Chapter 16

Creating a 2D Drawing in Paper Space

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

1. **Converting 3D Solids to 2D Orthographic Views**
2. **Open the Solid Part Drawing**
3. **Setting the Page Size in Paper Space**
4. **Insert the B Sized Border with Insert Block**
5. **Creating a Paper Space Layer**
6. **Making Four Windows into Model Space using MView**
7. **Insert an Isometric View**
8. **Dimension the Drawing**
9. **Complete the Title Block**
10. **Inserting a Standard Notes into the Paper Space Layout**
11. **Modifying Standard Notes in Paper Space**

Converting 3D Solids to 2D Orthographic Views

In the bracket problem of the Fundamentals of 2D drawing textbook, we learn how to draw an orthographic representation of a 3D part using front and top views. Then, we learn 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 part 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. The simple tool repetitively uses the View toolbar to look into Model Space and the Zoom command to make 2D views in each direction. We will use our sketch from chapter two which 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 the Fundamentals of 2D Drawing folder to accomplish those tasks quickly. All of these various procedures that we learn in this chapter, where we take previously completed and pre-checked work to give us a drawing that is made efficiently and without error.

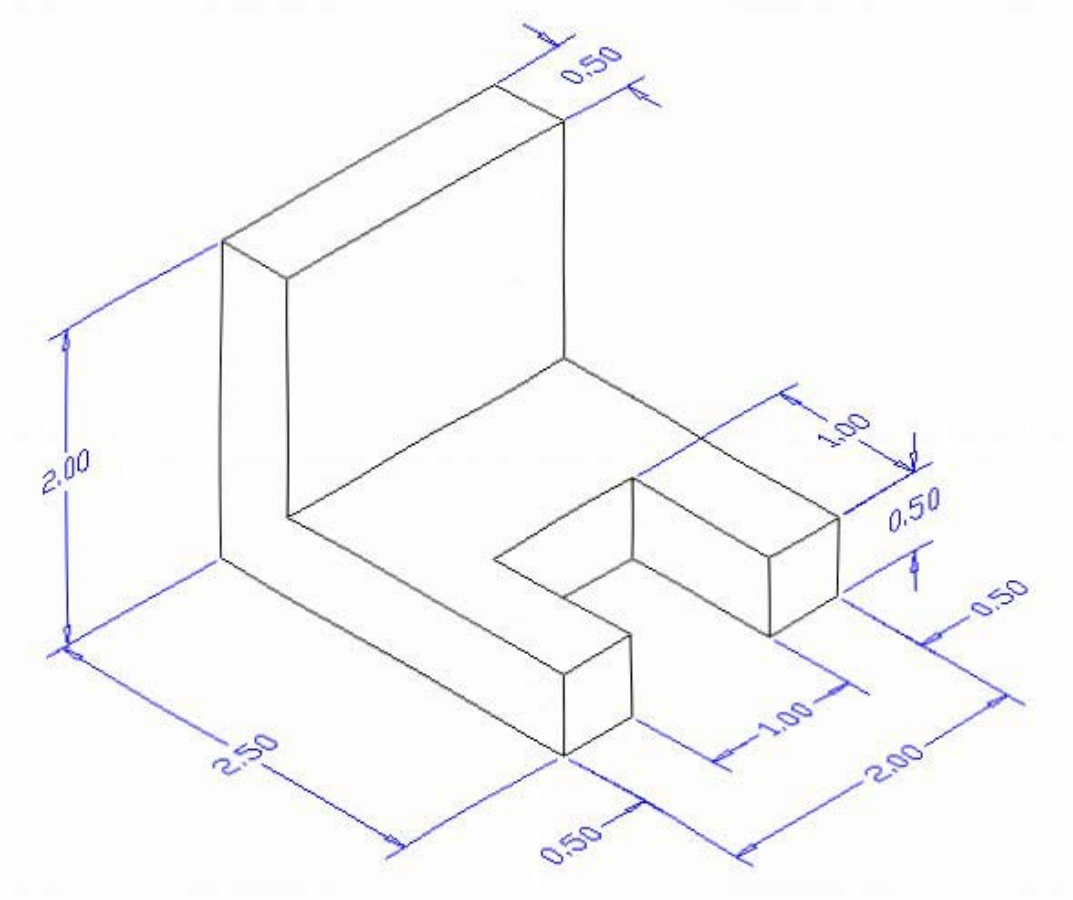


Figure 16.1 – Problem One Sketch

Remember, we will still need the sketch from problem 1, since we find that having this information available to assist in dimensioning is important to quickly finishing the 2D drawing. The dimensioning task will require the most work in this chapter and using the sketch to assist in placing the dimensions on the part can remove valuable time from the process and improve our world-class time. There are nine measurements on the sketch, so we expect to see nine on the finished drawing. On a drawing of this shape, we also suppose that we need three views, front, top and right or left side.

Open the Solids Part Drawing

Open the Solids Problem 1 in AutoCAD and view the drawing. We can 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, since the Hide function removes the rear Tessellation lines and now the part is easy to read.

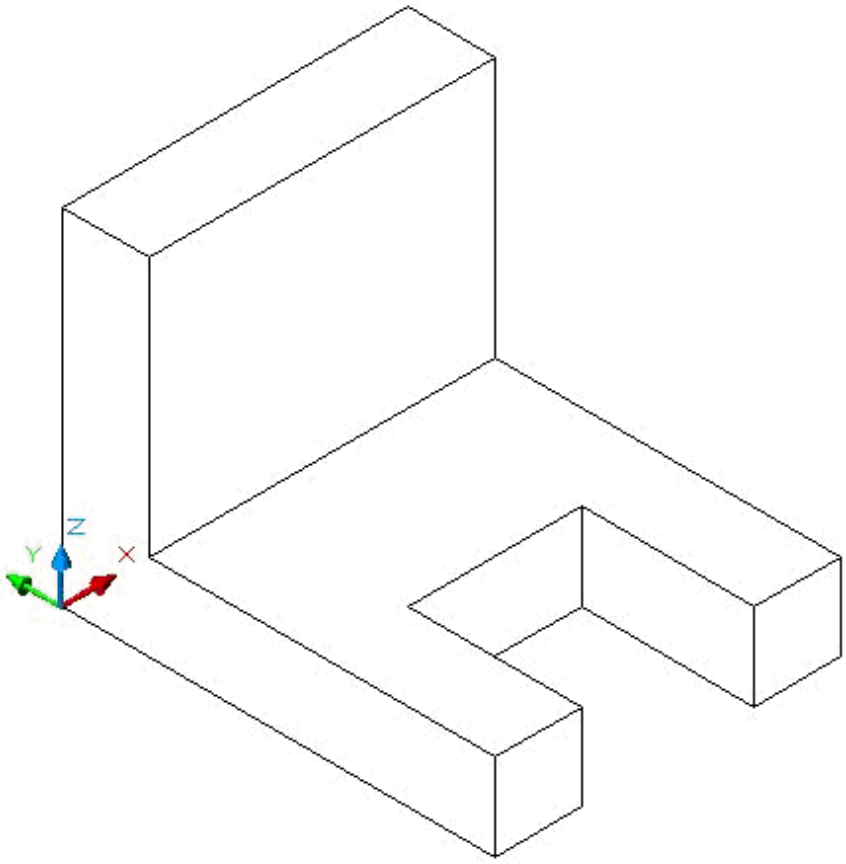


Figure 16.2 – Open Solids Problem 1

When we open the drawing, we see 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.

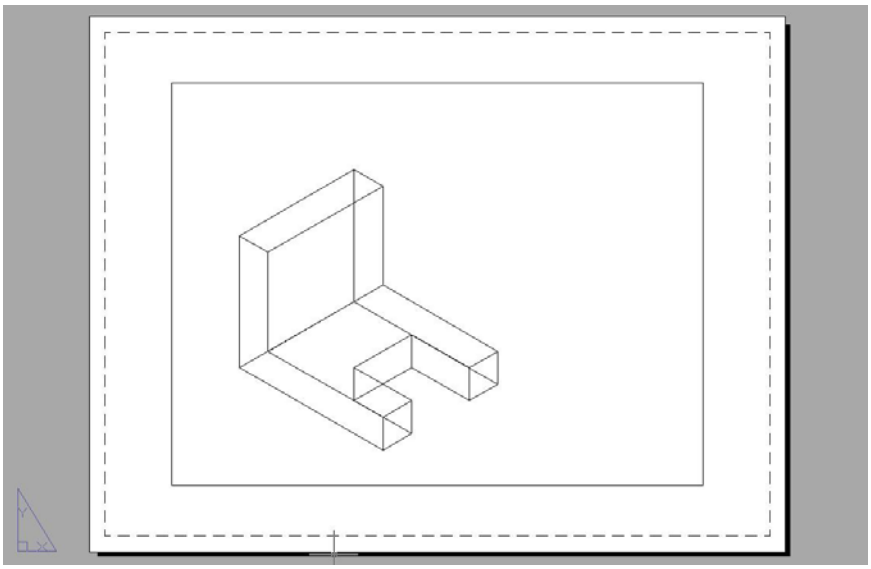


Figure 16.3 – Problem 1 in Layout1 (Paper Space)

