

Chapter 15

Tenviewing a Solid

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

- **Converting 3D Solids to 2D Orthographic Views**
- **Opening the Solid Part Drawing**
- **Loading Tenview.LSP**
- **Running the Tenview Program**
- **Setting the Page Size**
- **Inserting the A Sized Border with Insert Block**
- **Inserting the Orthographic Views and Moving Them into Position**
- **Inserting an Isometric View**
- **Dimensioning the Drawing**
- **Completing the Title Block**
- **Inserting Standard Notes**
- **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 using a special program.

In this chapter, you will learn a procedure for completing a multiview drawing including a border and notes using a LISP program and your dimensioning skills. The simple LISP routine repetitively uses the **Solprof** command to make 2D views of each side of the part. You will use our sketch from Chapter Two, which describes the dimensions for the project, as a guideline to dimension 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.

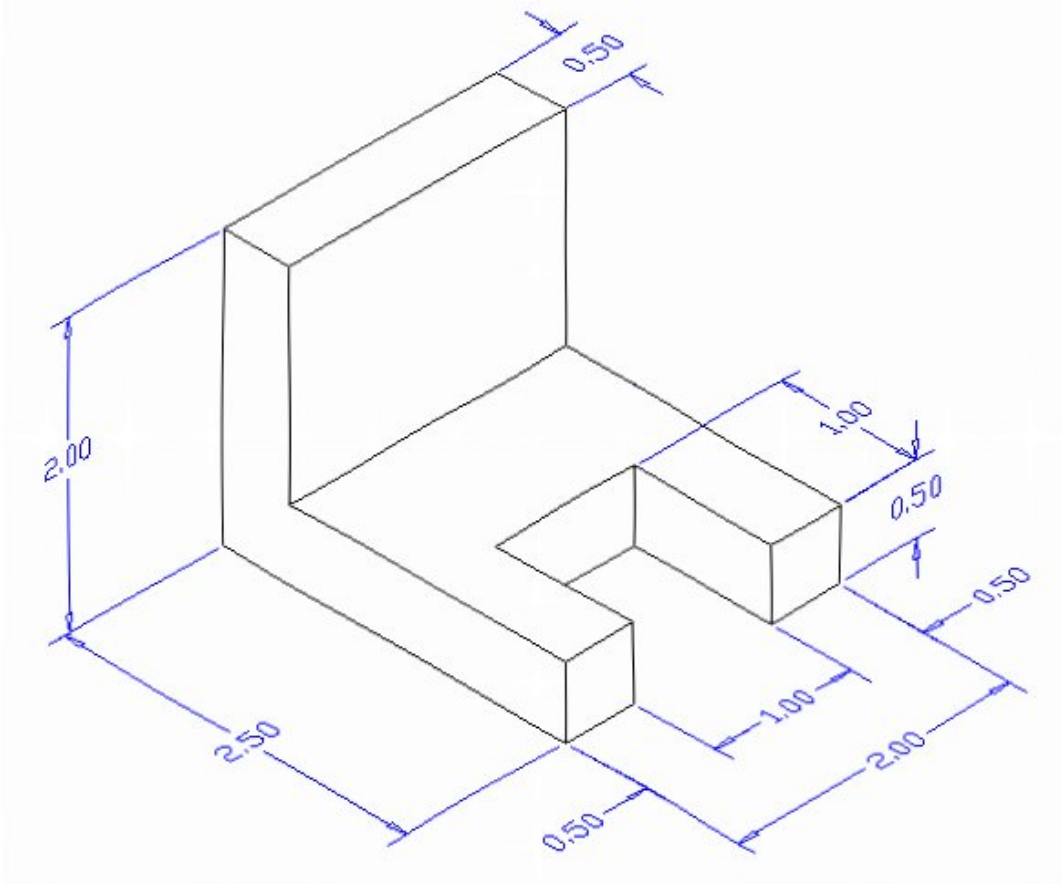


Figure 15.1 – Problem One Sketch

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.

Opening 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 15.2. Use the `Hide` command to view the part without confusion; the `Hide` function removes the rear tessellation lines.

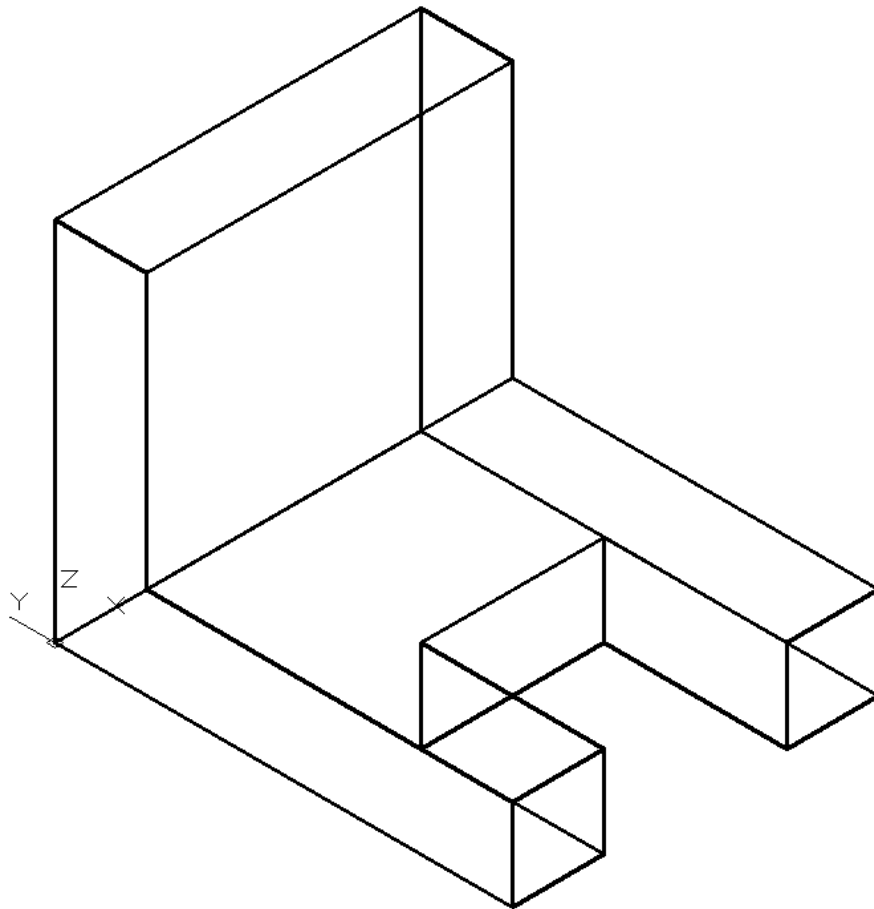


Figure 15.2 – Open Solids Problem 1

When you open the drawing, 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 use the `Tenview` LISP program to leave Model Space and generate the orthographic and isometric views.

