

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**
- **Open the Solid Part Drawing**
- **Setting the Page Size in Paper Space**
- **Insert the B Sized Border with Insert Block**
- **Creating a Paper Space Layer**
- **Making Four Windows into Model Space using MView**
- **Insert an Isometric View**
- **Dimension the Drawing**
- **Complete 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 each view requires. One of the biggest advantages to drawing a part in three dimensions is that you can construct 2D orthographic views in a matter of seconds from the 3D solid part. In the last chapter you did this using a LISP routine called Tenview. In this chapter you will achieve the same goal by using a different technique.

In this chapter, you will learn the procedure for completing a multiview drawing using the MView command, your dimensioning skills, and a predefined drawing border and notes. The MView command allows you to create a number of different views simultaneously. You will use your sketch from chapter two as a guideline for dimensioning the views. You will utilize the previously created borders and notes you have already created. Because you will be using different entities that have already been drawn and checked, you can create a drawing featuring the different views efficiently and without error.

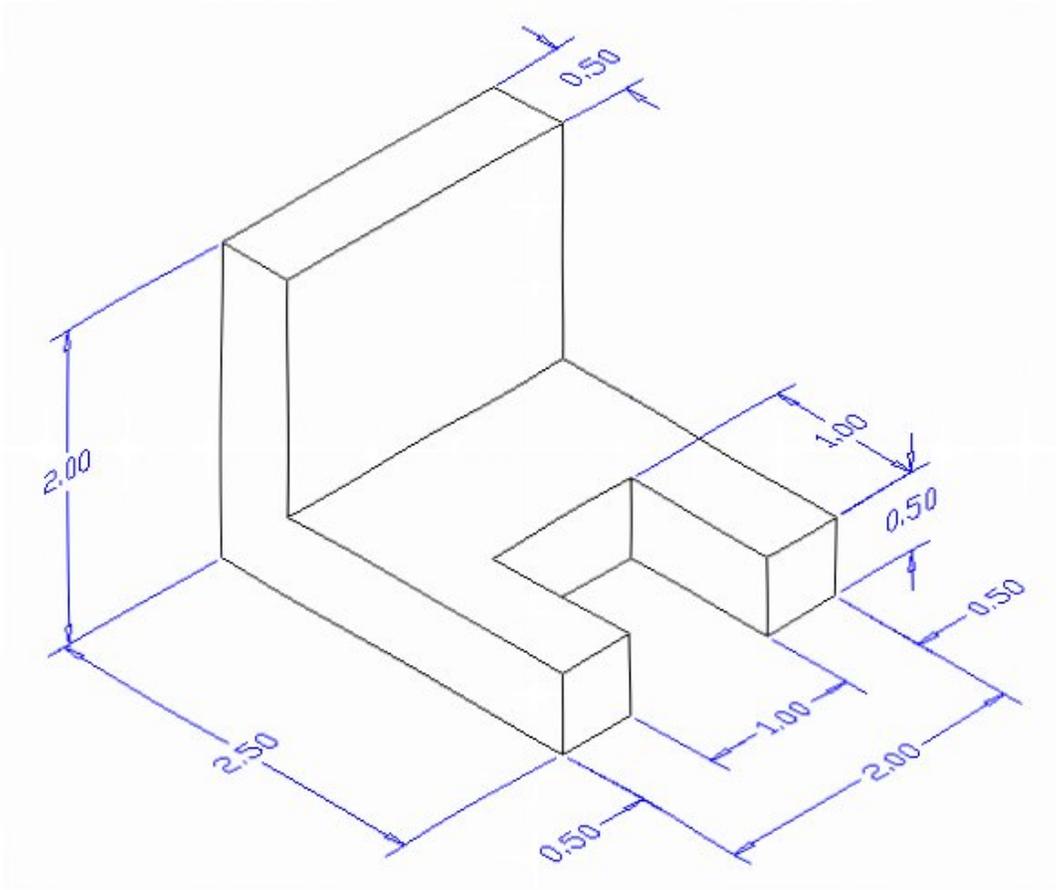


Figure 16.1 – Problem One Sketch

You will still need the sketch from Problem 1 because you may 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 subtract valuable time from the process. There are nine measurements on the sketch, so expect to see nine on the finished drawing. On a drawing of this shape, we also assume 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. You can type `Hide` at the command line to see the drawing as shown in Figure 16.2. Use the `Hide` command to view the part without confusion; the `Hide` function removes the rear tessellation lines.

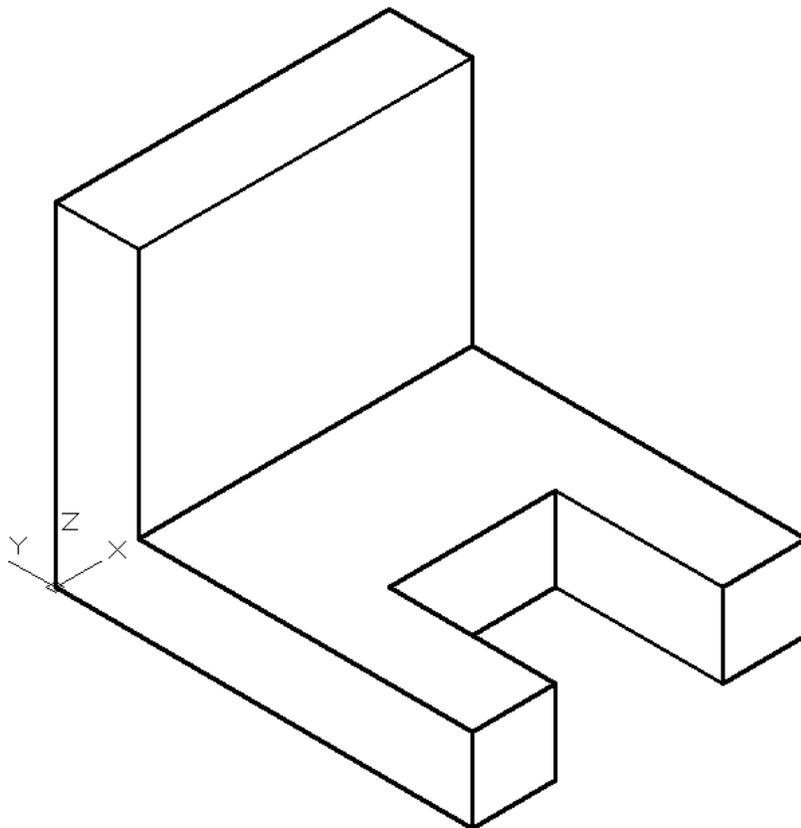


Figure 16.2 – Open Solids Problem 1

You will see that the solid resides in Model Space. The majority of the work you are about to do will be in Paper Space. You are now ready to leave Model Space and use the `MView` command to generate the orthographic and isometric views in Paperspace.

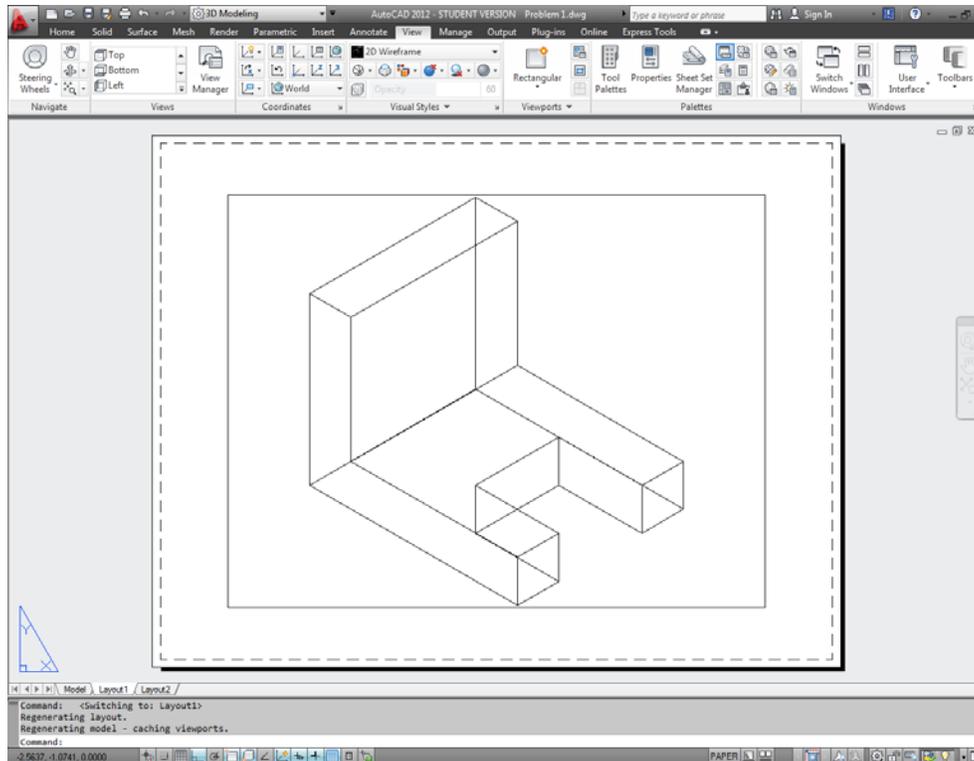


Figure 16.3 – Problem 1 in Layout1 (Paper Space)