Setting the Page Size in Paper Space

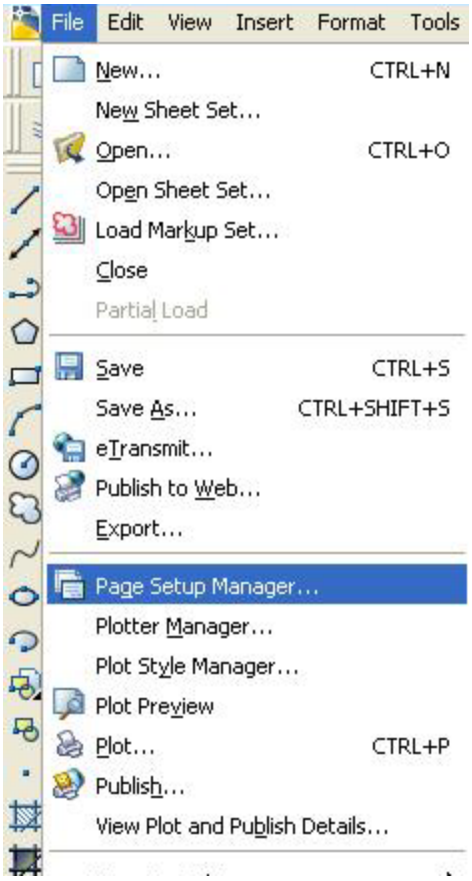
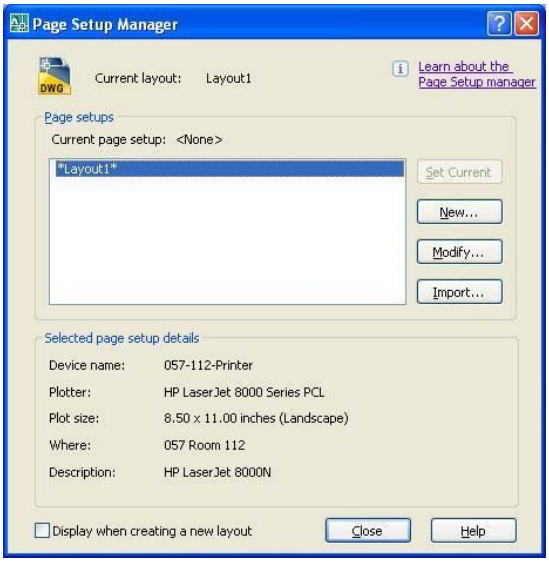


Figure 16.4 – File Manager

Select File and then Page Setup Manager as shown in Figure 16.4. Select Layout 1 and click Modify as shown in Figure 16.13 to reach the Page Setup – Layout 1 window. The first section in the Layout window is to name the plot device. At home, we probably just have a letter sized printer and a stock of 8.5 x 11 or even longer legal sized paper. By picking our printer in that case, we 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 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, then we can have that printer showing up in the Name window. Always consult with the department manager to which printers are accessible for our 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 we pay attention to any orientation to special tools in our list of offered devices, and keep any special instructions posted in our work space, so not to waste resources or time.

Now, we can choose our printer in the Name window and for the present we will ignore the Plot Style Table for pen assignments. Next, since we will be printing the drawing on A size (8.5 x 11), legal or B size (11 x 17) paper, we choose our paper size in the Paper Size window list. Every document size our printer supports is listed in the list box. We are going to select the ANSI B (11 x 17 inches) paper size.

Figure 16.5 – Page Setup Manager

Continuing with the queries into output options, in the Page Setup window, select the **Landscape** option button to rotate the image within the wider paper size of the B-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.

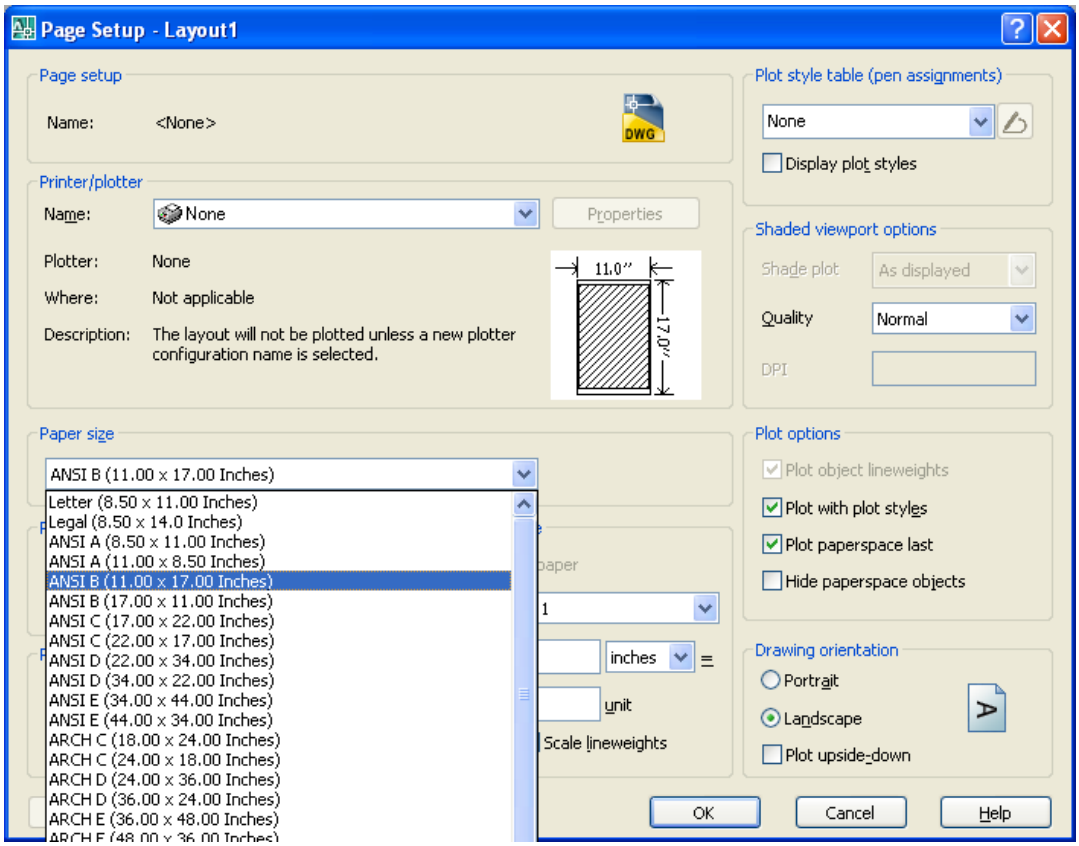
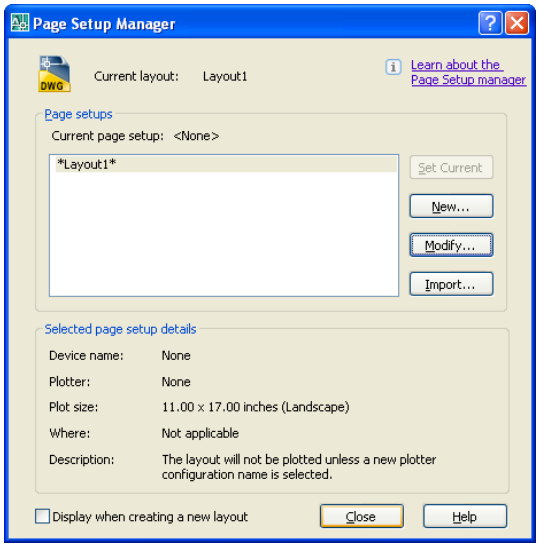


Figure 16.6 – Page Setup – Layout 1



The Plot Area can remain at Layout. On the Plot Scale frame of the Page Setup window the scale for our drawing is 1 to 1. Sometimes we will change the scale of all the drawing to fit the paper. After making all of these changes, we will select the OK command button on the Page Setup – Layout 1 window. In the Page Setup Manager, pick the Close button and we will return to the Layout 1 graphical display as shown in Figure 16.7. The B-size drawing sheet is shown.