Loading Tenview.LSP

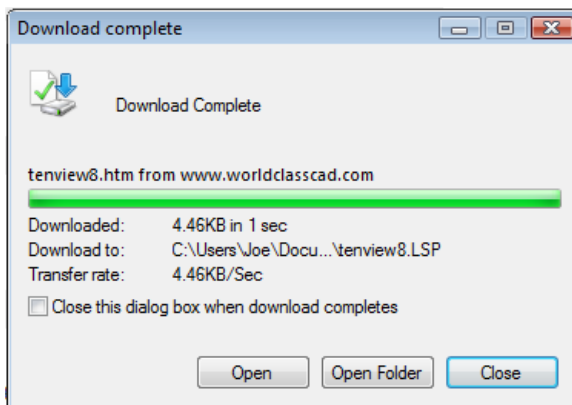
To obtain a copy of the Tenview LISP routine, visit the World Class CAD website at www.worldclasscad.com. From the World Class CAD home page, click on the Programming link to visit the Programming section of the site. Click on the link on the left side of the page that will take you to the Visual AutoLISP online textbook. As shown in Figure 15.3, you will see the phrase “Download LISP Routines” in the top center of the page. Select this link to download the LISP freeware. If you are having trouble finding the link, use this URL: http://www.worldclasscad.com/lisp_routines.htm



Figure 15.3 – Select “Download LISP Routines” on the Visual AutoLISP web page



Figure 15.4 – Select Tenview.LSP on the LISP Routines web page



Download the LISP routine by saving the Tenview file to a folder with your other 3D drawings. If your browser does not recognize the file type as a LISP application, save the file to a specific location and add the extension .LSP to the filename. The next step is to return back to AutoCAD, select the Manage tab, and click the Load Application tool on the Applications panel as shown in Figure 15.6 to open the Load / Unload Application window.

Figure 15.5 – Downloading Tenview.LSP

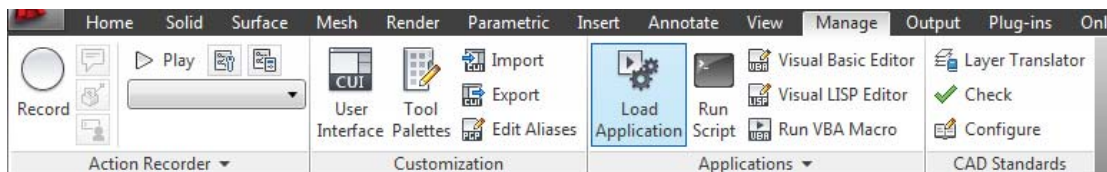
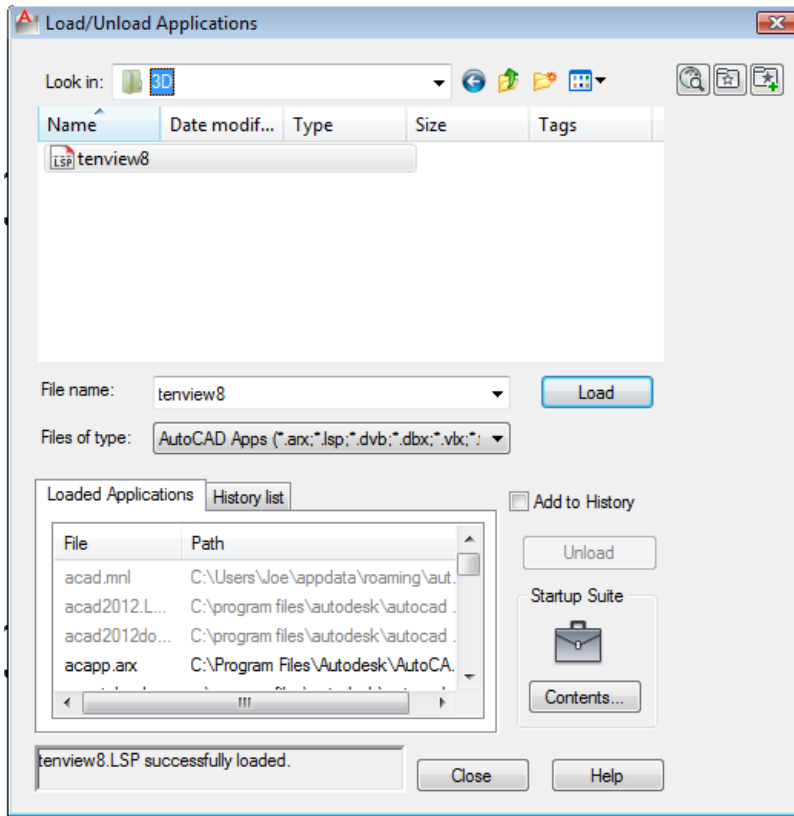


Figure 15.6 – Load Application



Find the folder where you saved the Tenview LISP file in the **Look in** list box. Select the program “Tenview8” and hit the Load button. At the bottom-left corner of the Load / Unload Applications window, you will see a small text display that was blank initially but now displays the text “Tenview8.LSP successfully loaded” (Figure 15.7). After making sure the program is loaded successfully, hit the Close button and an AutoCAD message window will appear in the middle of the screen stating: “Tenview.LSP – Copyright © 1998,1999,2001 by Charles Robbins. Type Tenview to start” (Figure 15.8).

Figure 15.7 – Select Tenview8.LSP to load the program in Solids Problem 1



Figure 15.8 – AutoCAD Message at the beginning of the Tenview program

Running the Tenview Program

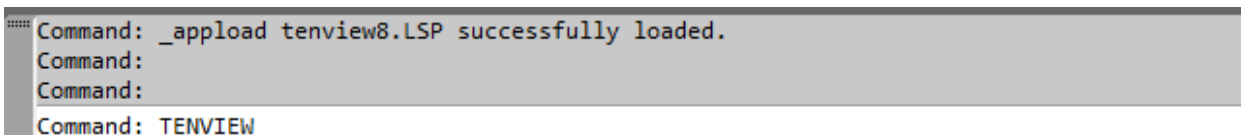


Figure 15.9 – Type Tenview at the command line

Hit the OK button on the AutoCAD message and follow its instructions by typing **Tenview** and ENTER at the command line as shown in Figure 15.9. The message “Pick your insertion point” appears on the command line so select a point at the lower left hand corner of the solid part as

shown in Figure 15.10. You may need to change the view of the part to the isometric view to select the correct point. The Tenview.LSP program will run in a few seconds, so you may not even notice the computer creating all of the six orthographic and four Isometric views. The program will end in Paperspace or Layout 1 as shown in Figure 15.11. The views are stored in the drawing's block library for future use. Now you need to make changes to Layout 1 to fit the B size border.