Setting the Page Size

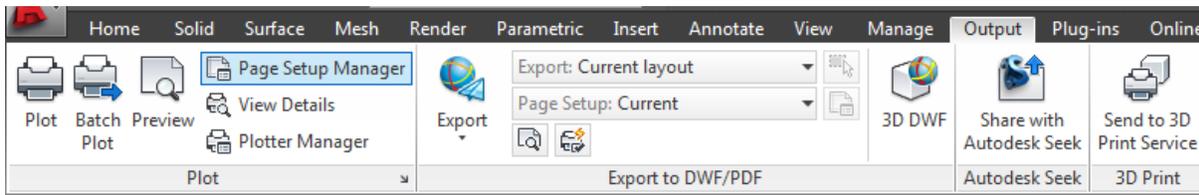


Figure 16.4 – Page Setup Manager, Output tab

Select the Output tab and pick the Page Setup Manager tool on the Plot panel as shown in Figure 16.4. Select Layout 1 and click Modify to reach the Page Setup – Layout 1 window. You probably remember configuring these plot settings from the Bracket problem in your 2D training. If you are printing at home you can set the printer to your preferred device. Otherwise, you can just set it to None. Also, change the Paper Size to 8.5 x 11 inches and make sure the Drawing orientation is set to Landscape. You should be able to leave all of the other settings alone so that your Page Setup window appears as shown in Figure 16.5. Hit OK and then Close the Page Setup Manager. Before inserting the border block, select the window surrounding the view of the 3D part so that the grips appear and hit DELETE. The window into Model Space will disappear. Your layout should appear in Paperspace as shown in Figure 16.6.

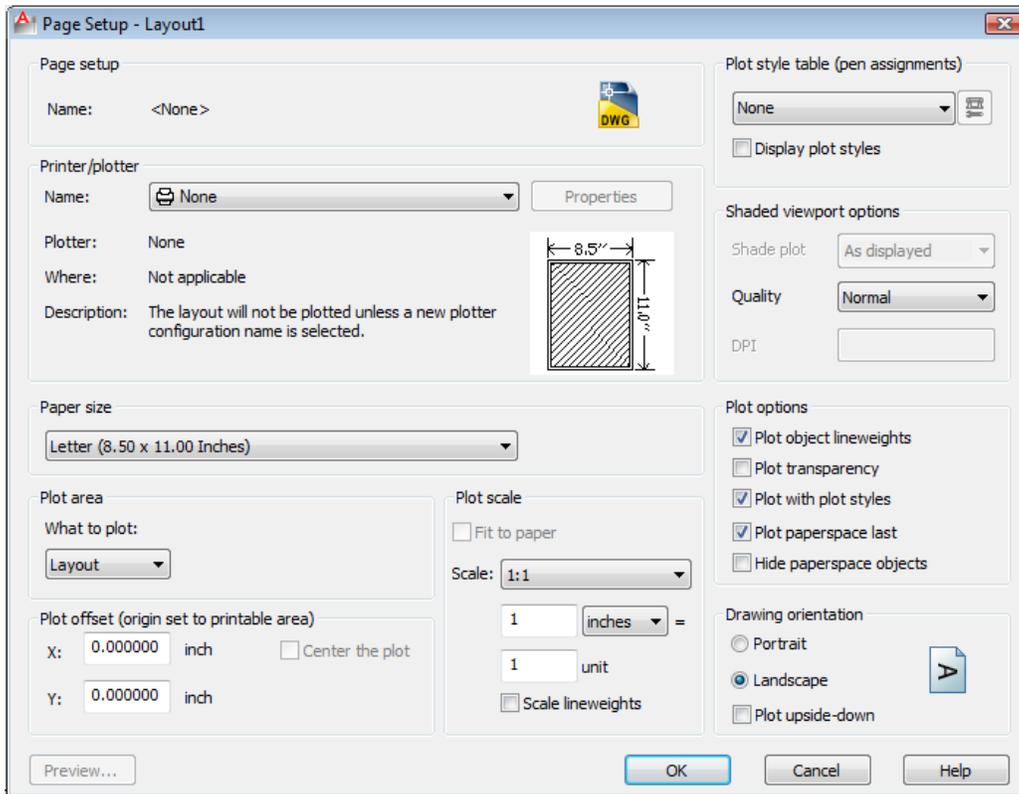


Figure 16.5 – Page Setup – Layout 1

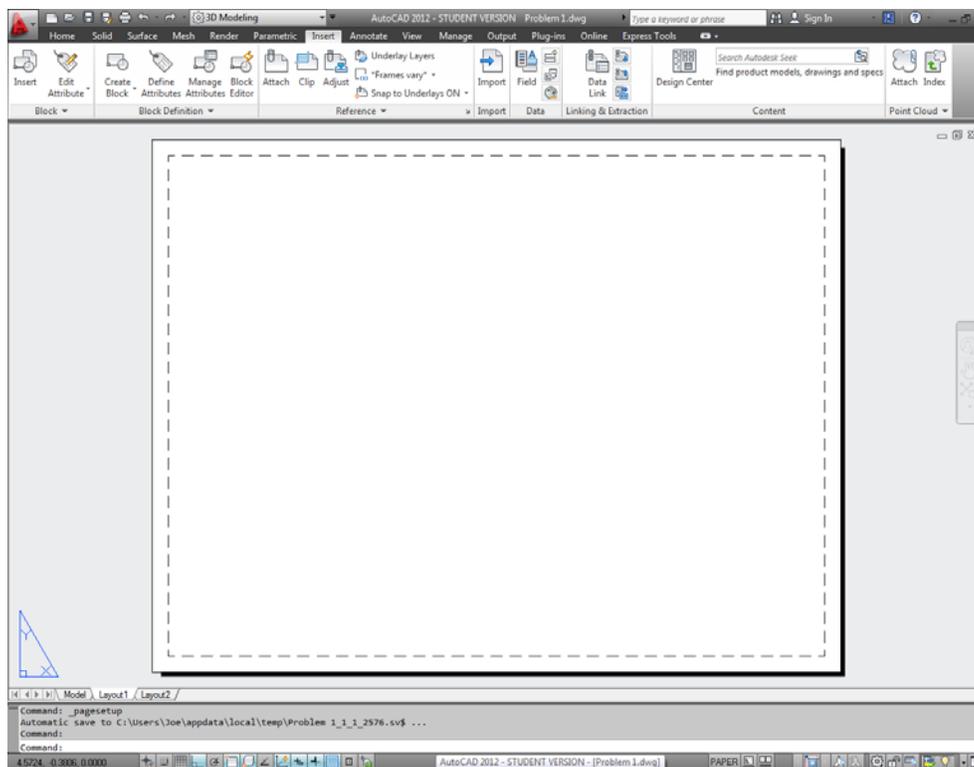
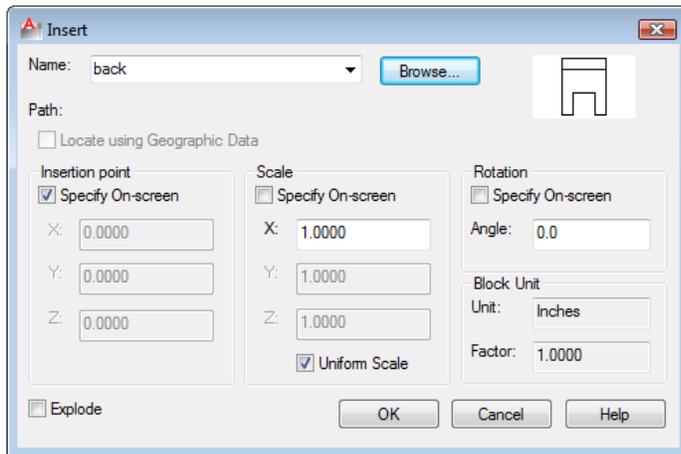