Figure 16.7 – Completed Page Setup Manager

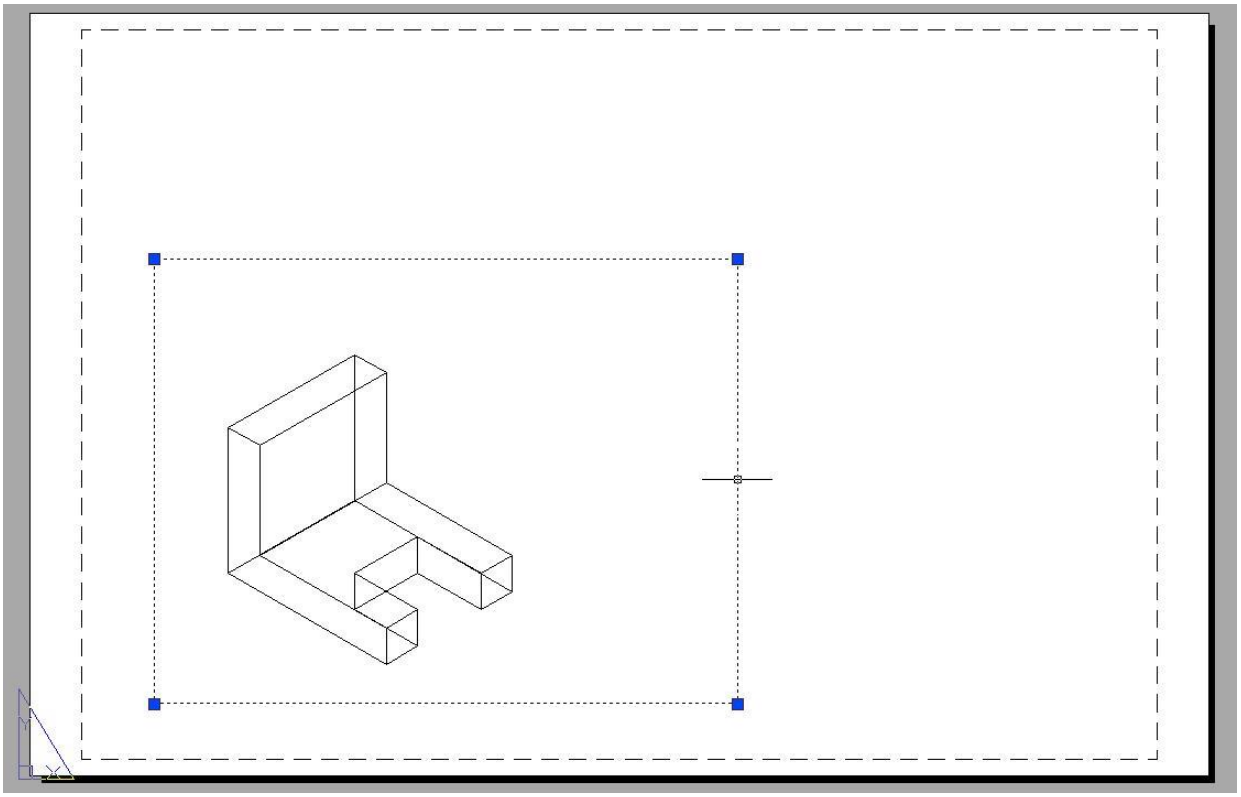
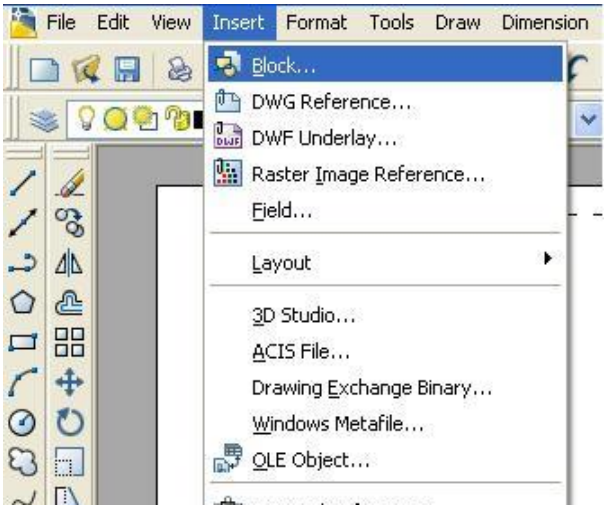


Figure 16.8 – B Size Sheet in Layout View

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

Insert the B Sized Border with Insert Block



To insert the block containing the B-Size Border into Layout 1, select the Insert tool on the Draw toolbar or select Insert, then Block on the Menu as shown in Figure 16.17. The Insert window (Figure 16.18) will appear in the graphical display. Use the Browse button to find the "b size mechanical" border in our Fundamentals of 2D Drawing folder (Figure 16.19). Keep the Specify On-screen checkbox unchecked and indicate that the Uniform Scale by checking the box (Figure 16.20).

Figure 16.9 – Insert Block

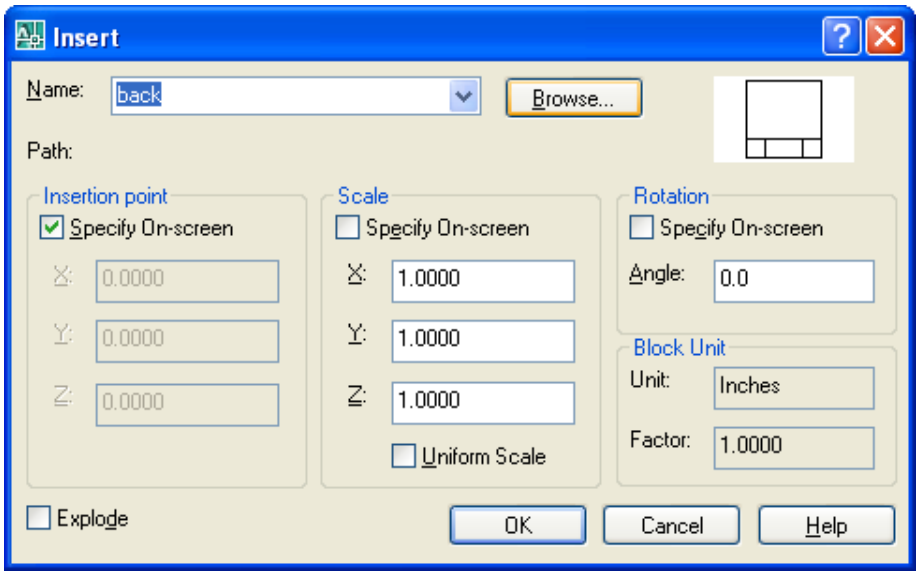


Figure 16.10 – Insert Window

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. We will see in Figure 16.21 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. 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 16.21.