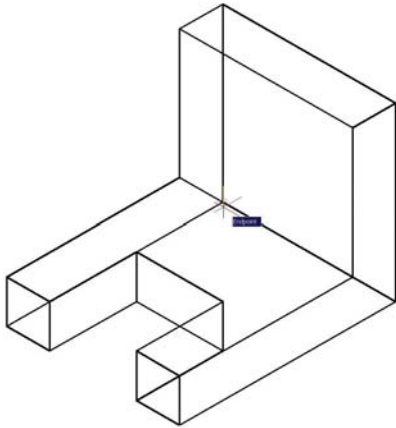


Figure 15.10 – Select insertion point

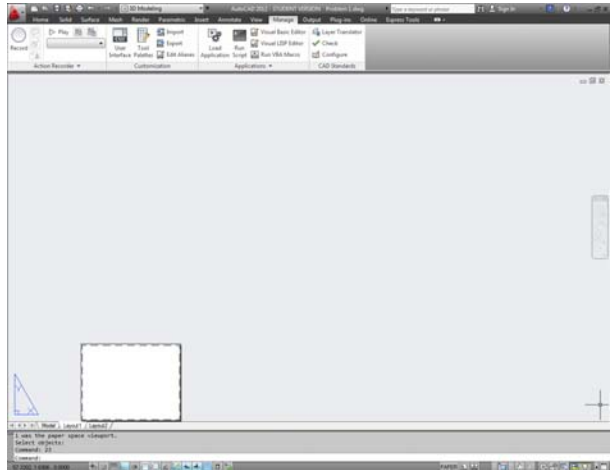


Figure 15.11 – Layout1 in Paperspace

Setting the Page Size

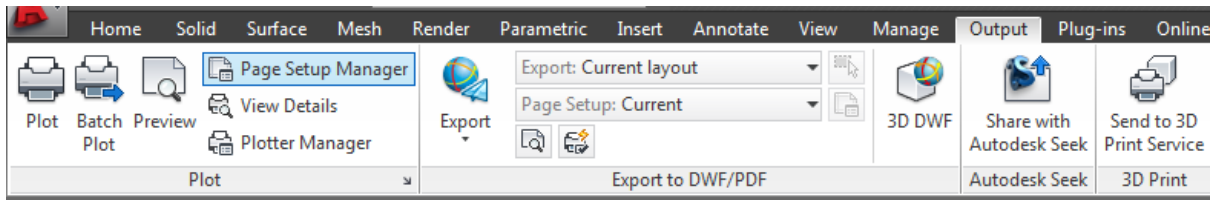


Figure 15.12 – Page Setup Manager, Output tab

Select the Output tab and pick the Page Setup Manager tool on the Plot panel as shown in Figure 15.12. 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 15.13. Hit OK and then Close the Page Setup Manager. Your layout should appear in Paperspace as shown in Figure 15.14.

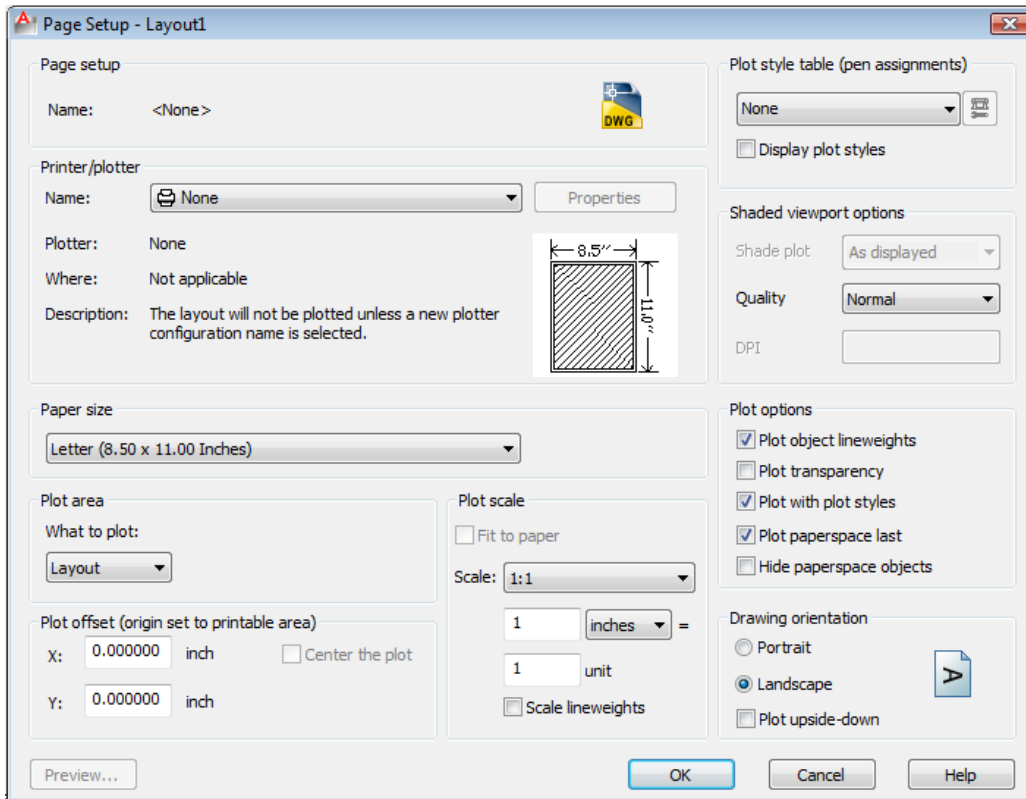


Figure 15.13 – Page Setup – Layout 1

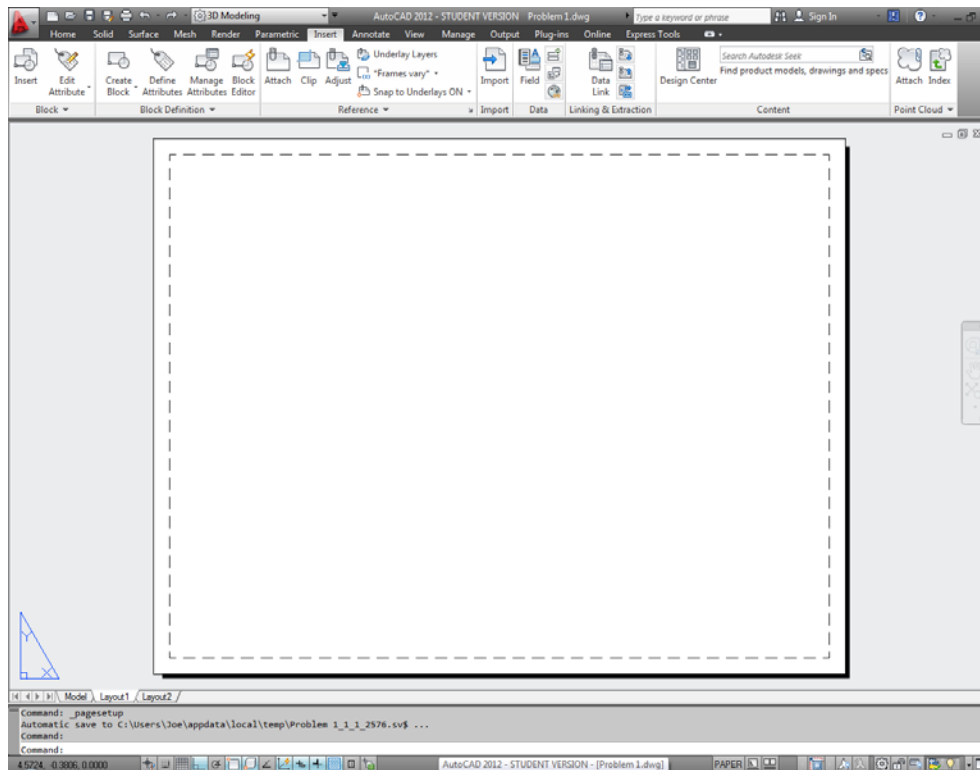
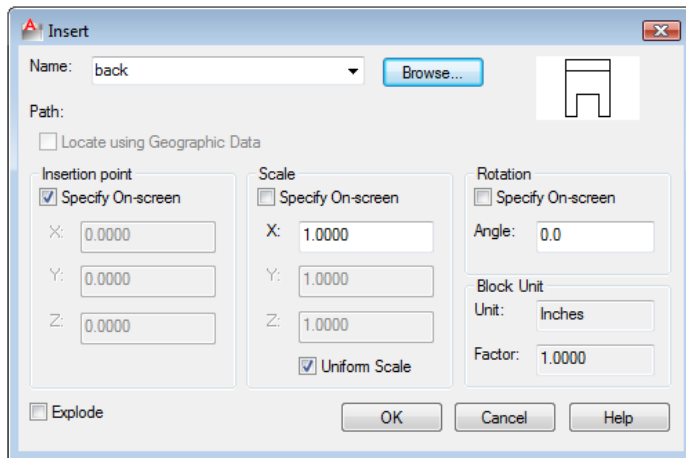


Figure 15.14 – A Size Sheet in Layout View

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 15.15) will appear. Use the Browse button to find the “a size border” in your Borders folder (Figure 15.16). Make sure the Specify On-screen box is checked in the Insertion Point area of the Insert window, and then hit OK (Figure 15.17).