Figure 16.6 – A Size Sheet in Layout View

Inserting the A Sized Border

To insert the block containing the A-Size Border into Layout 1, select the Insert tab and pick the Insert tool on the Block panel. The Insert window (Figure 16.7) will appear. Use the Browse button to find the “a size border” in your Borders folder (Figure 16.8). Make sure the Specify On-screen box is checked in the Insertion Point area of the Insert window, and then hit OK (Figure 16.9).



After hitting the OK button on the Insert window, pick the insertion point for the border, placing the entire contents inside the dotted printable area (Figure 16.10)

Figure 16.7 – Insert window

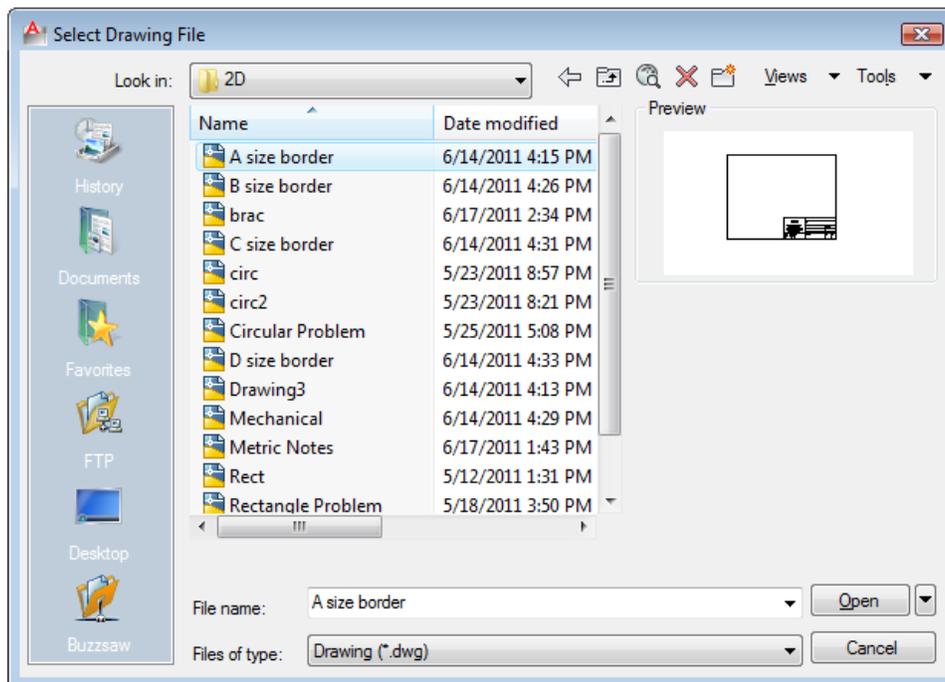


Figure 16.8 – Select Drawing File window

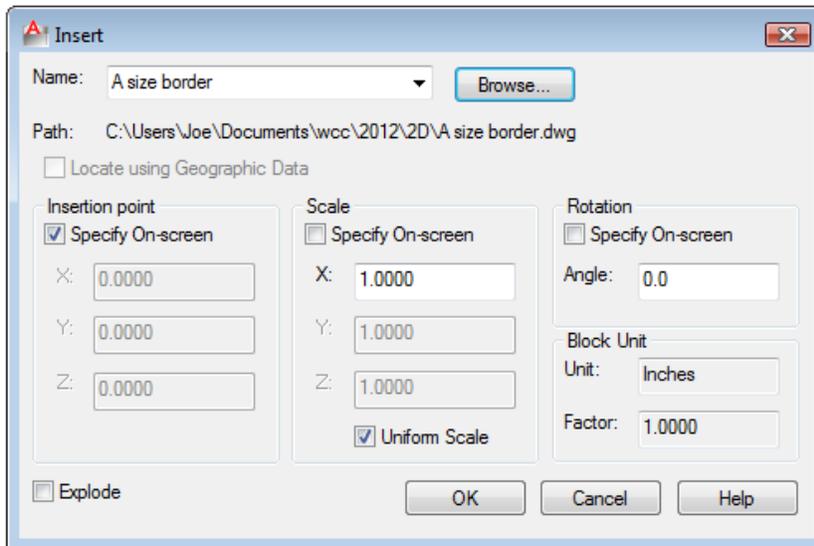


Figure 16.9 – A Size Border

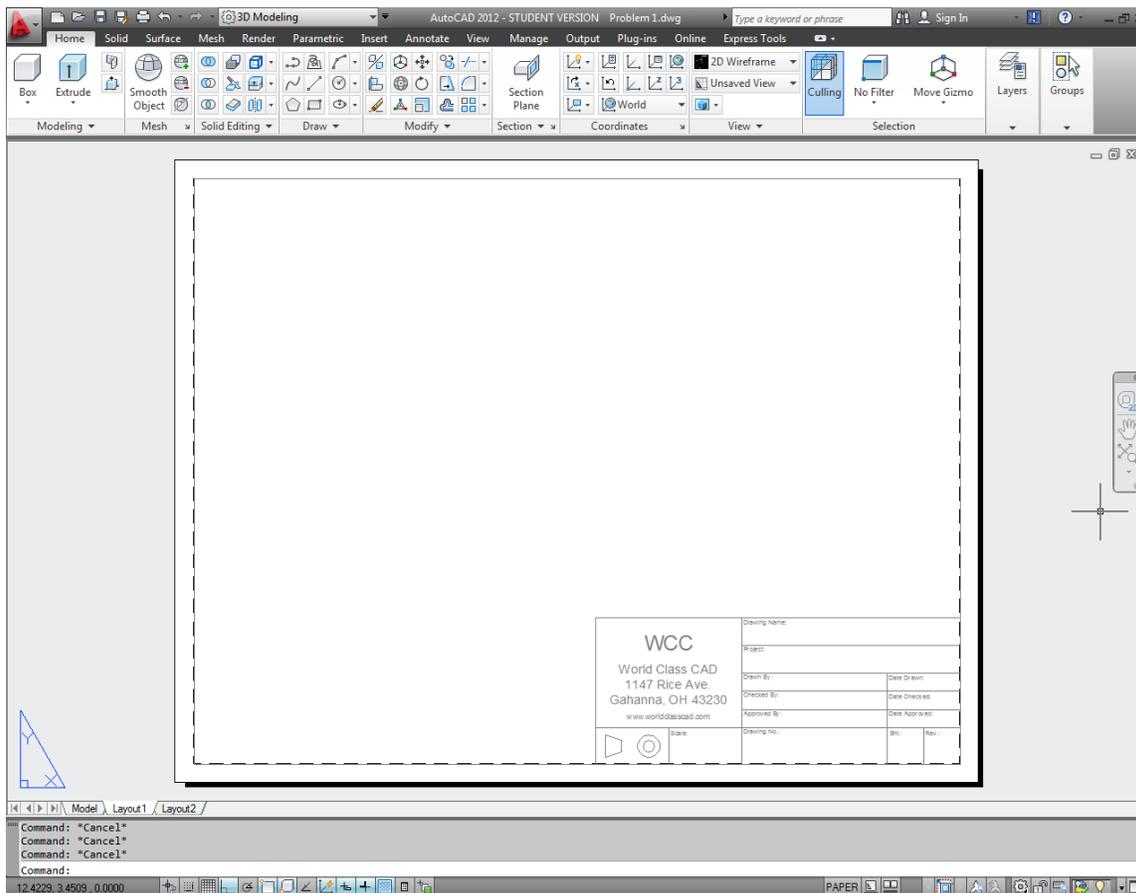


Figure 16.10 – A size border placed

You will insert the A size block on the 0 layer. Select the border and use the Layer Control list box to change it to the Border layer (Figure 16.10).