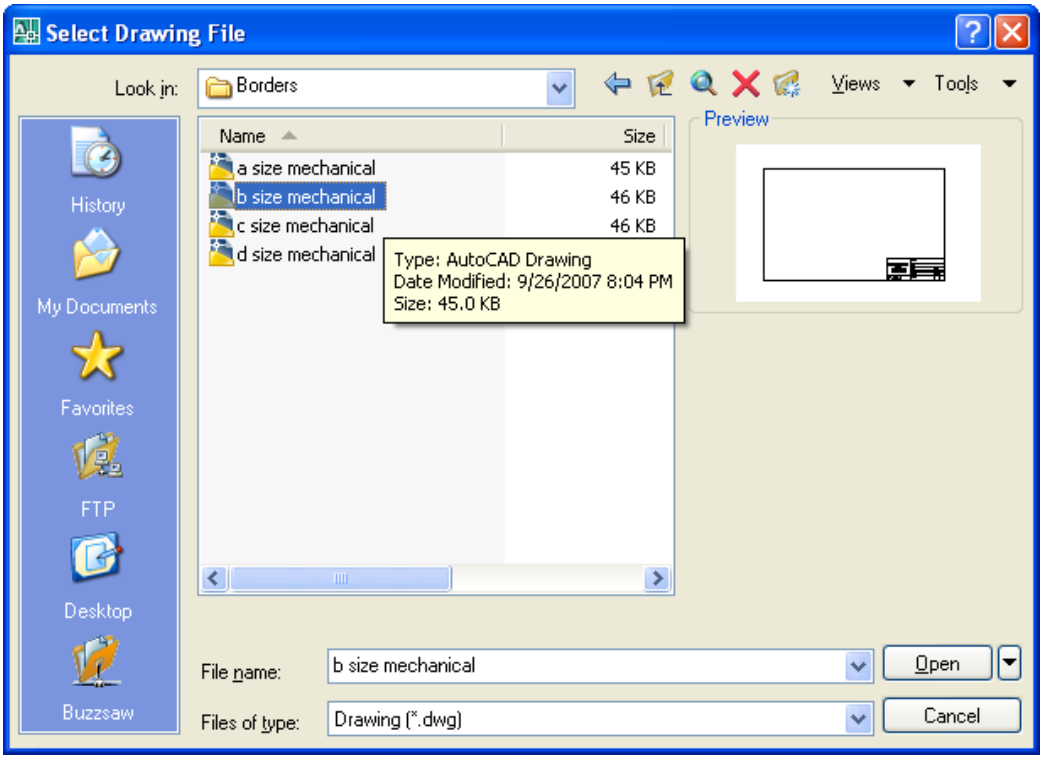


Figure 16.11 – Select Drawing File Window

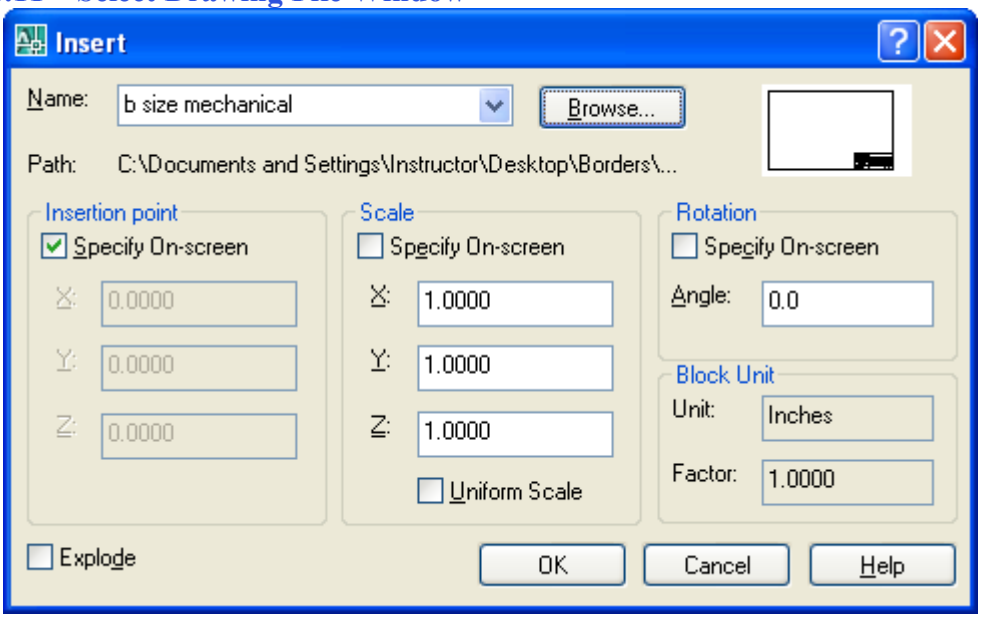


Figure 16.12 – B Size Mechanical

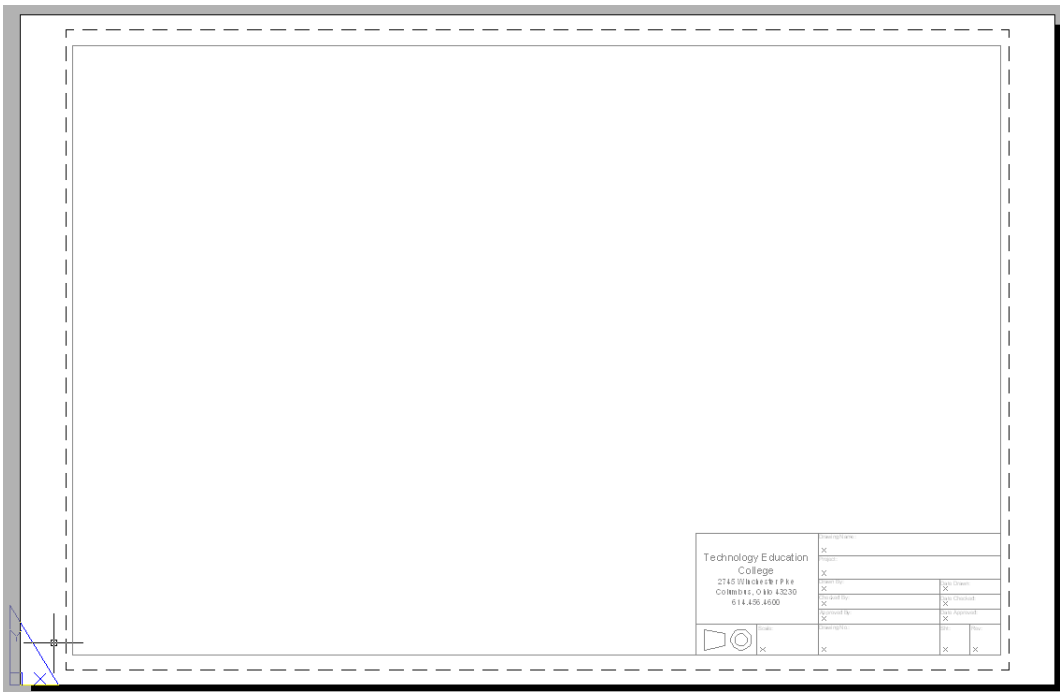


Figure 16.13 – B Size Mechanical in Layout1

When we insert the B-size border, the block will be on the 0 layer. Move the border and title block to the Border layer. Now we will insert the three orthographic views we need for the drawing.

Creating a Paper Space Layer

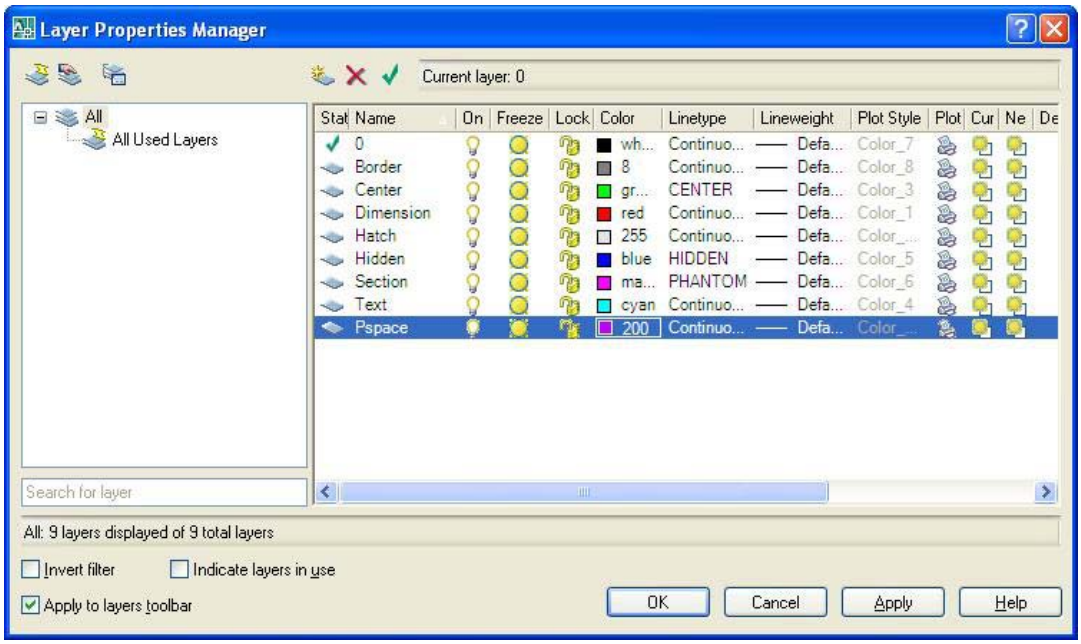
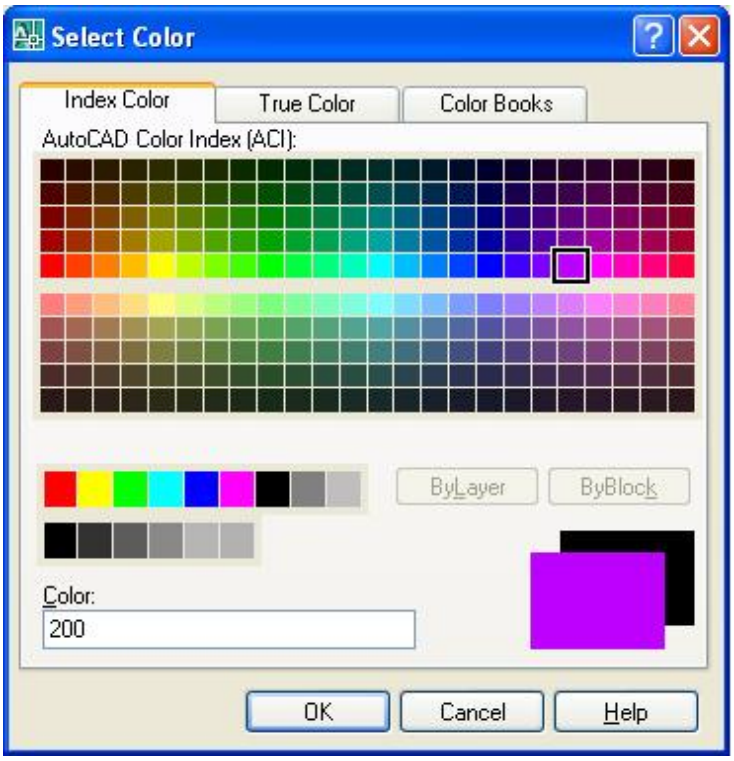


Figure 16.14 – Layer Properties Manager

Before we insert the three orthographic and one isometric views, we need to make a new layer called PSpace, which we will eventually freeze so that no one can see the windows into 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 is invisible.



Select the Layer Property Manager on the Layers toolbar and the Layer Property Manager window will open on the graphical screen. But the new layer button and a new layer will appear among the list of layers that we created in the mechanical template. Name the layer the PSpace and we gave the layer the color number 200. Double click on the layer to make the layer current so when we draw the MView window, the windows will appear on the PSpace layer.

Figure 16.15 – Select Color Window for the Paper Space Layer

Making Four Windows into Model Space using MView

At the command line type MView, and the prompt *"Specify corner of viewport or [ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/Restore/2/3/4] <Fit>:"* appears on the command line. We desire to have the three orthographic and one isometric view, so the type 4 and press Enter. Next, we will draw a window on the graphical screen as shown in Figure 16.16. Once the box is drawn, four viewports looking into Model Space are shown with the part oriented in the southwest isometric view. The next step is to position the views in their orthographic positions.

