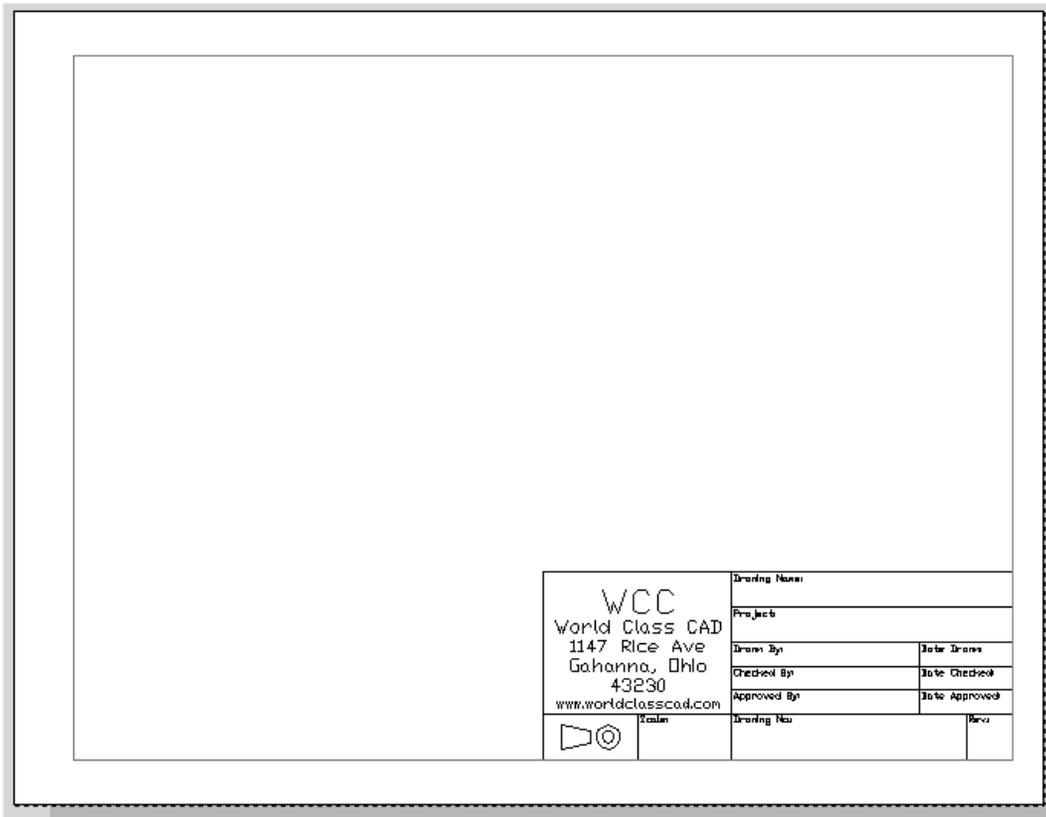


Figure 16.11 – B Size Mechanical in Layout1

## Creating a Paper Space Layer

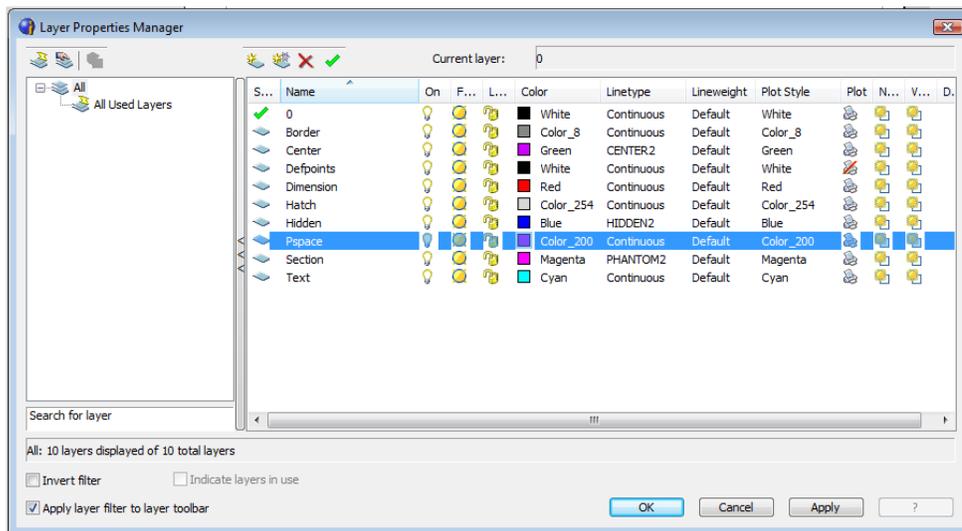


Figure 16.12 – Layer Properties Manager

Before we insert the three orthographic and one isometric view, we need to make layers so we can properly identify and control the different aspects of this design. Select the Explore Layers tool on the Entity Properties toolbar and the Layer Property Manager window will open on the graphical screen. Add the list of layers as shown in Figure 16.12, changing the Color and Linetype settings as shown. A new layer you will notice is PSpace, which we will eventually use to freeze our views so that no one can see the windows designating Model Space. The MView command creates one or more windows in Paper Space that allows the viewer to see a solid in Model Space. Besides seeing the model, we also see the outline of the viewing window. Our strategy is to create the window on a layer that we can freeze so the window outline will be invisible.