Creating a Paper Space Layer

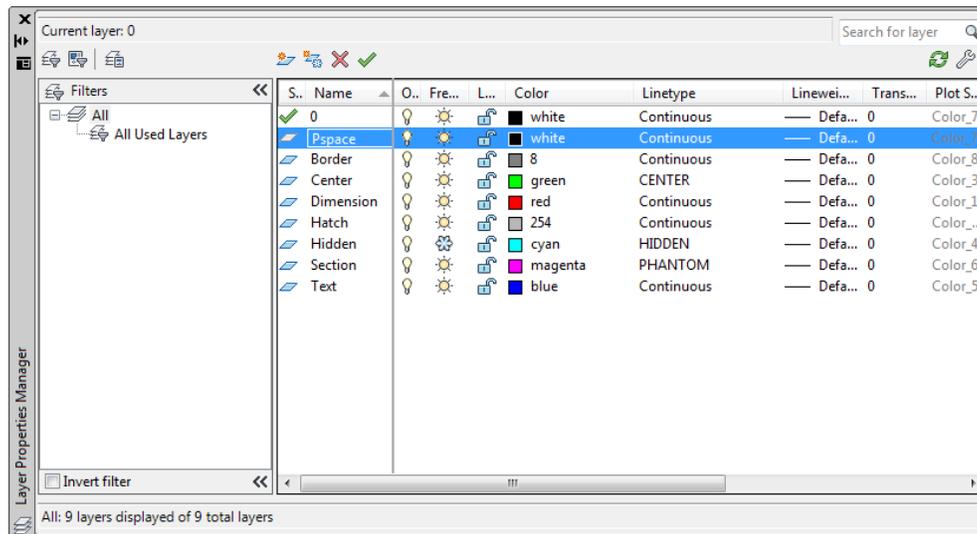


Figure 16.11 – Layer Properties Manager

Before you insert four viewports with MView, you need to make a new layer called PSpace that you will use to control the border around each viewport. The MView command creates windows, called viewports, in Paper Space that allow the viewer to see view of Model Space. You will put the viewports on the Pspace layer and then freeze that layer later so that the borders will be invisible.

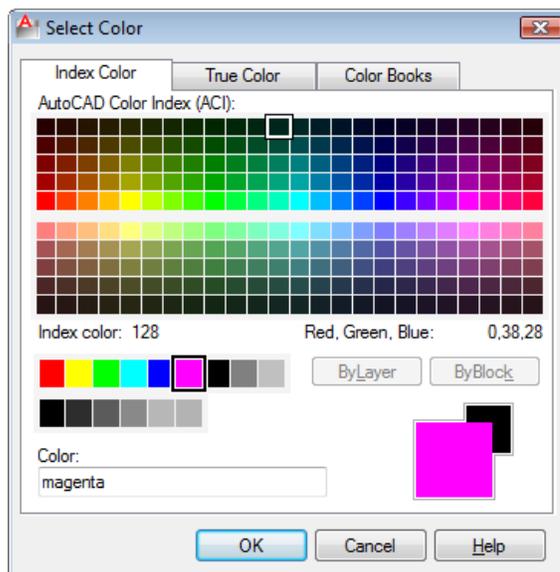


Figure 16.12 – Select Color window

Select the Layer Properties tool on the drop-down arrow of the Layers panel and the Layer Properties Manager window appear. Create a new layer named Pspace and change the color of the layer to Magenta. Double click on the new layer to make it active so that the viewports will appear on the PSpace layer.

Making Four Windows into Model Space using MView

Type MView at the command line and the prompt “Specify corner of viewport or [ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/Restore/2/3/4] <Fit>:” will appear on the command line. Type 4 for the number of viewports and hit ENTER. Next, draw a window on the as shown in Figure 16.13. Once you draw the box, four viewports looking into Model Space will appear showing four southwest views of problem 1.