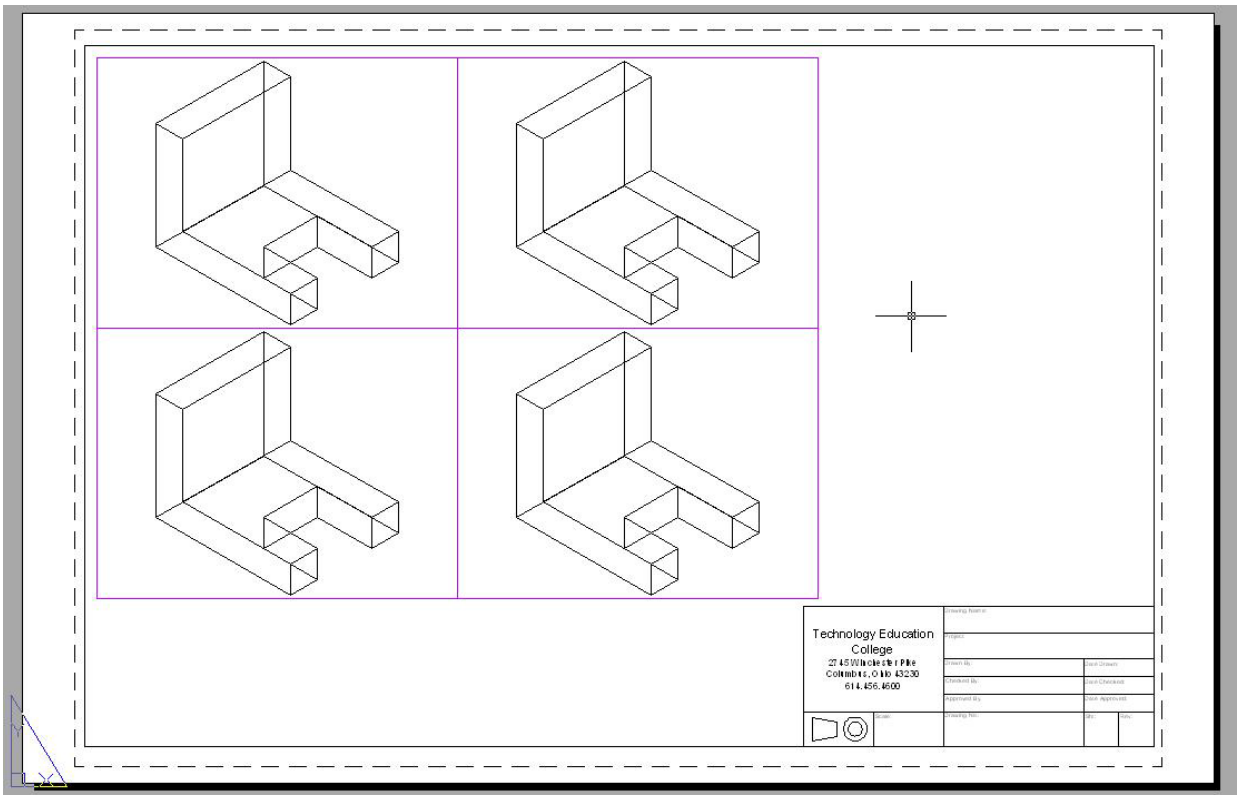


Figure 16.16 – B Size Mechanical in Layout1

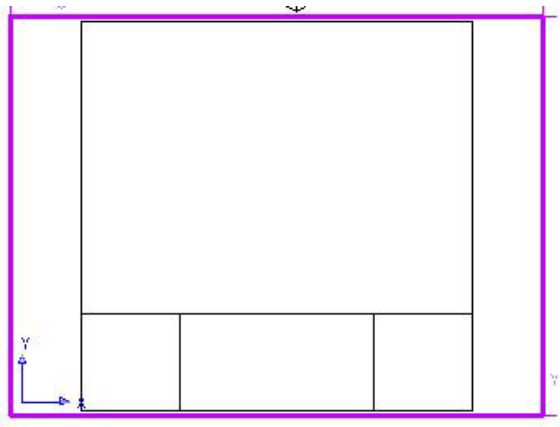


Figure 16.17 – Setting the Front View

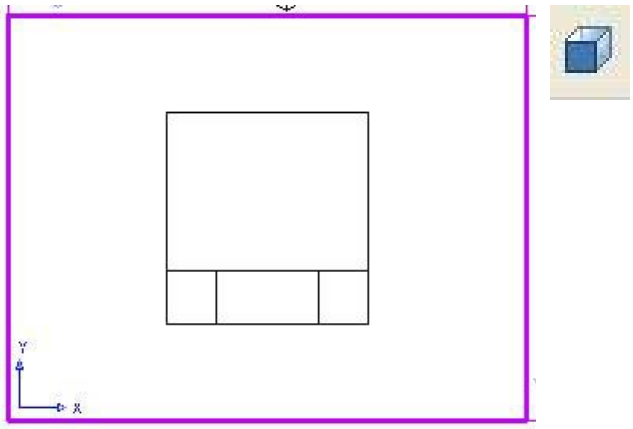
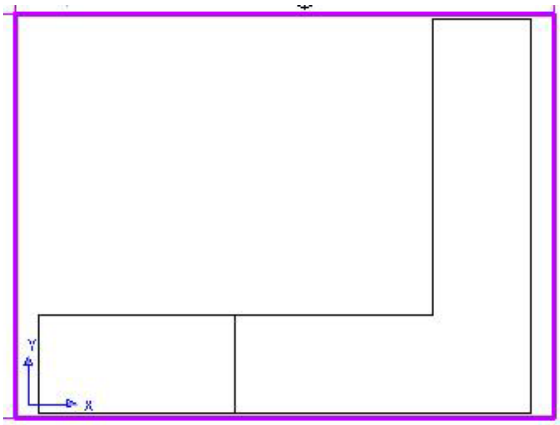


Figure 16.18 – Zooming 1 Times Paper Space

Double click in the lower left hand window and we will see the UCS icon appear. In this viewport, we are going to place the front orthographic view. On the View toolbar, select the Front view as shown in Figure 16.17. To bring this view into one to one scale, we will type Zoom, and then 1xp. 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 16.18.



Double click in the lower right hand window and we will see the UCS icon appear. In this viewport, we are going to place the Right orthographic view. On the View toolbar, select the right view as shown in Figure 16.19. To bring this view into one to one scale, we will type Zoom, and then 1xp. Again, by typing 1xp, the Model Space view is adjusted so we can dimension the part accurately in Paper Space.

Figure 16.19 – Setting the Right View

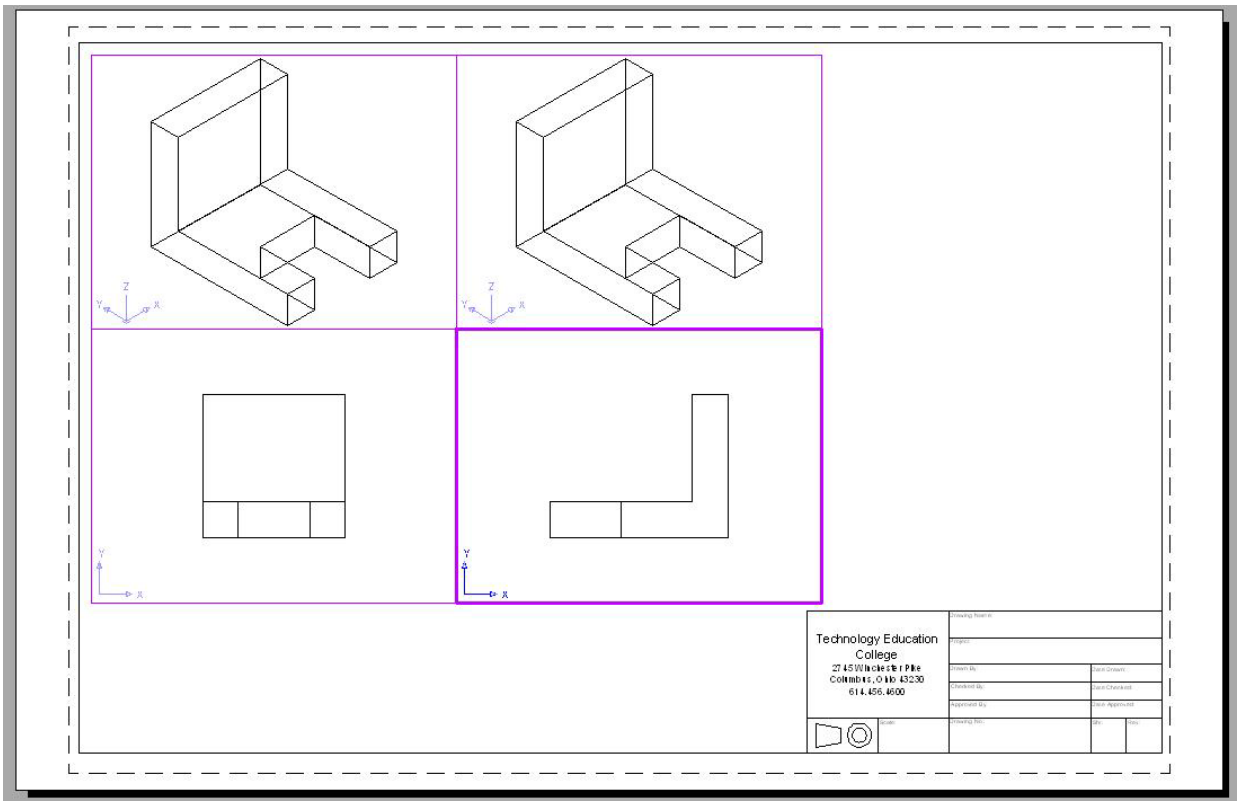
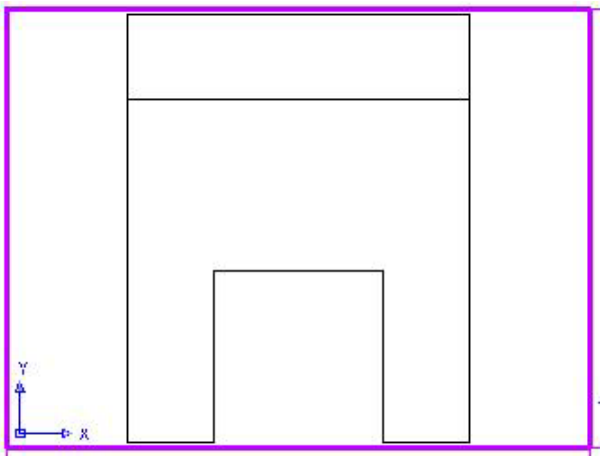