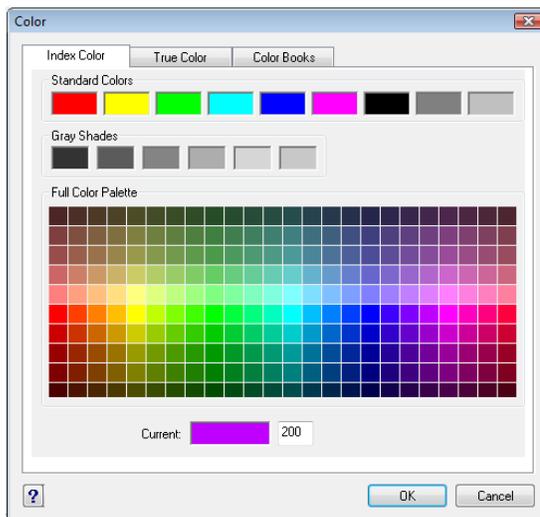


Figure 16.13 –Color Window for the Paper Space Layer

Hit Apply and OK to exit the Layer Properties Manager. Move the border and title block to the Border layer using the Layer Selection drop menu on the Entity Properties toolbar. Then select the Pspace as the current layer so that the MView windows will be placed on that layer. Now we will insert the three orthographic views we need for the drawing.

## Making Four Windows into Model Space using MView

At the command line type `MView` and the prompt “*Viewports: ON/OFF/Fit/Lock/Object/2/3/4/<First corner>:*” appears on the command line. You want to have three orthographic views and one isometric view, so the type **4** and hit **ENTER**. Next, we will draw a window on the graphical screen as shown in Figure 16.14. Once the box is drawn, four viewports displaying Model Space will appear with the part oriented in the southwest isometric view. The next step is to position each view into an orthographic position.