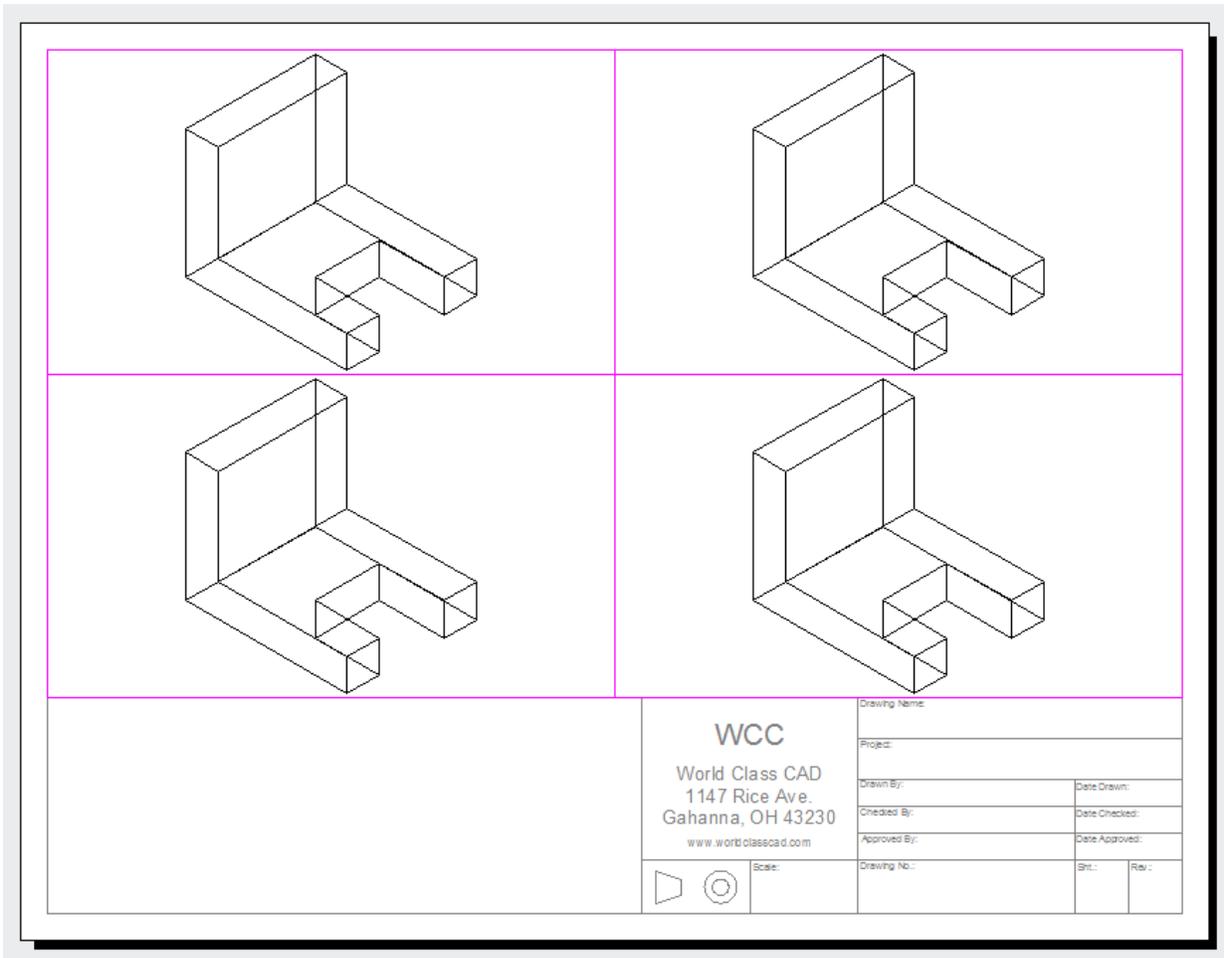


Figure 16.13 – 4 Viewports created with Mview

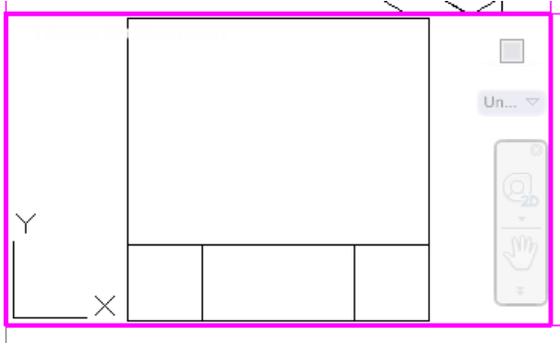


Figure 16.14 – Setting the Front View

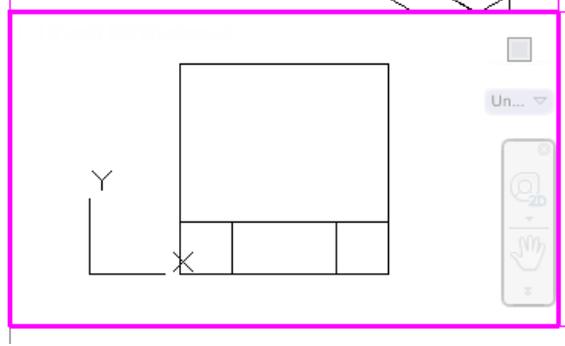


Figure 16.15 – Zooming 1xp

Double click in the lower left hand viewport and you will see the UCS icon appear along with a miniature viewcube and navigation bar. You will place the front view in this viewport. Select the Front view from the View panel so the viewport appears as shown in Figure 16.14. Zoom extents the viewport to center the part. Then, to bring this view into one to one scale, type Zoom and then 1xp as shown in Figure 16.15. Double-click anywhere outside of the viewport to return to Paper Space.

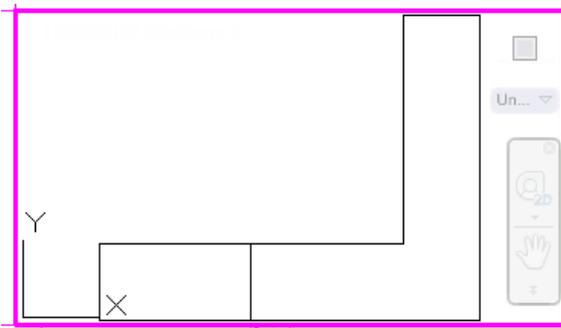


Figure 16.16 – Setting the Right View

Double click in the lower right hand viewport and you will see the UCS icon appear along with a miniature viewcube and navigation bar. You will place the right side view in this viewport. Select the Right view from the View panel so the viewport appears as shown in Figure 16.16. Zoom extents the viewport to center the part. Again, to bring this view into one to one scale, type Zoom and then 1xp as shown in Figure 16.17. Double-click anywhere outside of the viewport to return to Paper Space.

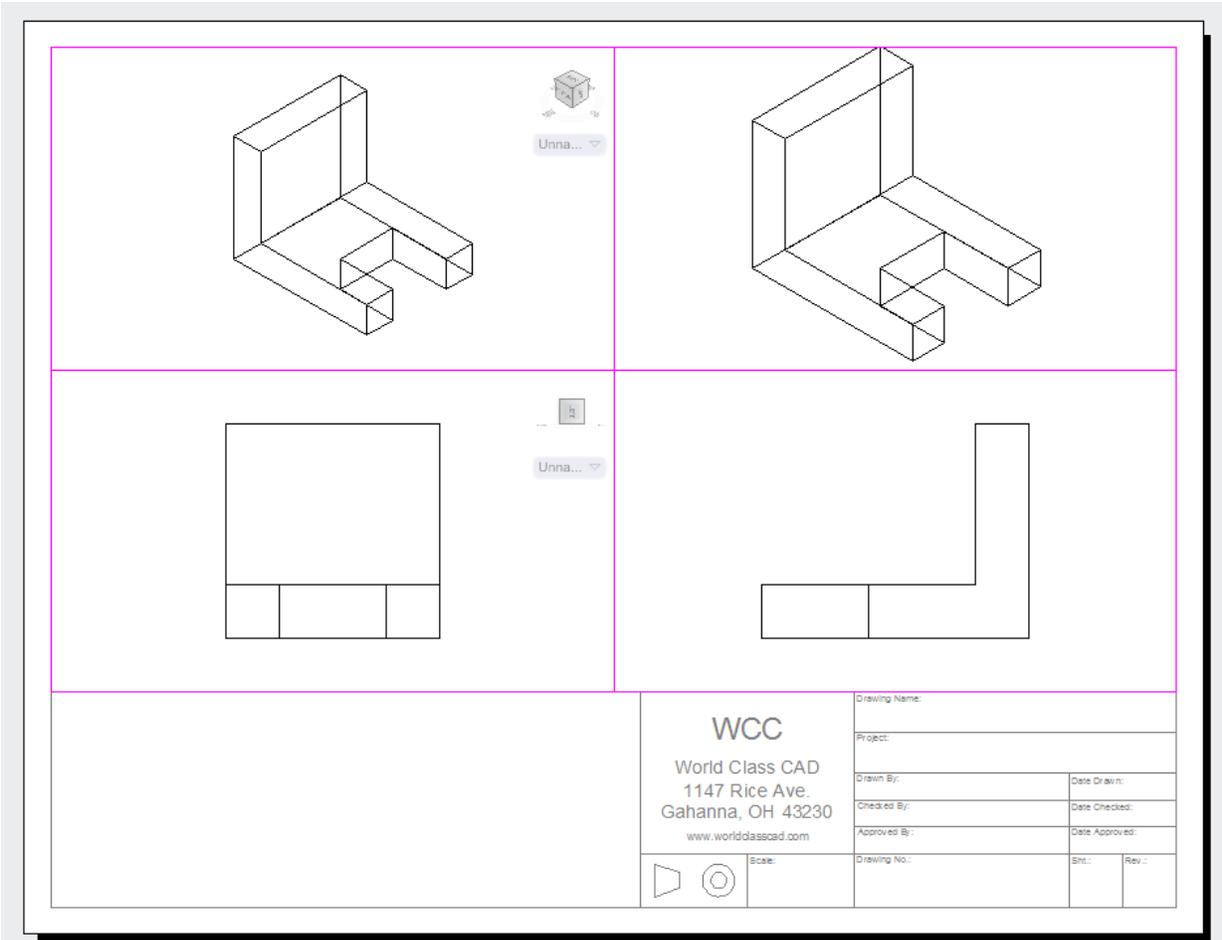


Figure 16.17 – Right orthographic view, aligned and to scale

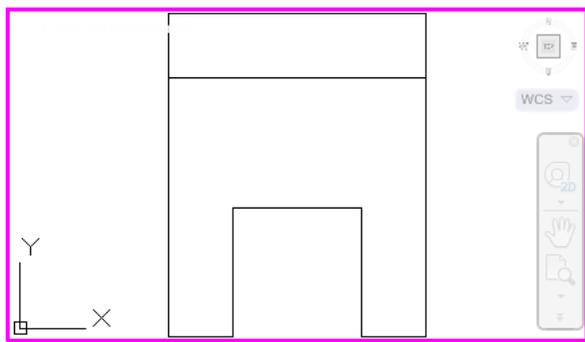


Figure 16.18 – The Top View

Double click in the upper left-hand viewport and you will see the UCS icon appear along with a miniature viewcube and navigation bar. You will place the top side view in this viewport. Select the Top view from the View panel so the viewport appears as shown in Figure 16.18. Zoom extents the viewport to center the part. Again, to bring this view into one to one scale, type Zoom and then 1xp as shown in Figure 16.19. Double-click anywhere outside of the viewport to return to Paper Space.