Figure 16.20 – Scaling the Right Orthographic View



Double click in the upper left hand window and we will see the UCS icon appear. In this viewport, we are going to place the top orthographic view. On the View toolbar, select the Top view as shown in Figure 16.21. To bring this view into one to one scale, we will type Zoom, and then 1xp. In Figure 16.22, we now can see all three orthographic views in proper position and properly scaled.

Figure 16.21 – Move the Top View

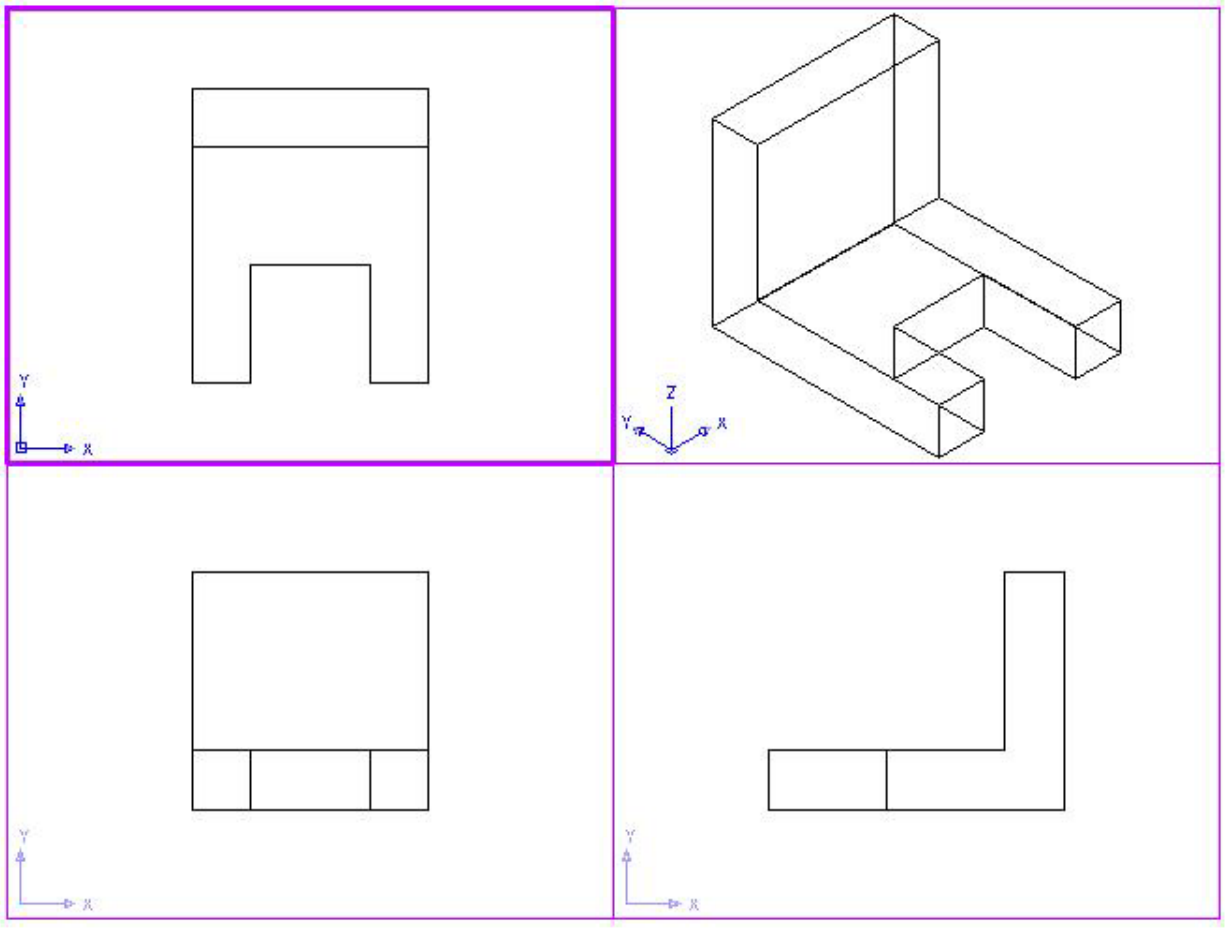


Figure 16.22 – Three Orthographic Views Finished

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 dimension in the front, right and top views. In the 20th century, isometric drawings could be costly to make, but with 3D solids and MView, this view is practically free. We will place the southwest isometric view in the upper right corner of the drawing.

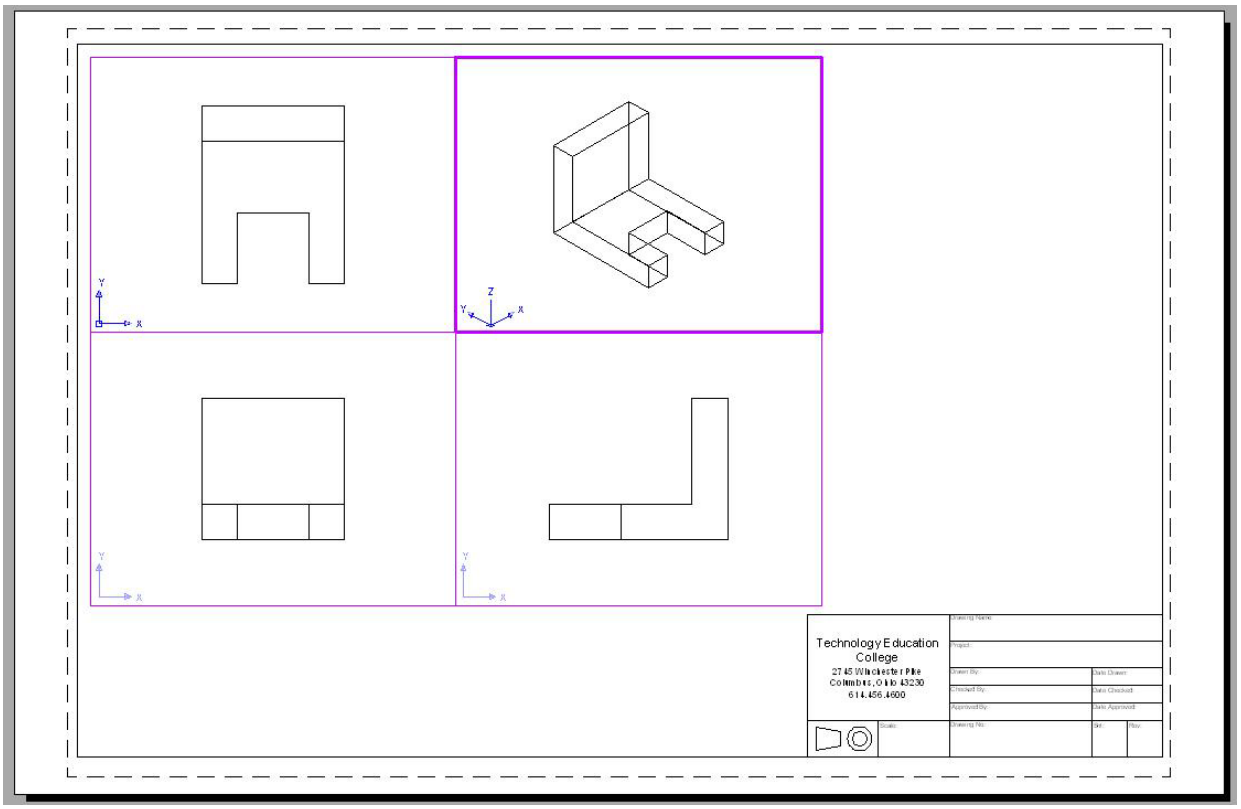


Figure 16.23 – Inserting the Southwest Isometric View

Double click in the upper right hand window and we will see the UCS icon appear. In this viewport, we are going to place the Southwest isometric view. On the View toolbar, select the SW Isometric view as shown in Figure 16.23. To bring this view into one to one scale, we will type Zoom, and then 1xp.