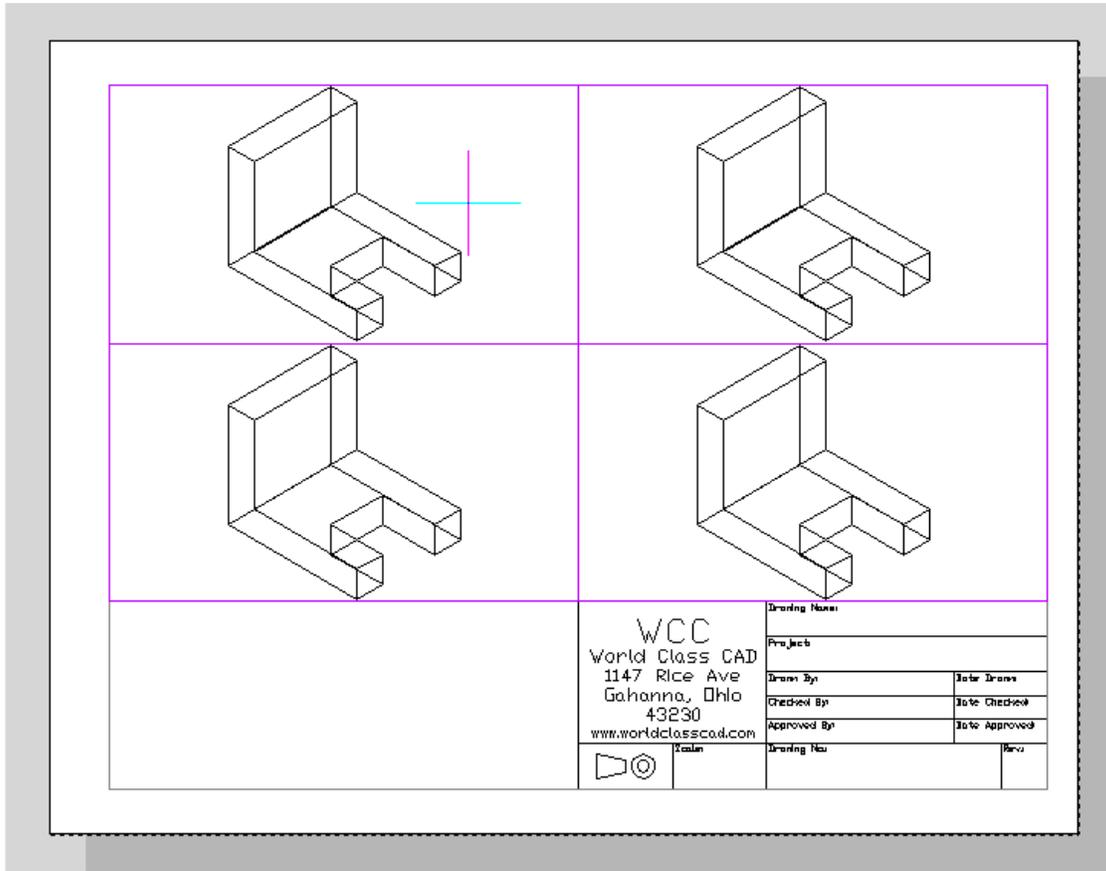


Figure 16.14 – The Viewports placed

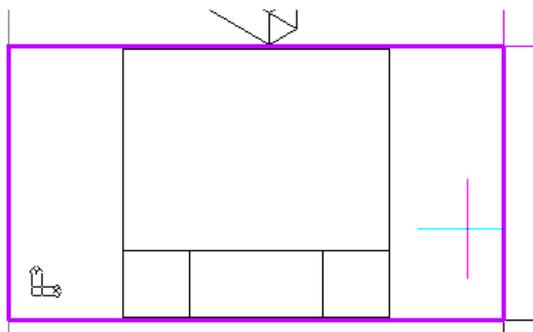


Figure 16.15 – Setting the Front View

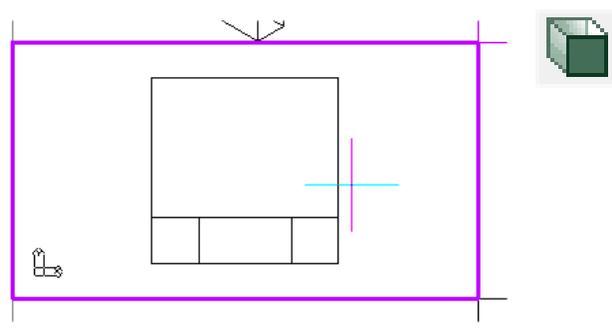
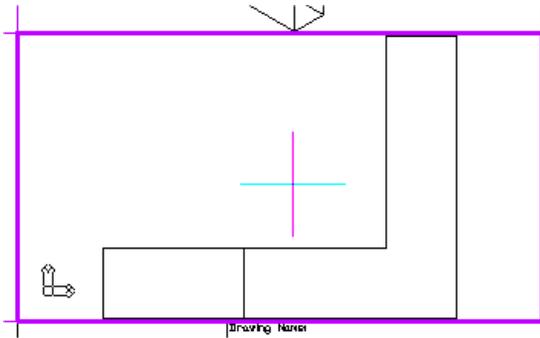


Figure 16.16 – Zooming 1x Paper Space

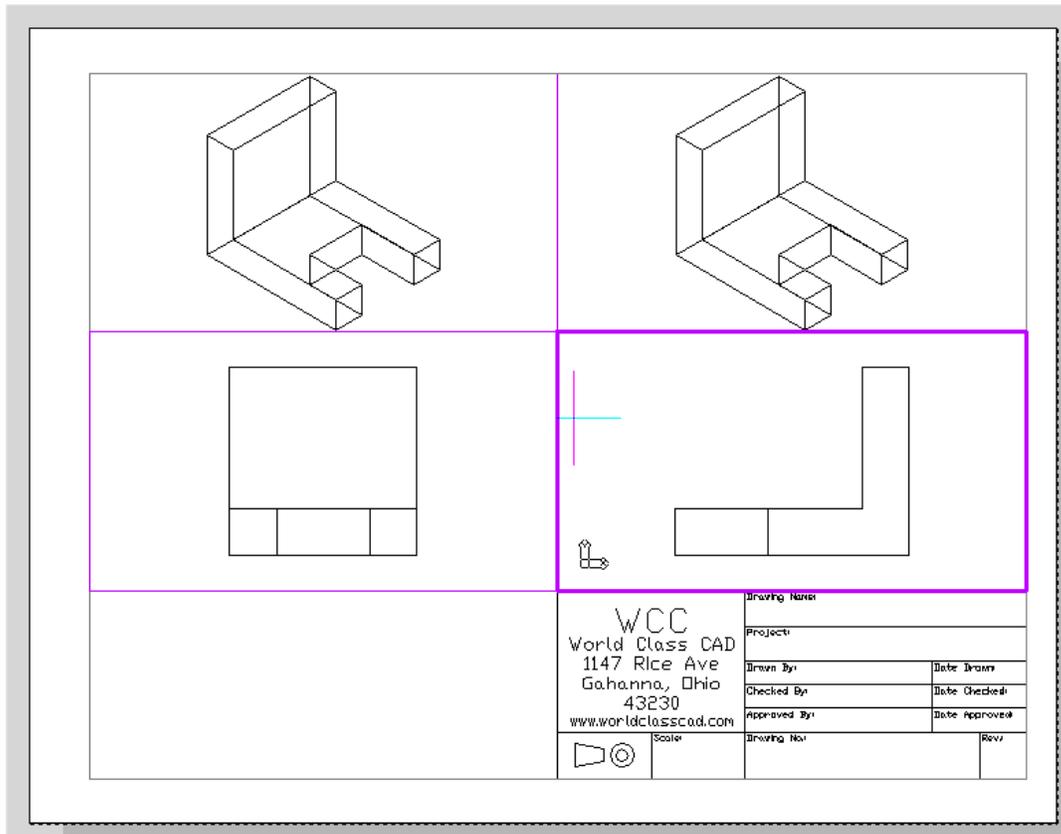
Double click in the lower left hand window and you will see the UCS icon appear. You are going to place the front orthographic view in this viewport. On the View toolbar, select the Front view as shown in Figure 16.15. To bring this view into one to one scale, we will type **Z** and **ENTER**, and then **1xp** and **ENTER**. By typing 1xp, the Model Space view is adjusted so we can dimension the part accurately in Paper Space. The finished from view is shown in Figure 16.16.