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 15.18)

Figure 15.15 – Insert window

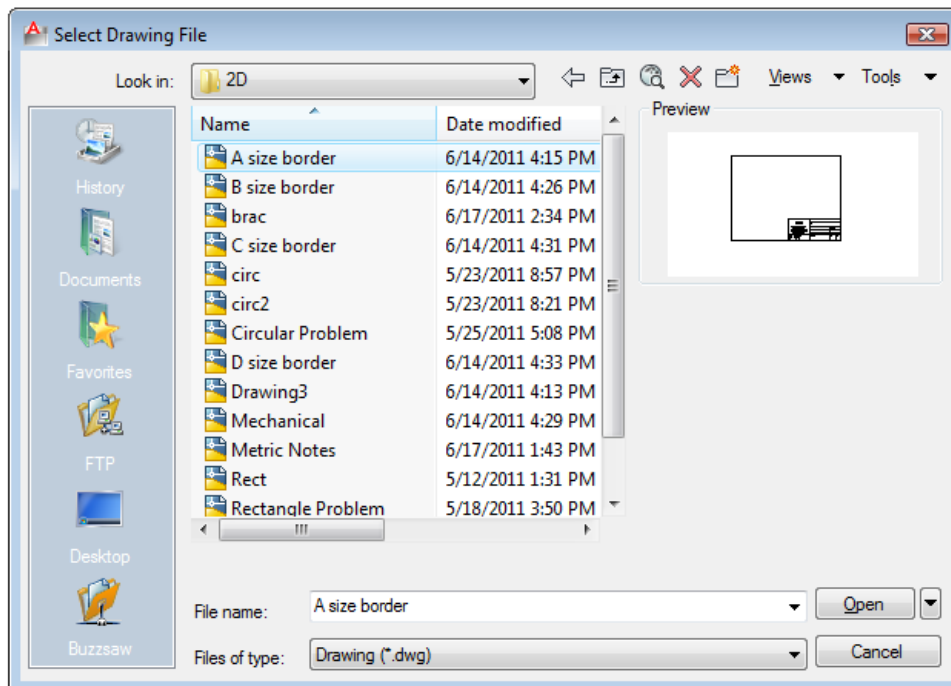


Figure 15.16 – Select Drawing File window

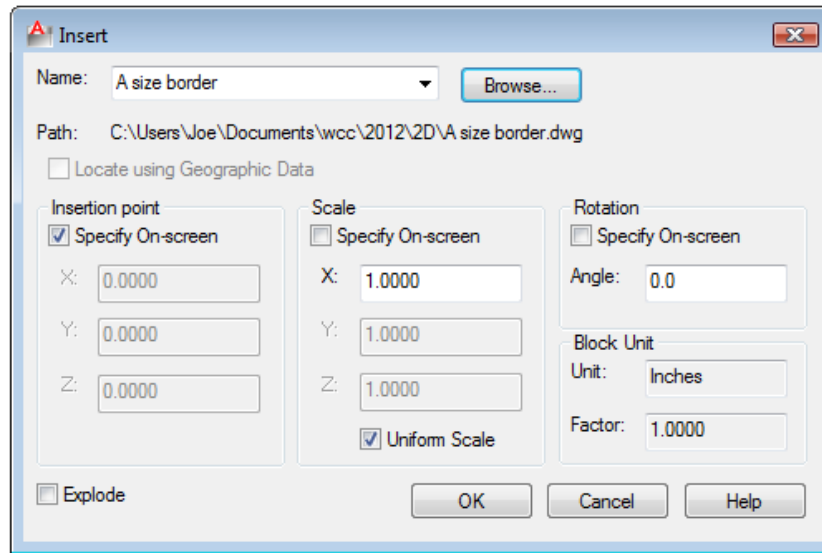


Figure 15.17 – A Size Border

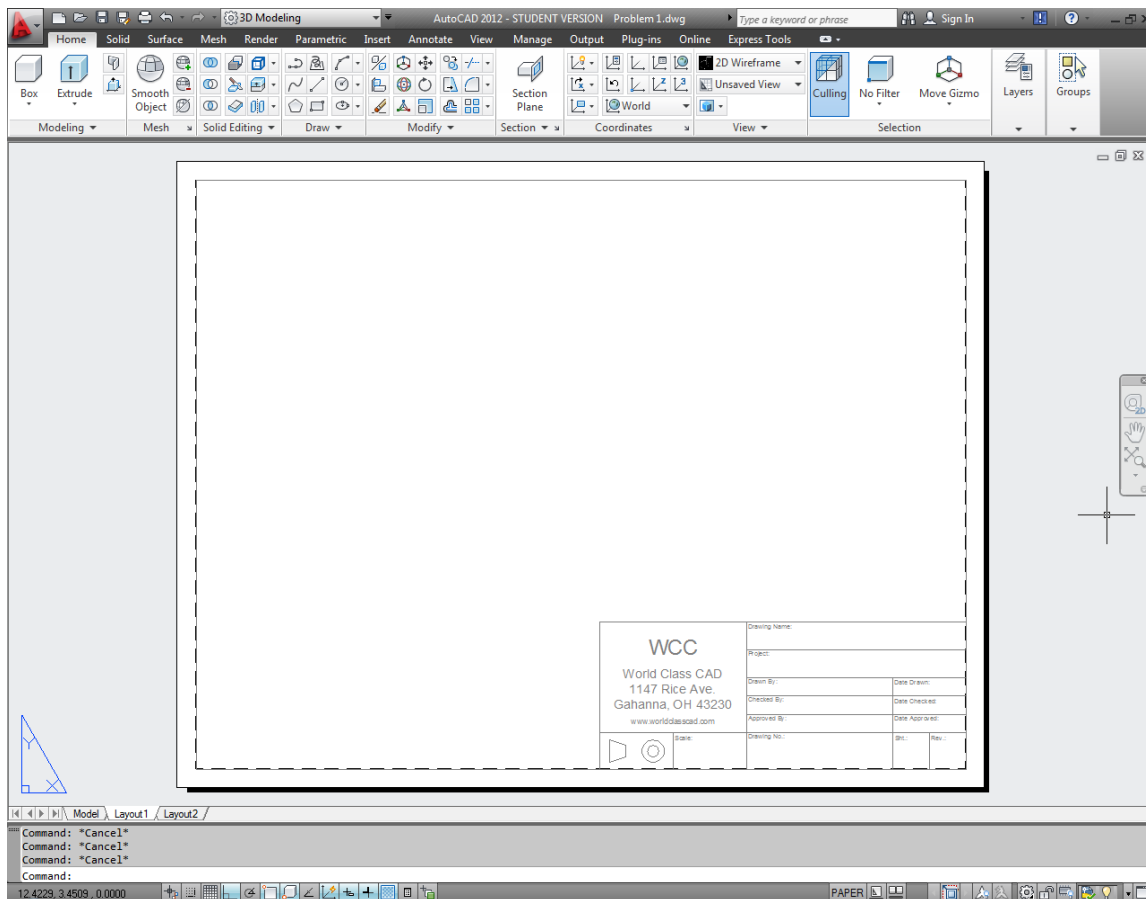


Figure 15.18 – A size border placed

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 15.18). Now you are ready insert the three orthographic views in this drawing.