You might have noticed that the 3D solids in the orthographic views have continuous lines where there should be hidden or no lines. In the front view in the lower right viewport, we will make one line a hidden line by selecting the Tools Menu, next Palettes and then Visual Styles. Click in the viewport to make the lower right window current. In the Visual Style Manager window as shown in Figure 16.25, change the Obscured Lines to Dashed in the right hand view.

Repeat the process for the Isometric View but in this case change the Obscured Lines to Off. When we take a look at the drawing in the plot preview mode, we will see that the lines are hidden (dashed) of invisible.

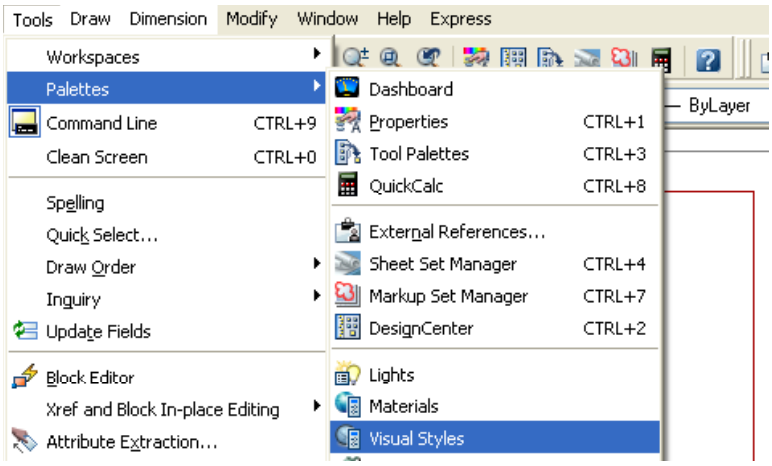


Figure 16.24 – Opening the Visual Styles Property Box

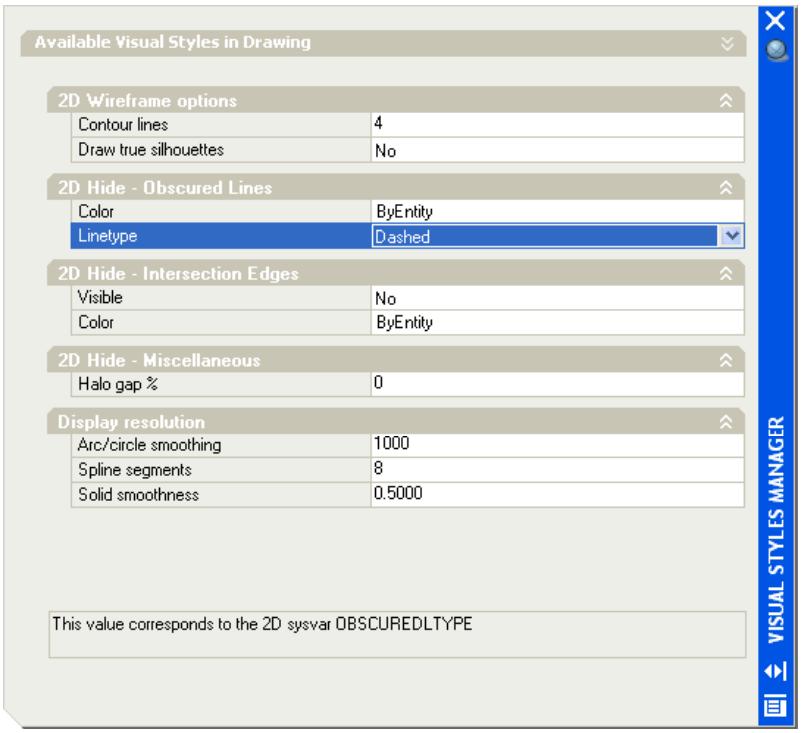
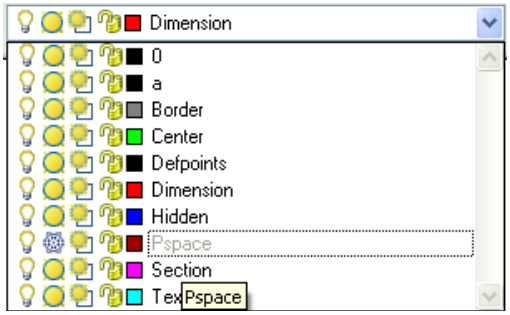


Figure 16.25 – Setting the Obscured Lines to Dashed



Now, double click in the Paper Space border and the UCS icon looking into Model Space will disappear. Select all four MView windows and in the Layers Display window select PSpace and as shown in Figure 16.26, freeze the layer. The four MView viewports will disappear.

Figure 16.25 – Freezing the PSpace Layer

Dimension the Drawing

When learning how to dimension 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 in which the part is facing when the machining or construction practice is in progress. For example, if a worker is cutting a 1 inch notch in the top plate of the Bracket, then 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 guaranteeing that there are not any missing measurements.

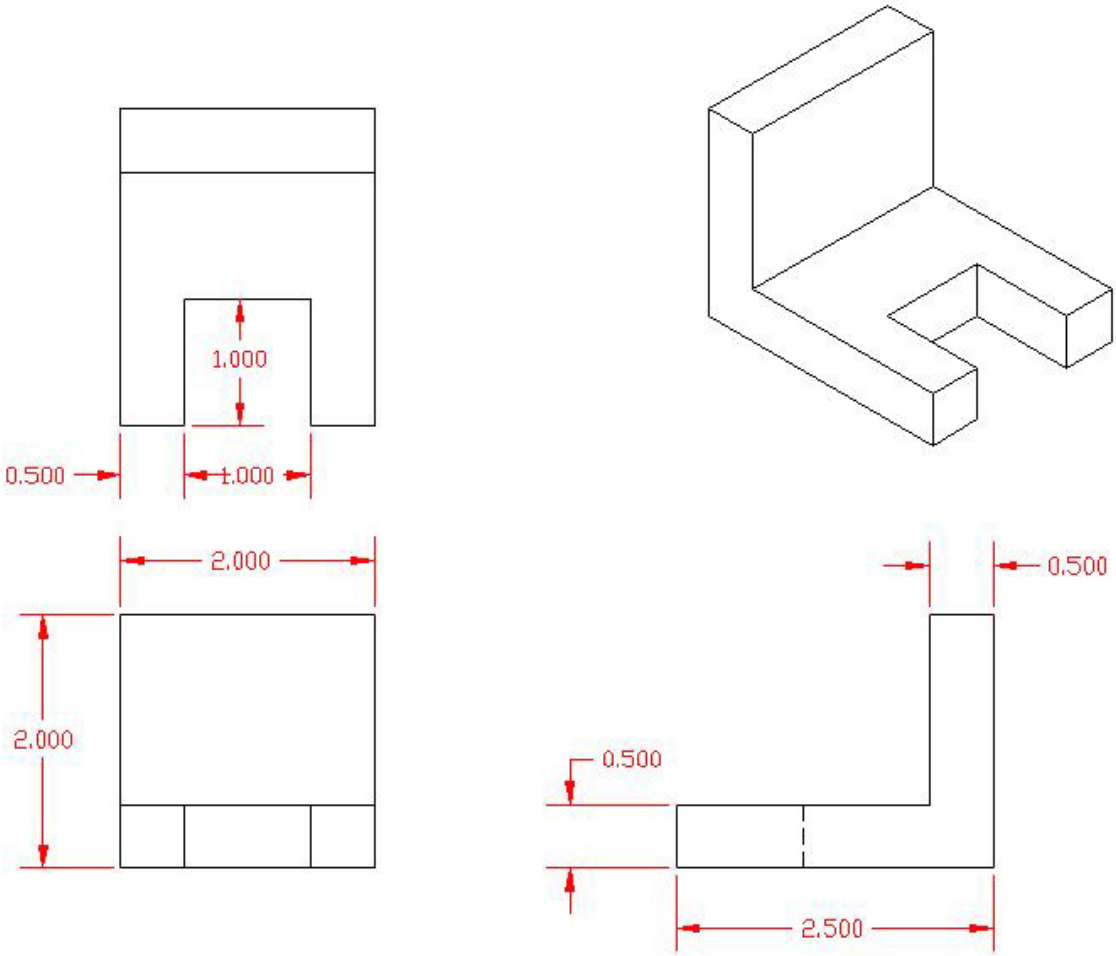


Figure 16.27 – Dimensioning the Solids Problem

In Figure 16.27, we see the dimensions in position that will satisfy a project manager.