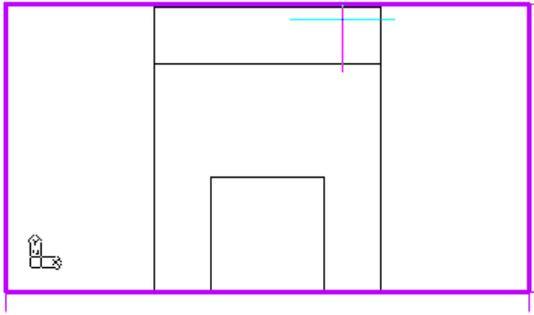
Double click in the lower right hand window and you will see the UCS icon appear. You are going to place the Right orthographic view in this viewport. On the View toolbar, select the right view as shown in Figure 16.17. To bring this view into 1:1 scale type **Z** and **ENTER** and then **1xp** and **ENTER**. Again, by typing 1xp, the Model Space view is adjusted so we can dimension the part accurately in Paper Space (Figure 16.18).



**Figure 16.17– Setting the Right View**



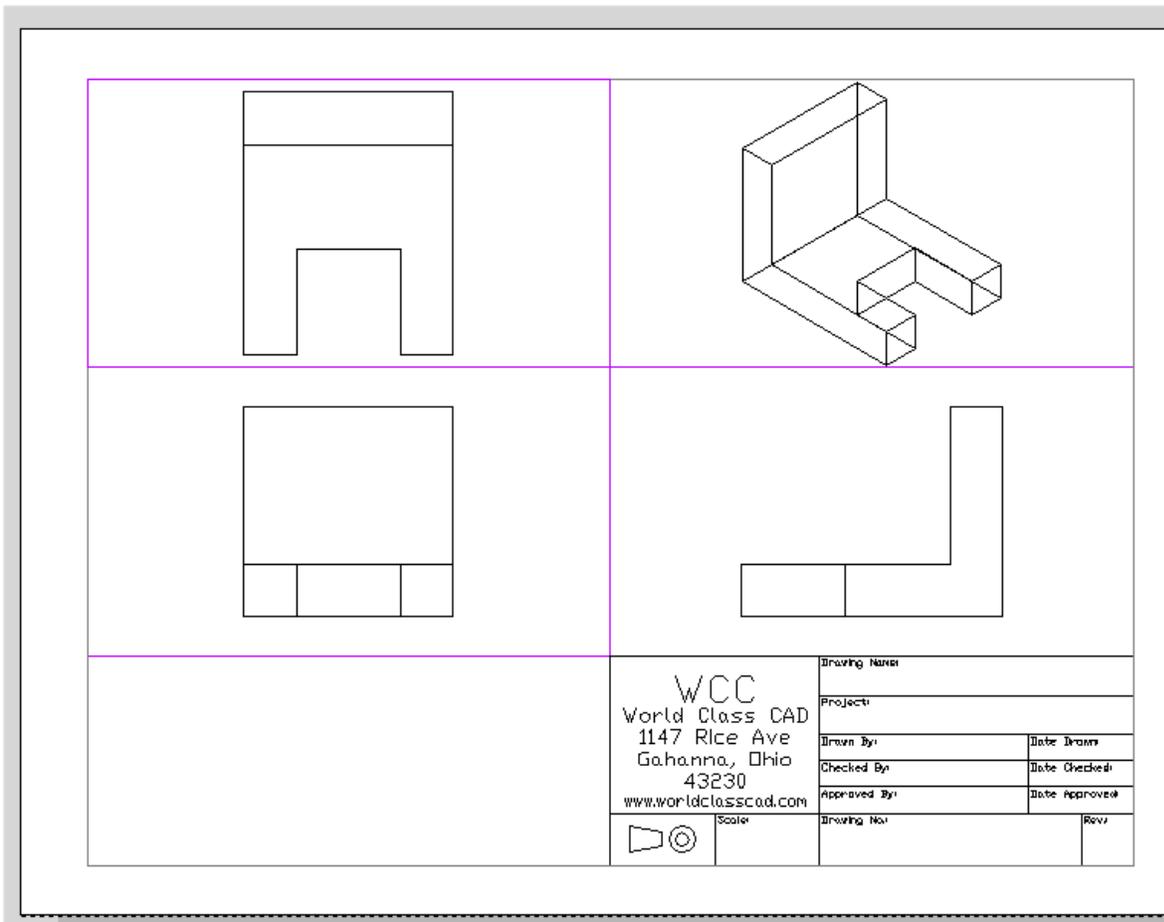
**Figure 16.18 – Scaling the Right Orthographic View**



Double click in the upper left hand window and again you will see the UCS icon appear. To place the top orthographic view select the Top view from the View toolbar as shown in Figure 16.19. Bring this view scale by executing the 1xp Zoom command. In Figure 16.20, you can see all three orthographic views in proper position and properly scaled.