Inserting the Orthographic Views and Moving Them into Position

Select the Insert tool on the Block panel of the Insert tab and the Insert window will appear (Figure 15.19). Select “front” from the Name list box. Keep all of the other settings the same and hit OK. Place the Front view in the bottom left corner of the border but not below the titleblock as shown in Figure 15.20.

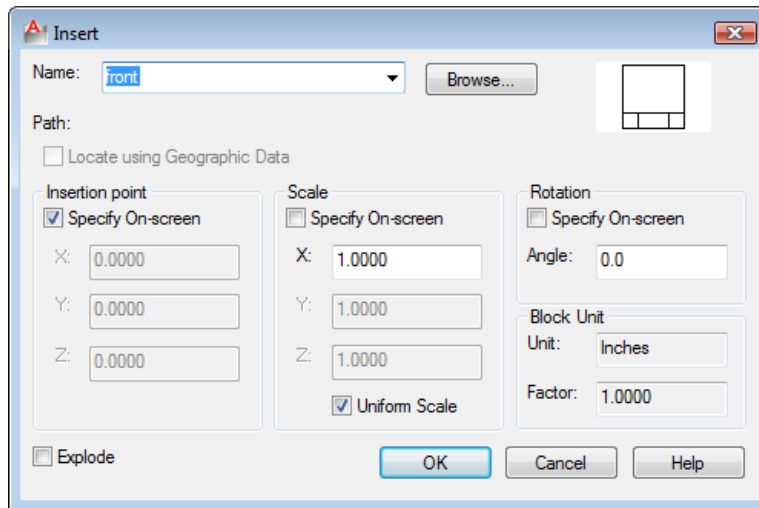


Figure 15.19 – Inserting the Front view

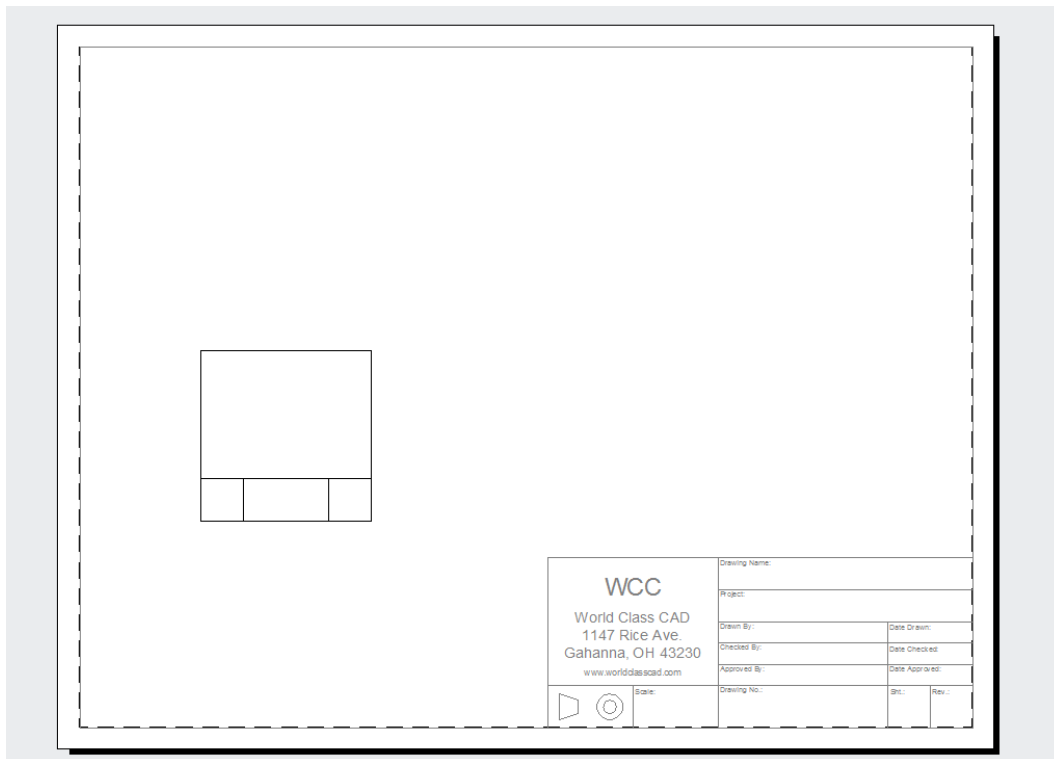


Figure 15.20 – The front view placed in the bottom left corner of the border

Select the Insert tool on the Block panel again to insert the Right side view. Select “right” from the Name list box on the Insert window and hit OK (Figure 15.21).

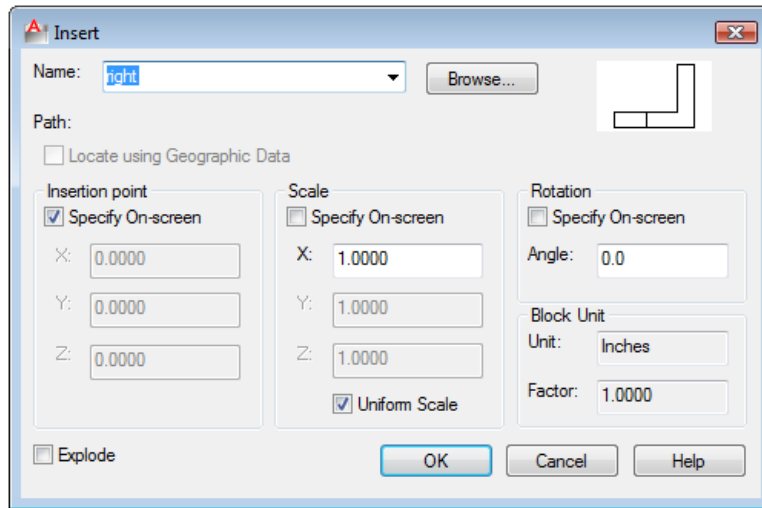
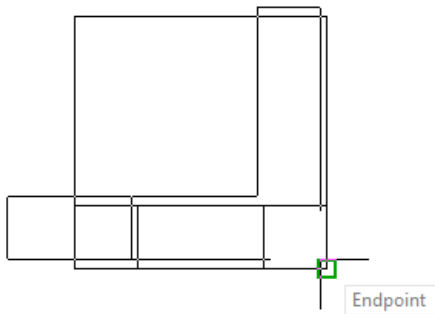


Figure 15.21 – Inserting the Right view



Place the right view block so that its top and bottom edge align with the front view by placing it on an endpoint Osnap point of the front view (Figure 15.22). Next, move the Right view to the right as shown in Figure 15.23. How do you know how far to separate the views? Mostly experience will help with this decision; keep enough space to place a dimension on the drawing without causing confusing between the two views.