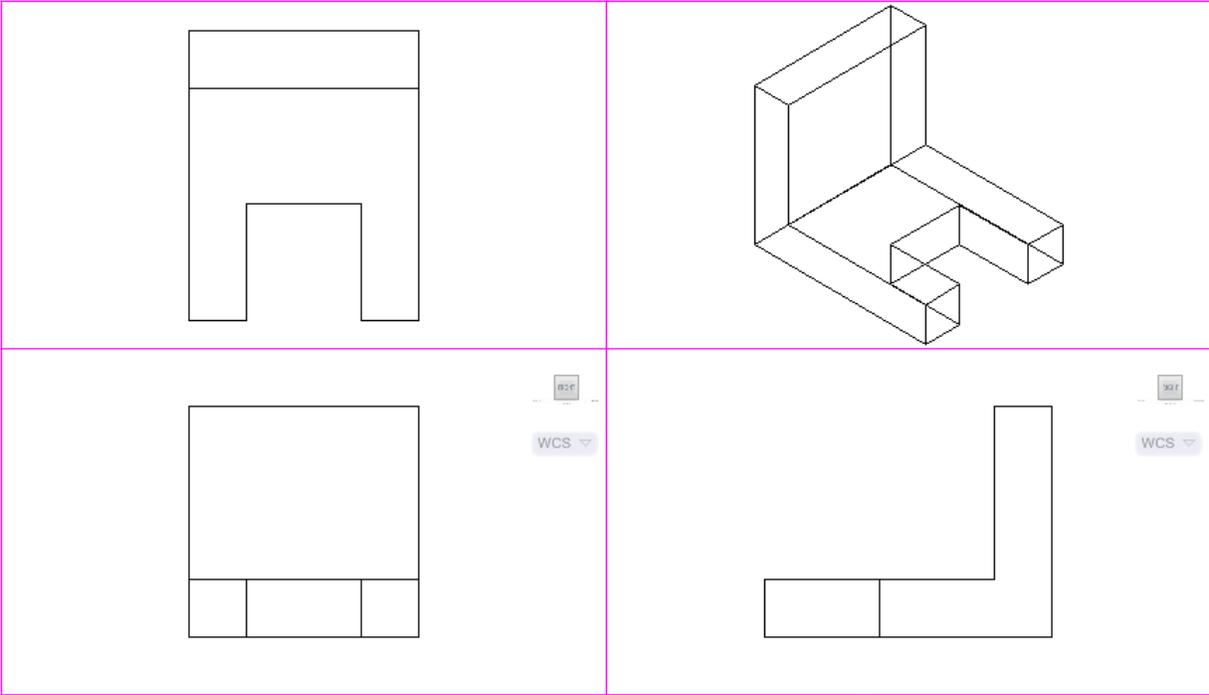


Figure 16.19 – Three Orthographic Views Finished

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 with 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 virtually painless. You will place the southwest isometric view in the upper right corner of the drawing.

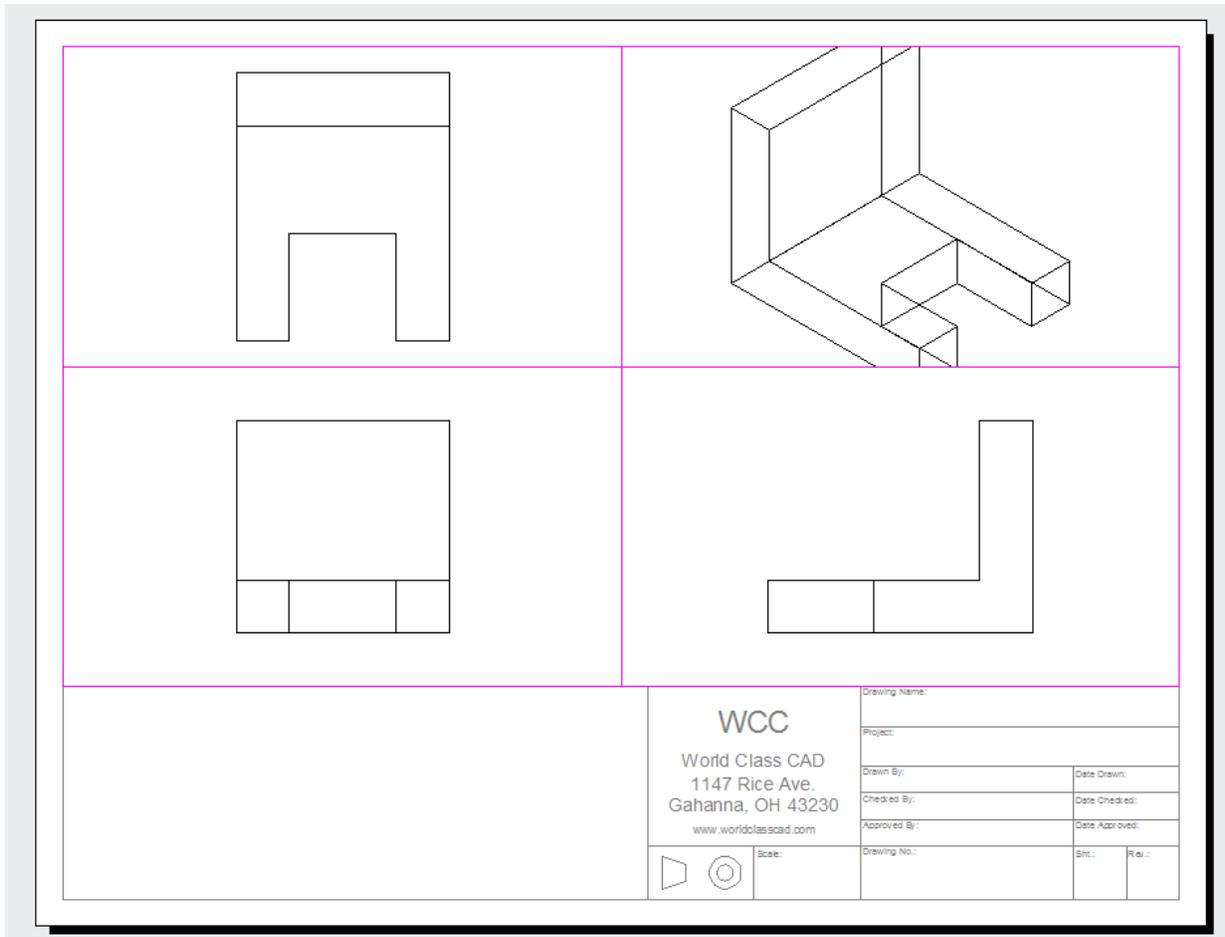


Figure 16.20 – Inserting the Southwest Isometric View

Double click in the upper right-hand viewport and you will see the UCS icon appear along with a miniature viewcube and navigation bar. You will place the isometric view in this viewport. Select the Southwest Isometric view from the View panel so the viewport appears as shown in Figure 16.20. Zoom extents the viewport to center the part. Again, to bring this view into one to one scale, type Zoom and then 1xp as shown in Figure 16.21. As you can see, the viewport may not be large enough to show the entire part, so move and resize the border as necessary, leaving room to dimension the three orthographic views. Remember that you will freeze the viewport borders so that they will not appear. Also remember to Hide all of the views so no tessellation lines appear. Double-click anywhere outside of the viewport to return to Paper Space.