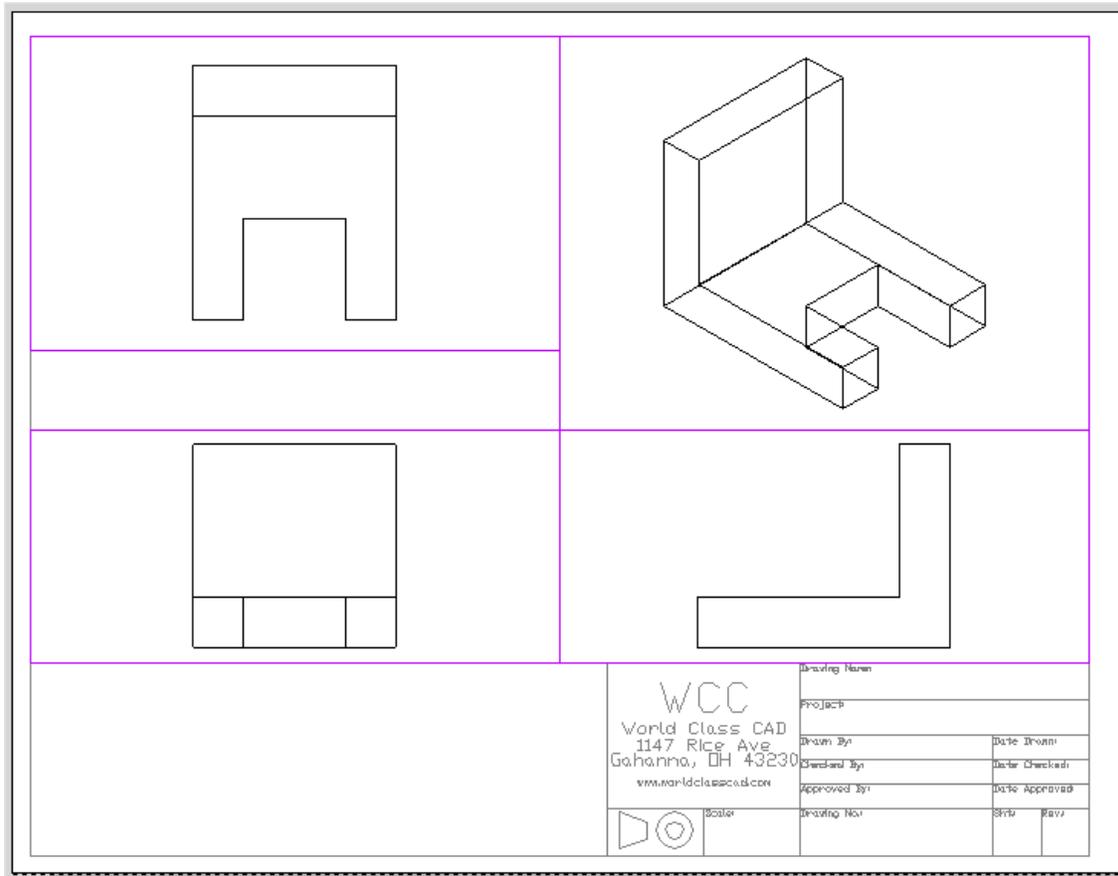
**Figure 16.19 – Move the Top View**



**Figure 16.20 – Three Orthographic Views Finished**

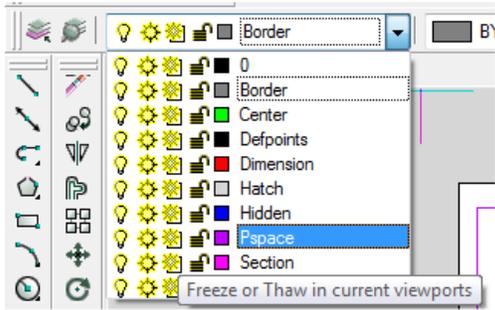
## Insert an Isometric View

An isometric view is helpful for a customer, machinist or an assembly worker to visualize the part when they have trouble seeing the features in two dimensions in the front, right and top views. In the 20<sup>th</sup> century, isometric drawings could be costly to make, but with 3D solid models and MView, this view is amazingly easy to generate. We will place the southwest isometric view in the upper right corner of the drawing.



**Figure 16.21 – Inserting the Southwest Isometric View**

Double click in the upper right hand window and we will see the UCS icon appear. On the View toolbar, select the SW Isometric view as shown in Figure 16.21. Bring this view into 1:1 scale by executing the 1xp Zoom command. Finally, make sure all of the views are displayed in the Hidden visual style by using the Hide command on the Shade toolbar. If you need to move or resize any of your viewports to best display all of the drawings in your Paperspace, do it now.