Figure 15.22 – Placing the Right view

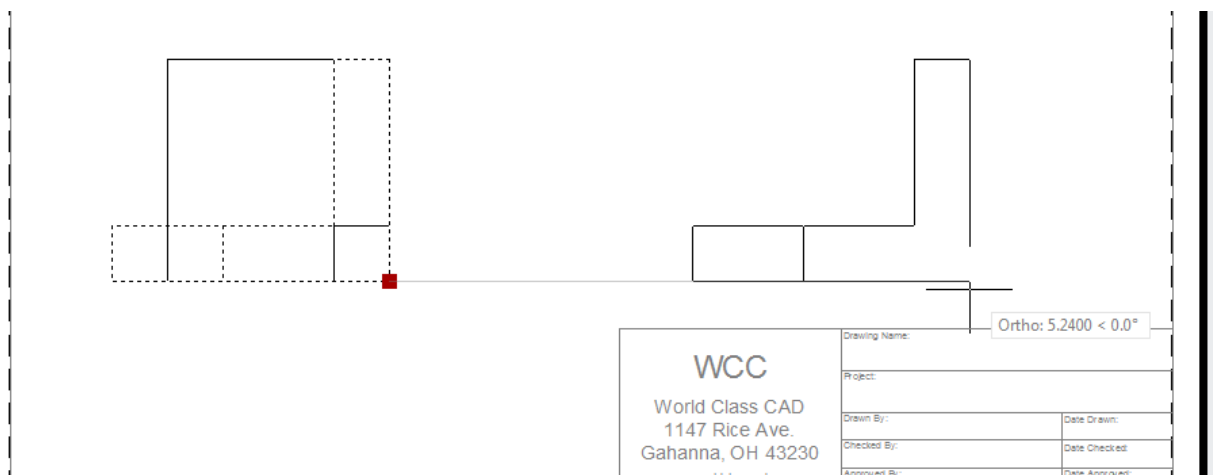


Figure 15.23 – Moving the Right view into position

Select Insert from the Block panel under the Inert tab once again to open the Insert window (Figure 15.24). Select the “top” view and hit OK to place the Top view.

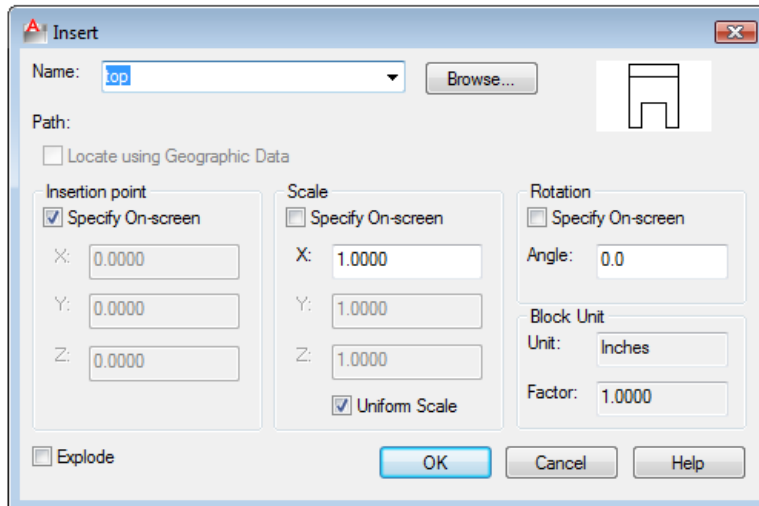


Figure 15.24 – Insert the Top view

Place the Top view block so that its left and right edge align with the front view by placing it on an endpoint Osnap point of the front view. Next, move the Top view above the front view as shown in Figure 15.25.

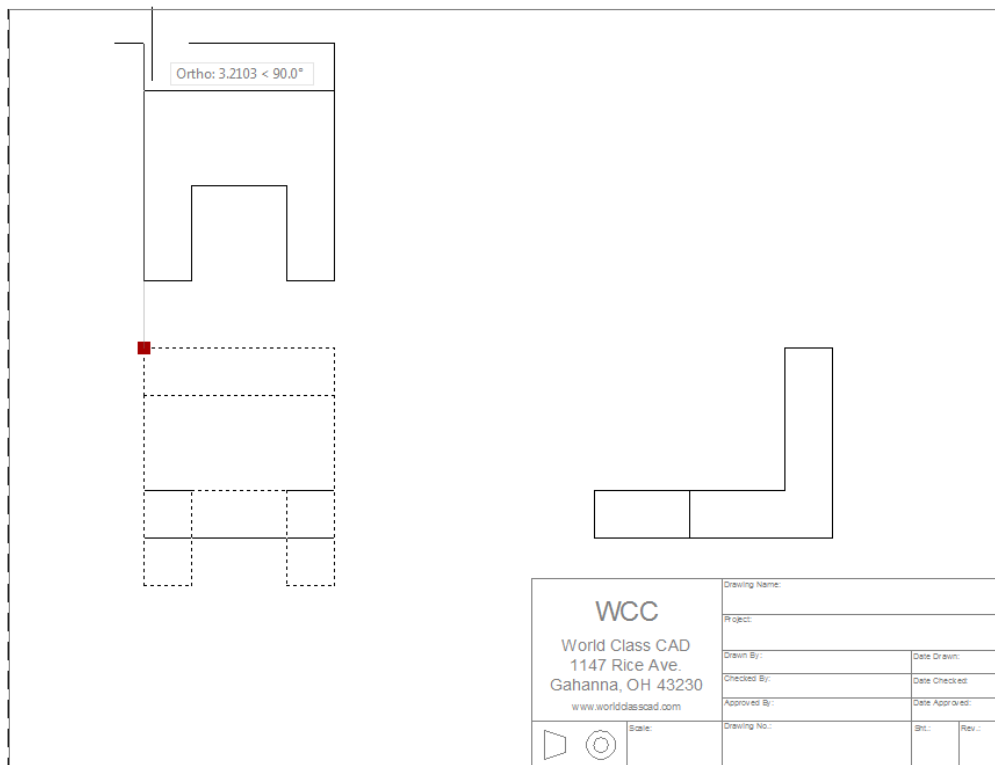
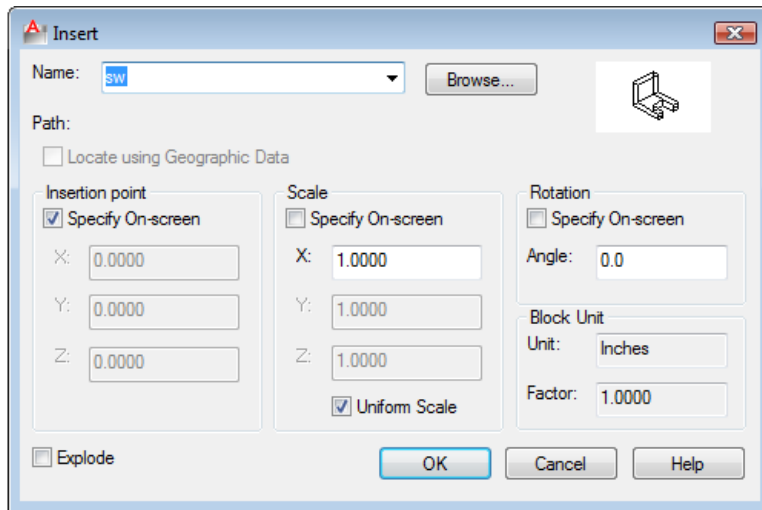


Figure 15.25 – Move the Top view

Inserting 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 Tenview, adding an isometric view is surprisingly easy.



You will place the southwest isometric view in the top right corner of the drawing. Select the Insert tool on the Block panel to insert the SW view. The Insert window (Figure 15.27) will appear in the graphical display. Select SW from the Name list box and hit OK.