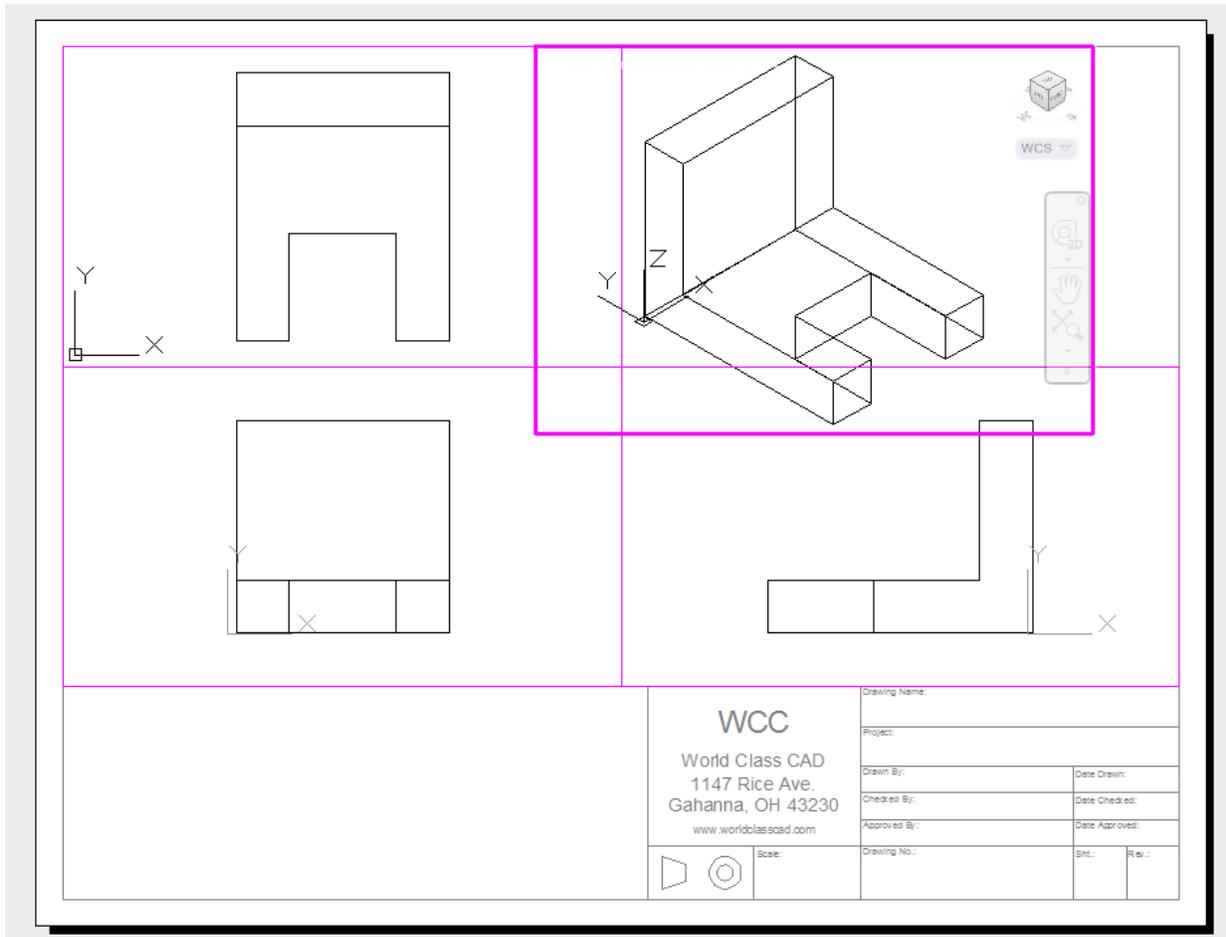


Figure 16.21 – Resizing the Isometric View’s viewport

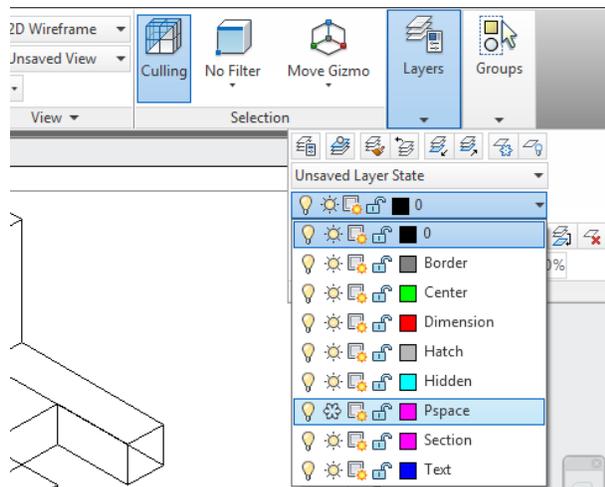


Figure 16.22 – Freezing the PSpace Layer

Now you are ready to freeze the viewports. Select the Layer control list box from the layers panel on the home tab. Find the sun icon in the Pspace layer row and click it so that it turns into a snowflake (Figure 16.22). This means that the layer has been frozen. You should see the magenta borders of the viewports disappear from paperspace. You are still able to modify the views of the individual viewports by double-clicking inside them. You can unfreeze the Pspace layer at any time to modify the size and location of the viewports.

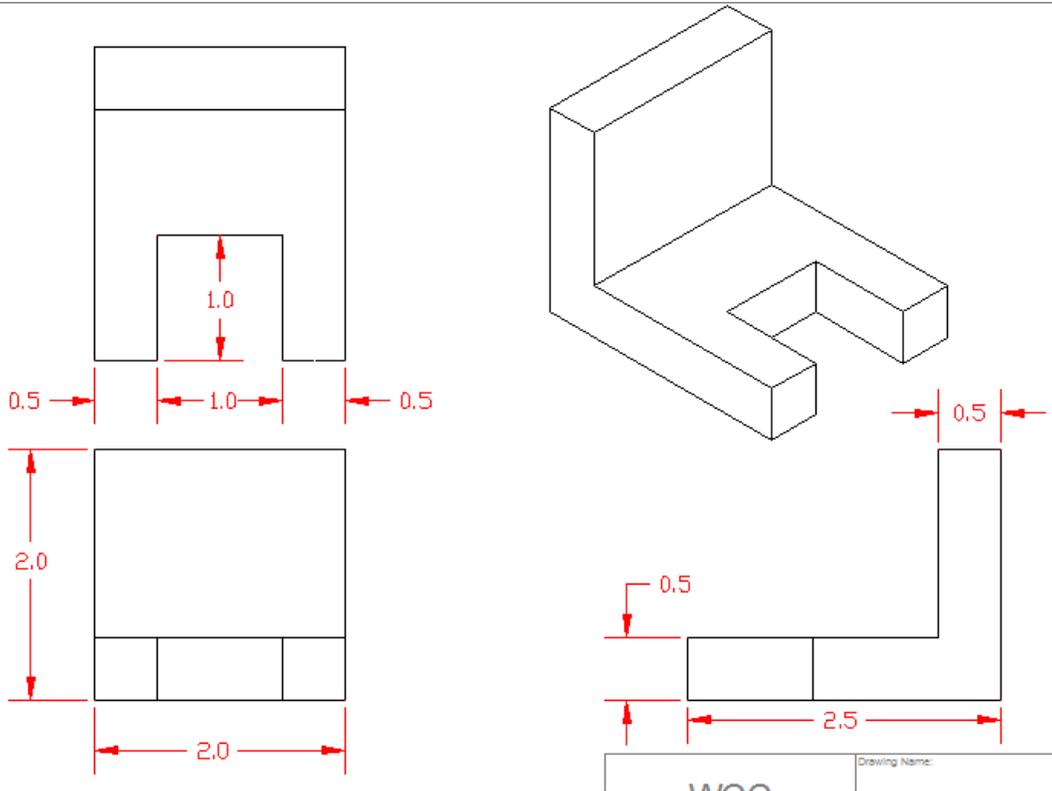


Figure 16.23 – Dimensioning the Solids Problem

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

Completing 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 2001, the sheet number is 1 and the revision is the letter A.

<p style="text-align: center;">WCC</p> <p style="text-align: center;">World Class CAD 1147 Rice Ave. Gahanna, OH 43230</p> <p style="text-align: center;">www.worldclasscad.com</p>	Drawing Name:		Solid Problem 1	
	Project:		Fundamentals of 3D Drawing	
	Drawn By:	Joe Robbins	Date Drawn:	9/15/2011
	Checked By:	Charles Robbins	Date Checked:	9/15/2011
	Approved By:	Mark Bughman	Date Approved:	9/15/2011
	Scale:	Drawing No.:		Sht.:
	1:1	2001		1
				Rev.:
				A

Figure 16.24 – Filling out the title block

Inserting Standard Notes into Paper Space

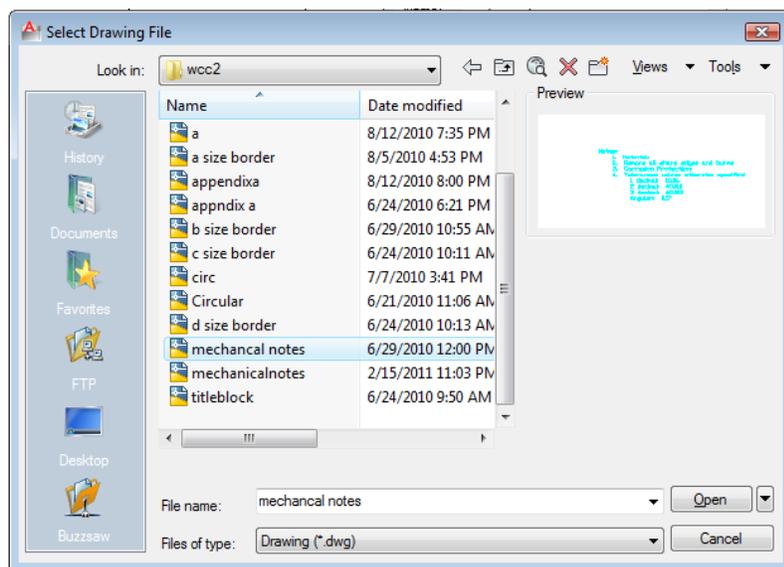


Figure 16.25 – Inserting the mechanical notes

To insert the block containing the standard notes into paperspace, select the Insert tool on the Block panel and the Insert window will appear. Use the Browse button to find the “mechanical notes” in your Fundamentals of 2D Drawing folder (Figure 16.25). Hit the Open button to proceed to the Insert window as shown in Figure 15.32.