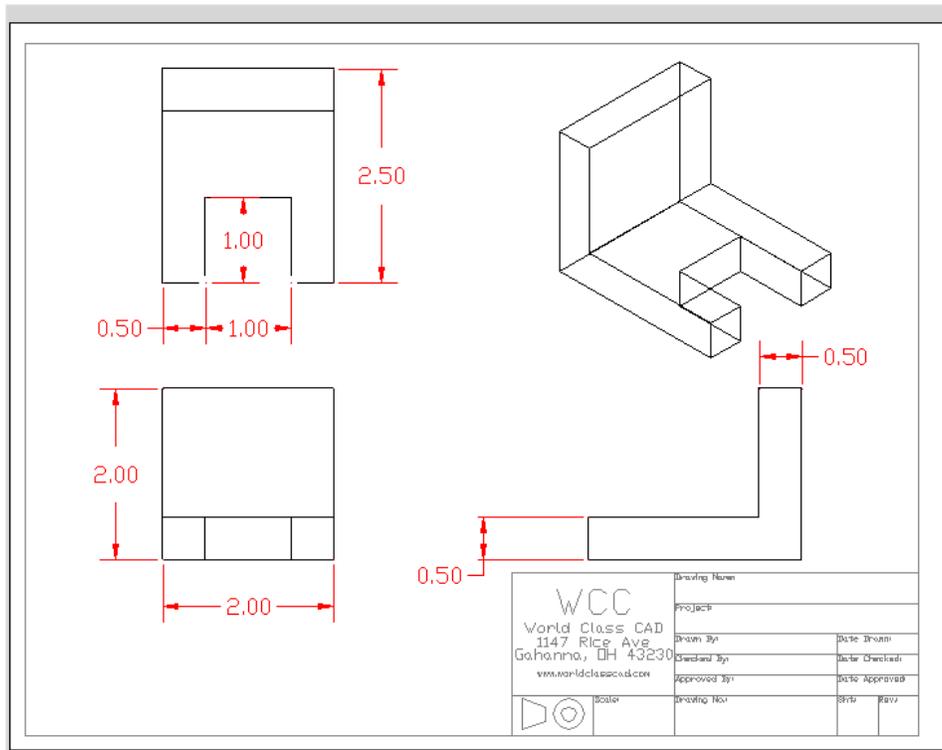


**Figure 16.22 – Freezing the PSpace Layer**

Next, make sure you are out of all of the viewports by checking to see if there is a UCS on your display. If not, select PSpace from the Layer Selection drop menu and freeze the layer as shown in Figure 16.22. The four MView viewports will disappear.

## Dimension the Drawing

When dimensioning a multiple view drawing, the goal is to portray a clear and concise visual representation of the part or assembly. A great rule of thumb is to place the dimension in the view from which a machining or construction practice will take place. For example, if a worker is cutting a 1 inch notch from above the top plate of the Bracket, draw the dimensions in the same view that the work is being done. Likewise, in a forged or cast part, add the radius or fillets in the proper view so the pattern maker can easily read the drawing and make the correct choices. Check every circle, line and arc at the end of the dimensioning process to guarantee that there are no missing measurements.



**Figure 16.23 – Dimensioning the Solids Problem**

In Figure 16.23, you can see the dimensions in position that would satisfy a project manager.

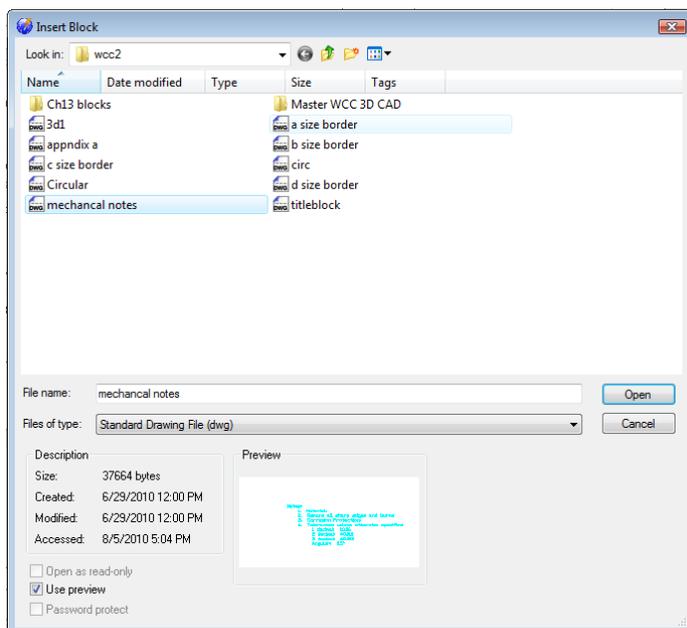
## Complete the Title Block

Zoom close to the drawing's titleblock and give the drawing the name Bracket. The project name is 3D Drawing with progeCAD. Place your name in the Drawn By box with the date the drawing is initially completed. Have another individual check your drawing. Place their name and date in the titleblock. The scale of the drawing is 1=1, the drawing number can be 1000, the sheet number is 1, and the revision is the letter A.