Figure 15.26 – Inserting the southwest isometric view

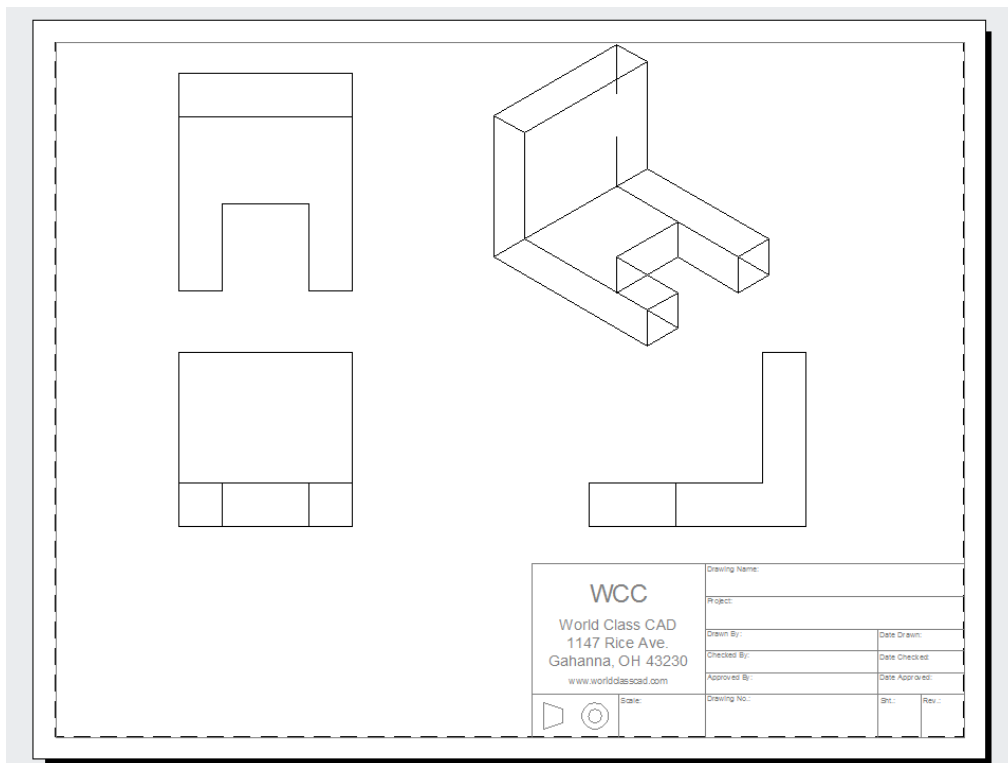
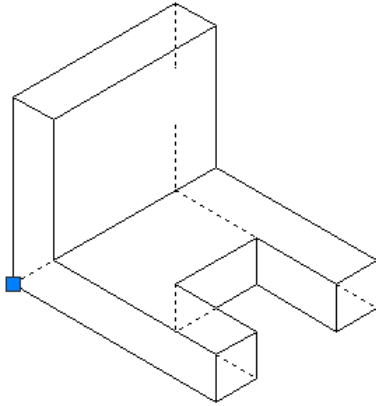


Figure 15.27 – The southwest view placed in the top right corner of the border



Pick the insertion point anywhere in the upper right corner of the drawing as shown in Figure 15.26. Use the Explode tool on the Modify panel (Figure 15.28) and remove the hidden tessellation lines with the Erase command. Now you can easily visualize the 3D solid.

Figure 15.28 – Remove the hidden lines

Dimensioning 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.

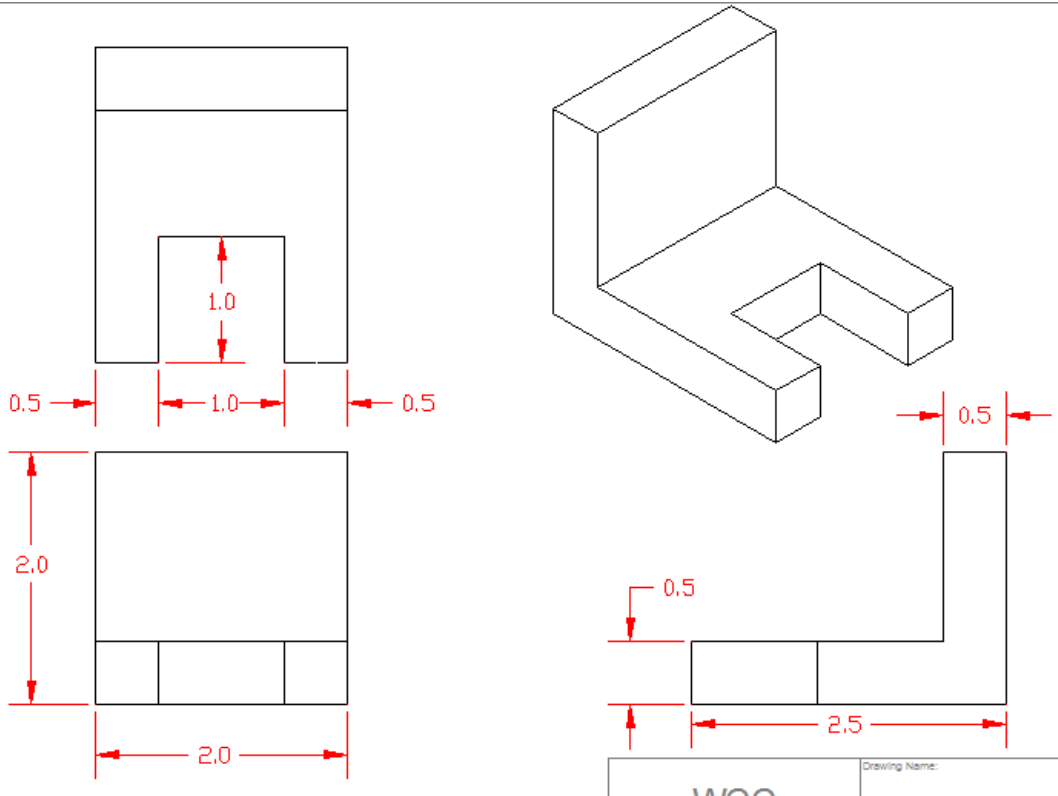


Figure 15.29 – Dimensioning the Solids Problem

In Figure 15.29, 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: 1:1	Drawing No.:	Sht.:	Rev.:
		2001	1	A

Figure 15.30 – Filling out the title block

Inserting Standard Notes into Paper Space

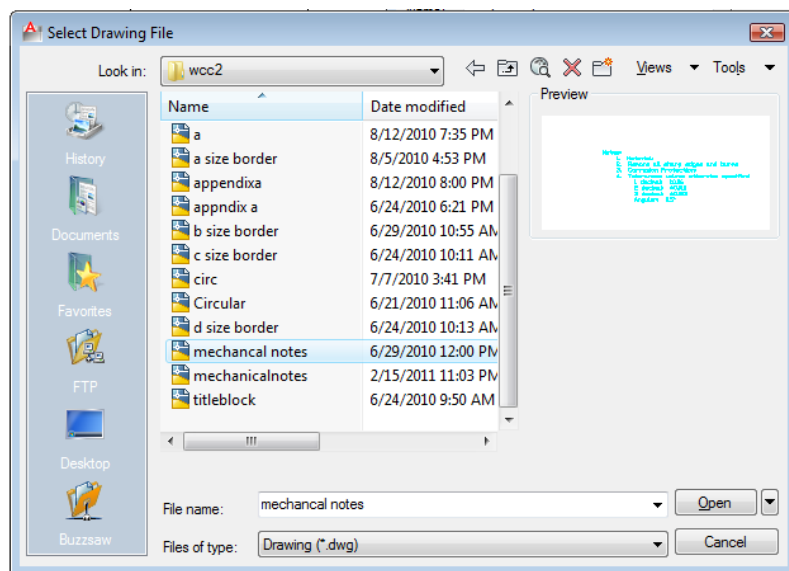


Figure 15.31 – Inserting the mechanical notes

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 15.31). Hit the Open button to proceed to the Insert window as shown in Figure 15.32.