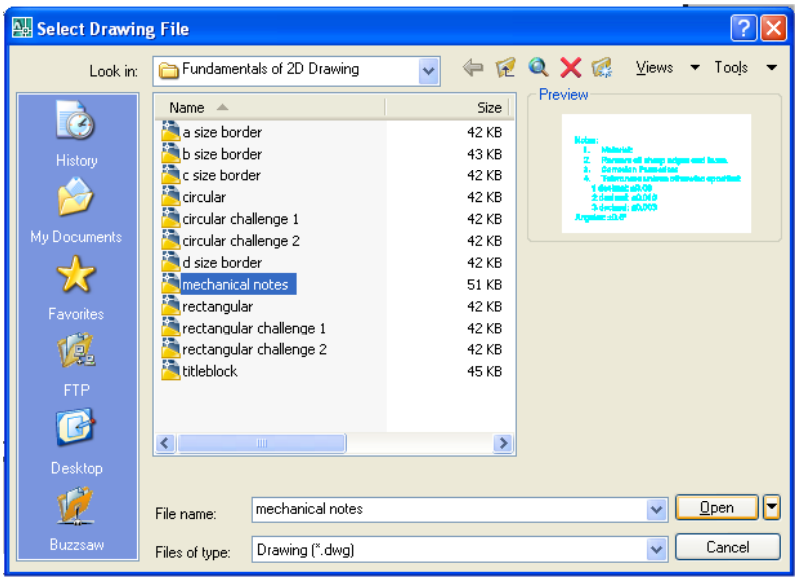
Complete the Title Block

Zoom close to drawing's titleblock and give the drawing the name Bracket. The project is the Fundamentals of 3D drawing. 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 World Class Cad 1147 Rice Ave. Gahana, OH 43230 www.worldclasscad.com	Drawing Name: Bracket			
	Project: Fundamentals of 3D Drawing			
	Drawn By: Charles Robbins		Date Drawn: 09/28/2007	
	Checked By: Mark Bughman		Date Checked: 09/28/2007	
	Approved By:		Date Approved:	
	Scale: 1=1	Drawing No.: 1001	Sht.: 1	Rev.: A

Figure 16.28 – Filling Out the Title Block

Inserting a Standard Notes into the Paper Space Layout



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 as shown in Figure 16.29. Use the Browse button to find the "mechanical notes" in our Fundamentals of 2D Drawing folder. Press the Open button to proceed to the Insert window.

Figure 16.29 – Inserting the Metric Mechanical Notes

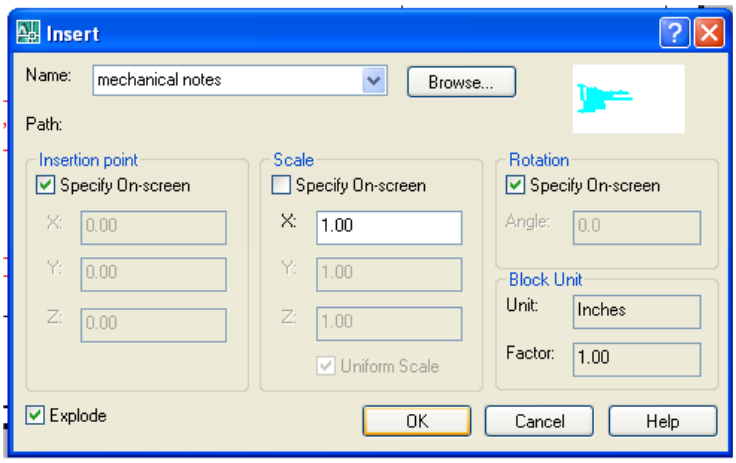


Figure 16.30 – Check Explode in the Insert Window

Now, pick the OK button on the Insert window, pick a point to the right of the Solids problem to place the notes. Next, we will modify our standard mechanical notes using the Edit Text tool.

Modifying Standard Notes in Paper Space

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 we 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: 2.00 Nylon

And remove the corrosion note

Select "OK" to close the window. (See Figure 16.31)

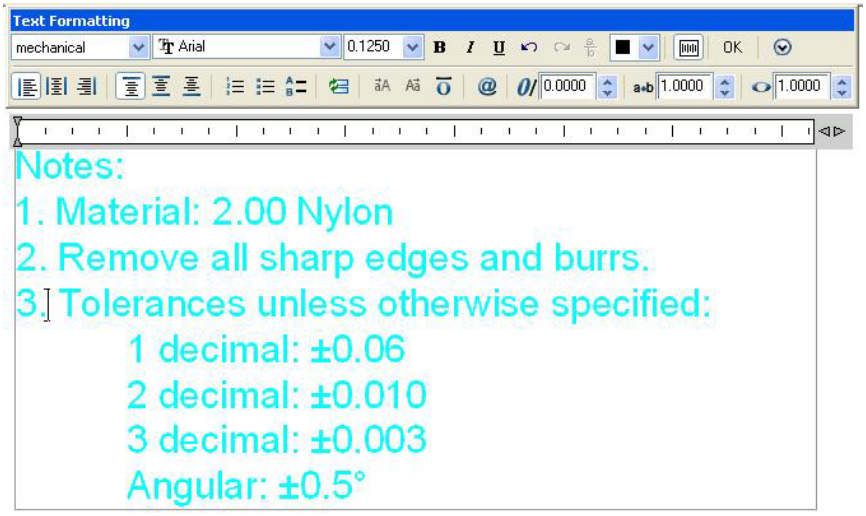


Figure 16.31 – Modifying the Notes by Adding Material and Corrosion Details

If we are proficient in the use of the tools provided by a Computer Aided Design software supplier then we can create complex orthographic viewed drawings as shown in Figure 16.32 for the production release and still have the solid available for the CAD assembly. In this chapter, we created a drawing that is independent of the 3D model. For some organizations, they do not desire the complexity of 3D design and the Tenview program gives them the flexibility to design in 3D and leave there to communicate in the two dimensional arena. In another lesson, we will learn to obtain our view by opening windows into Model Space to acquire the front, right, top and isometric views.

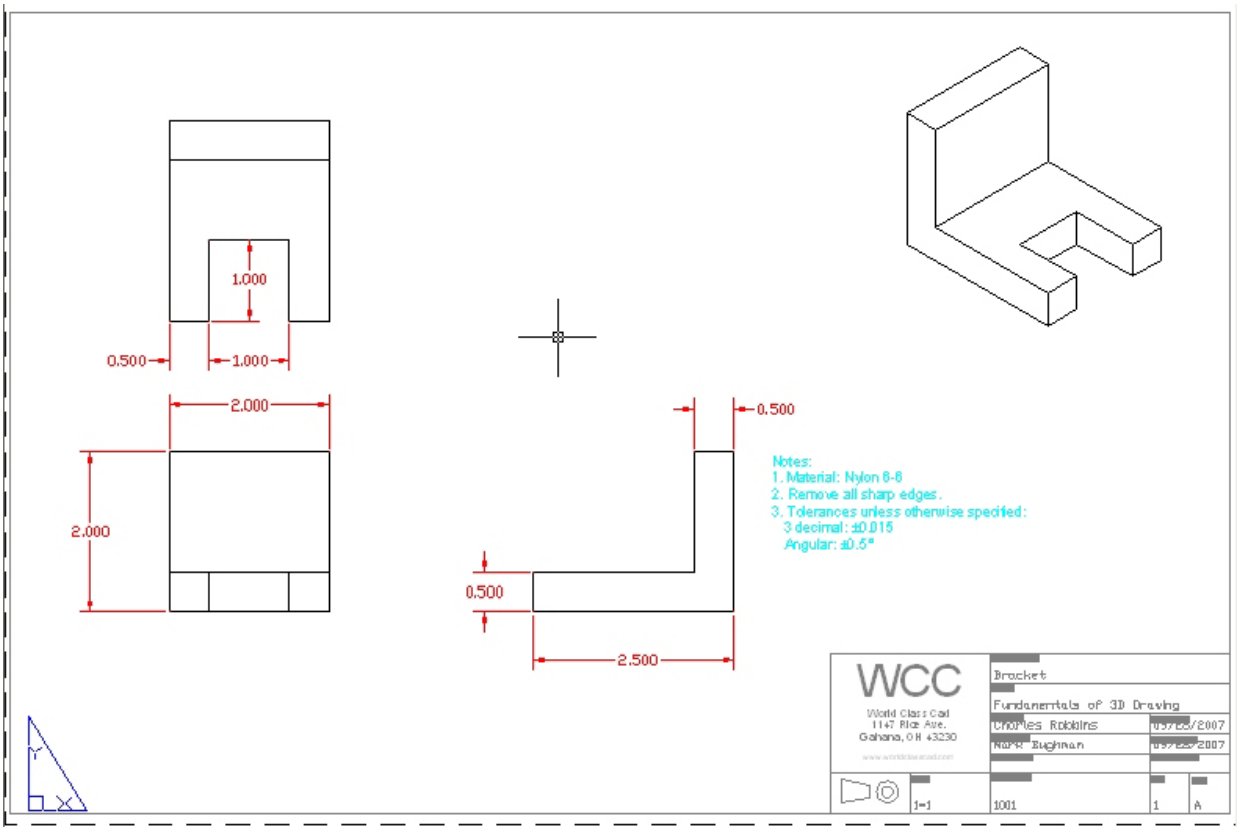


Figure 16.32 – The Finished Solids Problem 1 Drawing

*** World Class CAD Challenge 03-14 * - 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 to obtain your world class ranking.**