|   |                                      |                           |       |       |
|---|--------------------------------------|---------------------------|-------|-------|
| WCC<br>World Class CAD<br>1147 Rice Ave<br>Gahanna, OH 43230<br>www.worldclasscad.com | Drawing Name:<br>Bracket             |                           |       |       |
|   | Project:<br>3D Drawing with progeCAD |                           |       |       |
|   | Drawn By:<br>Charles Robbins         | Date Drawn:<br>08/05/10   |       |       |
|   | Checked By:<br>Mark Bughman          | Date Checked:<br>08/05/10 |       |       |
| Approved By:<br>Bracket   |                                      | Date Approved:            |       |       |
|      | Scale:<br>1=1                        | Drawing No.:              | Sht.: | Rev.: |
|   |                                      | 1001                      | 1     | A     |

Figure 16.24 – Filling Out the Title Block

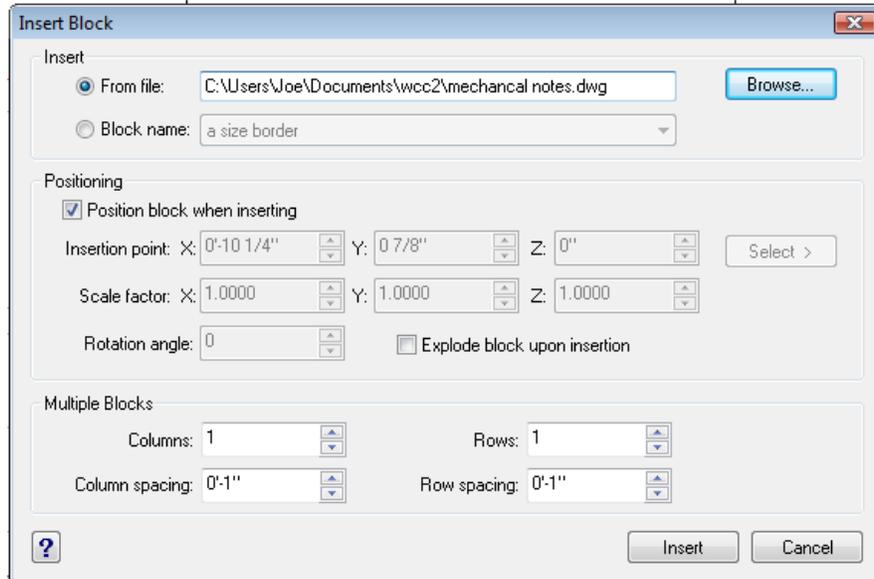
## Inserting a Standard Notes into the Paper Space Layout



To insert the block containing standard notes into Layout 1, select the Insert Block tool on the Draw toolbar and the Insert Block window will appear in the graphical display as shown in Figure 16.26. Use the Browse button to find the “mechanical notes” in your 2D Drawing folder. Hit the Open button to continue.



Figure 16.25 – Inserting the Metric Mechanical Notes



**Figure 16.26 – Check Explode in the Insert Window**

Now, pick the OK button and pick a point to below the views to place the notes. Next, we will modify our standard mechanical notes using the Edit Text tool.

## Modifying Standard Notes in Paper Space

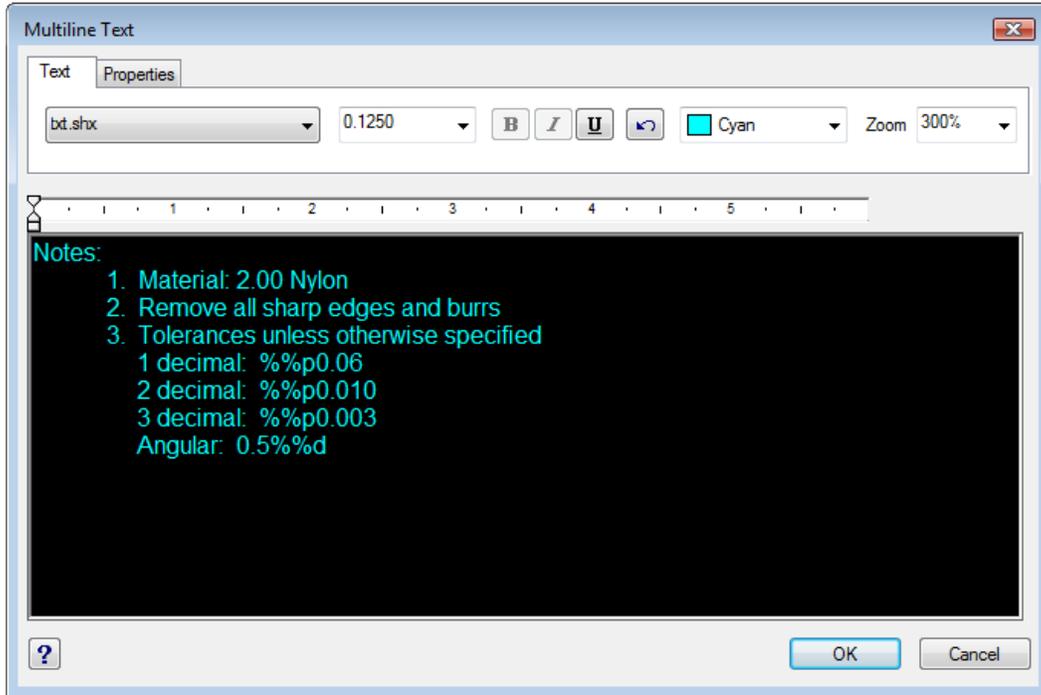
---

To edit the standard notes, double-click on the notes. When asked to activate local editing, click OK. Select the Edit Text tool on the Text toolbar, select the notes, and then hit ENTER. The Multiline Text window will appear in the graphical screen. Just like you would use a word processing program, make the following changes to notes one and three, while leaving notes two and four alone.