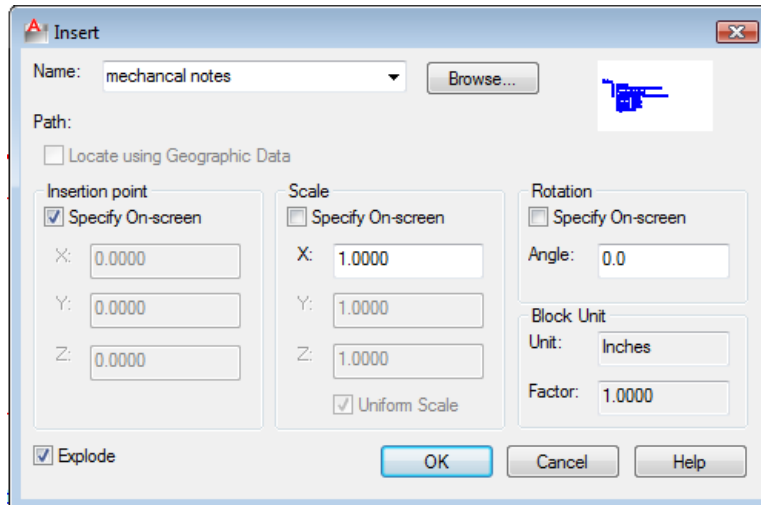


Figure 15.32 – Check Explode in the Insert Window

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 15.33).

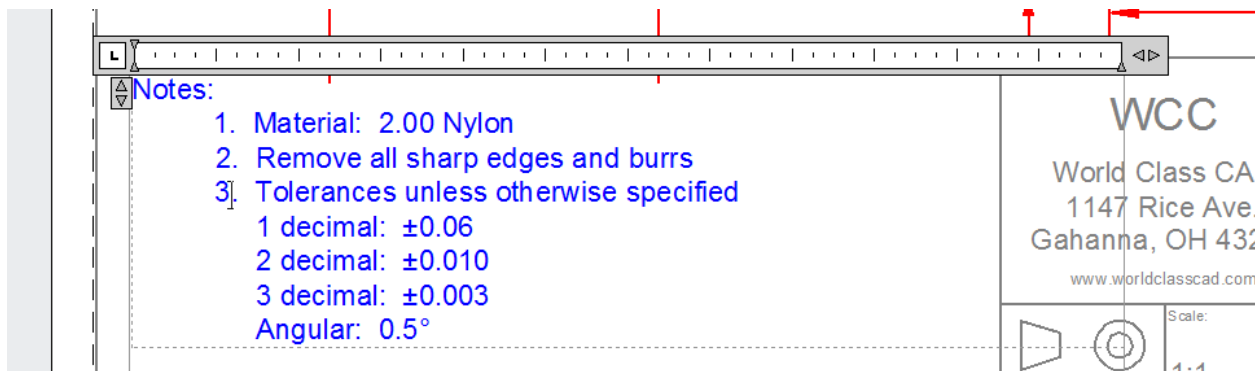


Figure 15.33 – Modifying the notes

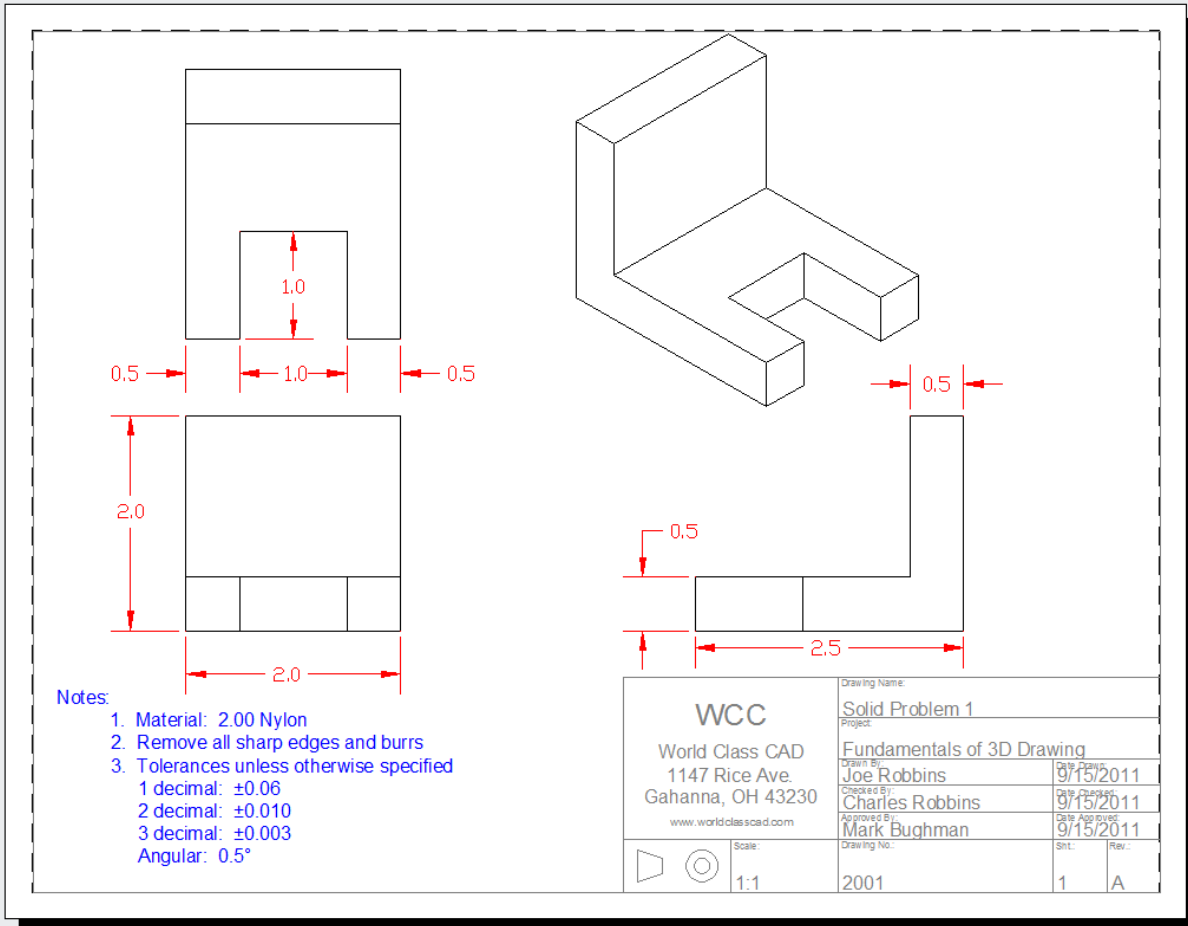


Figure 15.34 – The finished Solid Problem 1 drawing

* World Class CAD Challenge 03-15 * - Open Problems 1-10 and Revolved Solids 1-3 and create 2D drawings for each solid. Open the file, run Tenview.LSP, 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.