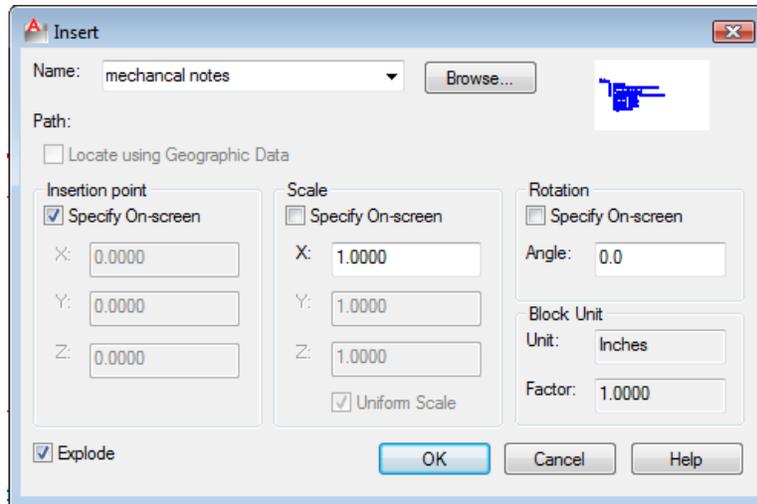


Figure 16.26 – Check Explode in the Insert Window

Check the Explode box in the Insert window and then hit OK. Pick a point to below the front view to place the notes. Next, you will modify the standard mechanical notes using the Edit Text tool.

Modifying Standard Notes in Paper Space

To add text to the standard notes, double-click the notes to open the Text Editor tab. Add these changes to the notes, making sure to renumber the list appropriately.

1. Material: 2.00 Nylon
- and
- remove the corrosion note

Select Close Text Editor from the Close panel to close the window (Figure 16.27).

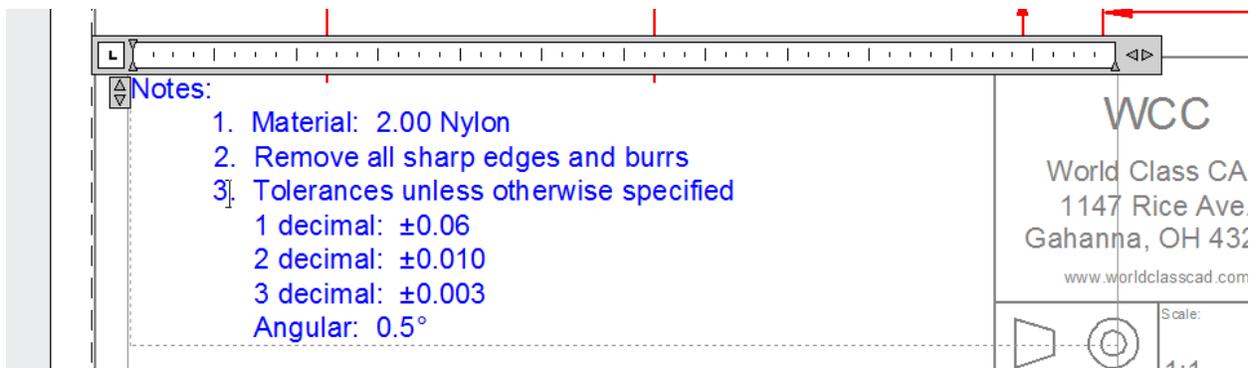


Figure 16.27 – Modifying the notes

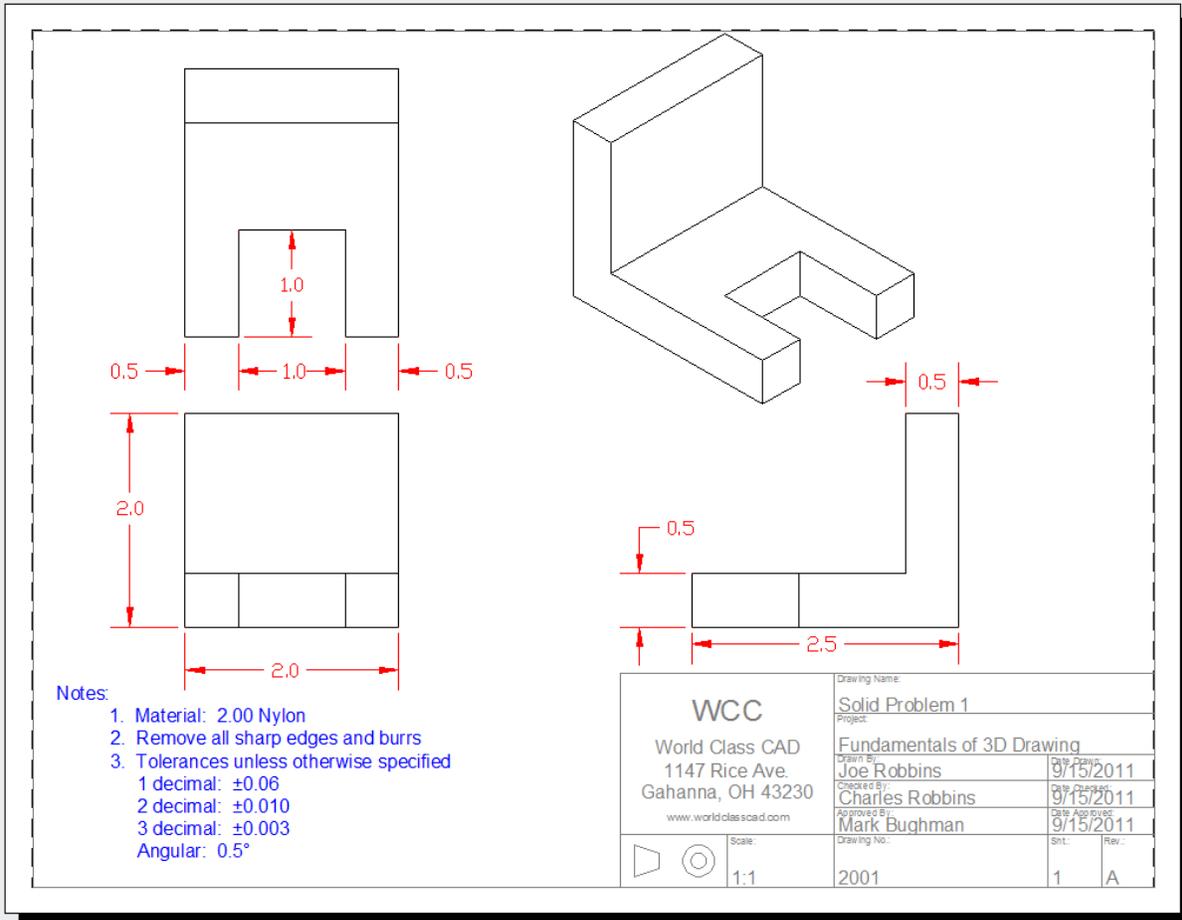


Figure 16.28 – The finished Solid Problem 1 drawing

* World Class CAD Challenge 03-16 * - 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.