1. Material: 2.00 Nylon

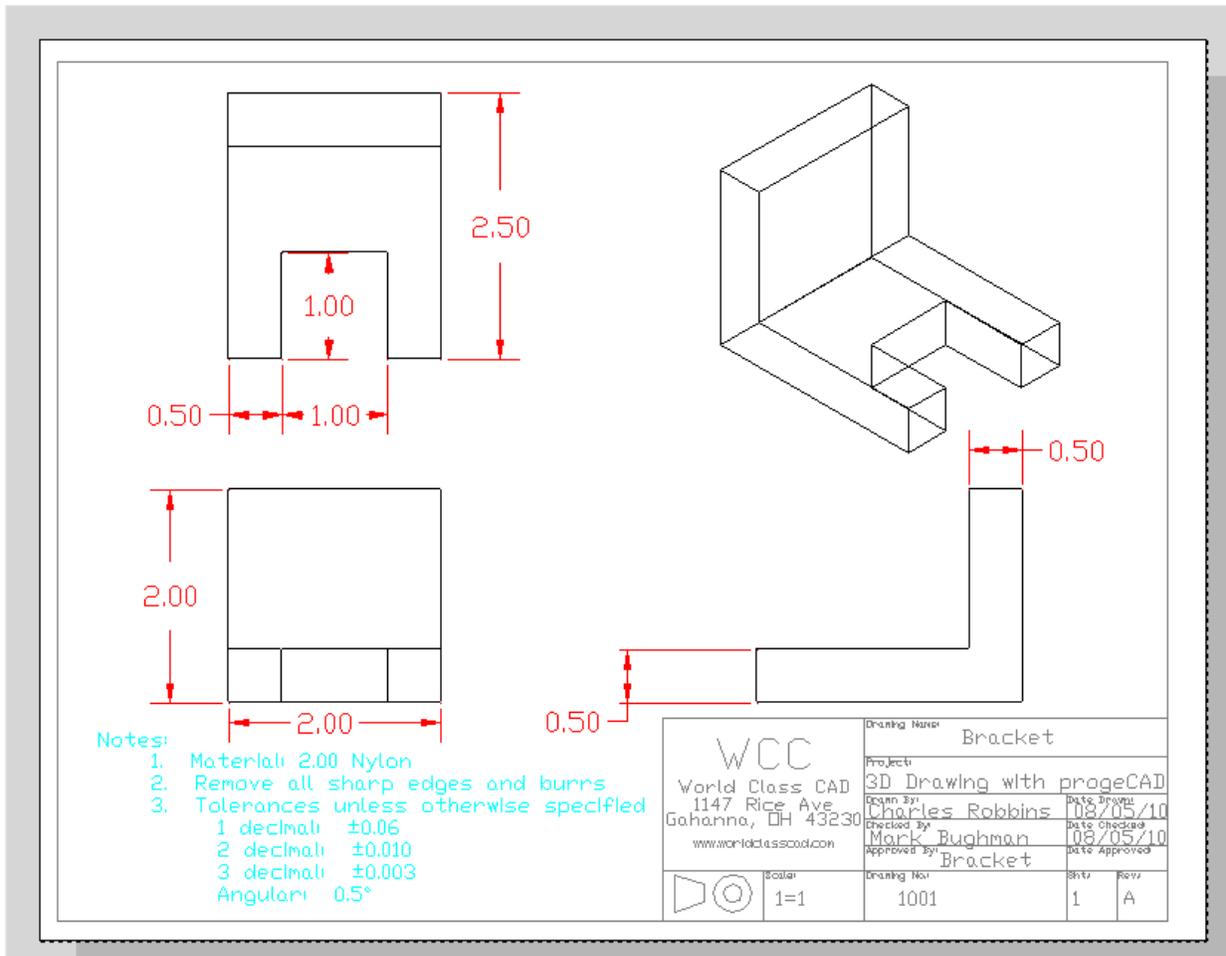
And remove the corrosion protection note

Select “OK” to close the window (Figure 16.27).



**Figure 16.27 – Modifying the Notes by Changing Material and Corrosion Details**

If you are proficient in the use of the tools provided by Computer Aided Design software, you can create complex orthographic views and drawings as shown in Figure 16.32 for production release while retaining the solid available for the CAD assembly. In this chapter, we created a drawing independent of the 3D model.



**Figure 16.28 – The Finished Solids Problem 1 Drawing**

**\* World Class CAD Challenge 101-14 \* - Open Problems 1-10 and Revolved Solids 1-3 and create 2D drawings for each solid. Open the file, use MView, insert the border, dimension, and add notes in less than 30 minutes.**

**\* World Class CAD Challenge \* - Report your best times to World Class CAD at [www.worldclasscad.com](http://www.worldclasscad.com) to obtain your world class ranking.**