

Chapter 15

Tenviewing a Solid

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. Load Tenview.LSP**
- 4. Run the Tenview Program**
- 5. Setting the Page Size and Paper Space**
- 6. Insert the B Sized Border with Insert Block**
- 7. Insert the Orthographic Views and Move Them into Position**
- 8. Insert an Isometric View**
- 9. Dimension the Drawing**
- 10. Complete the Title Block**
- 11. Inserting a Standard Notes into the Paper Space Layout**
- 12. 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 a computer program.

In this chapter, we will learn the procedure for completing a multiview drawing using a LISP program, our dimensioning skills, a predefined drawing border and notes. The simple LISP routine repetitively uses the Solprof command to make 2D views in each direction, which is what a good software routine does. 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.

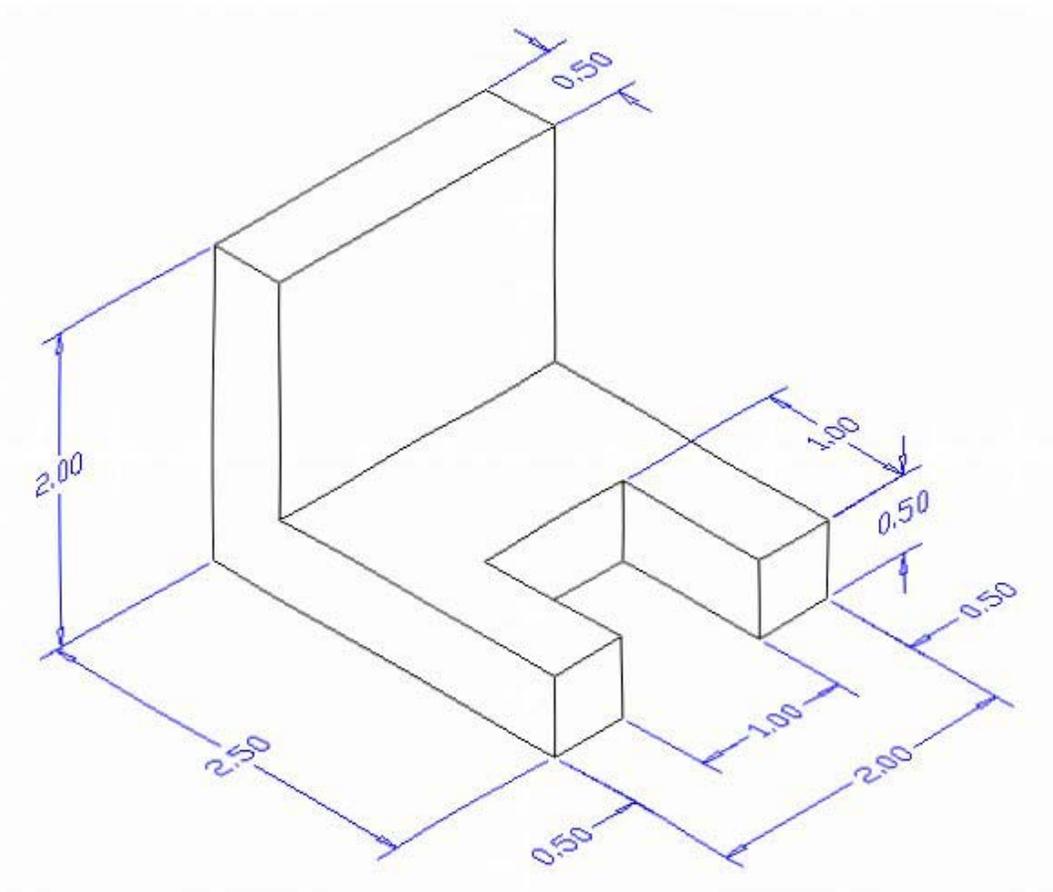


Figure 15.1 – Problem One Sketch

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 15.2. 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.

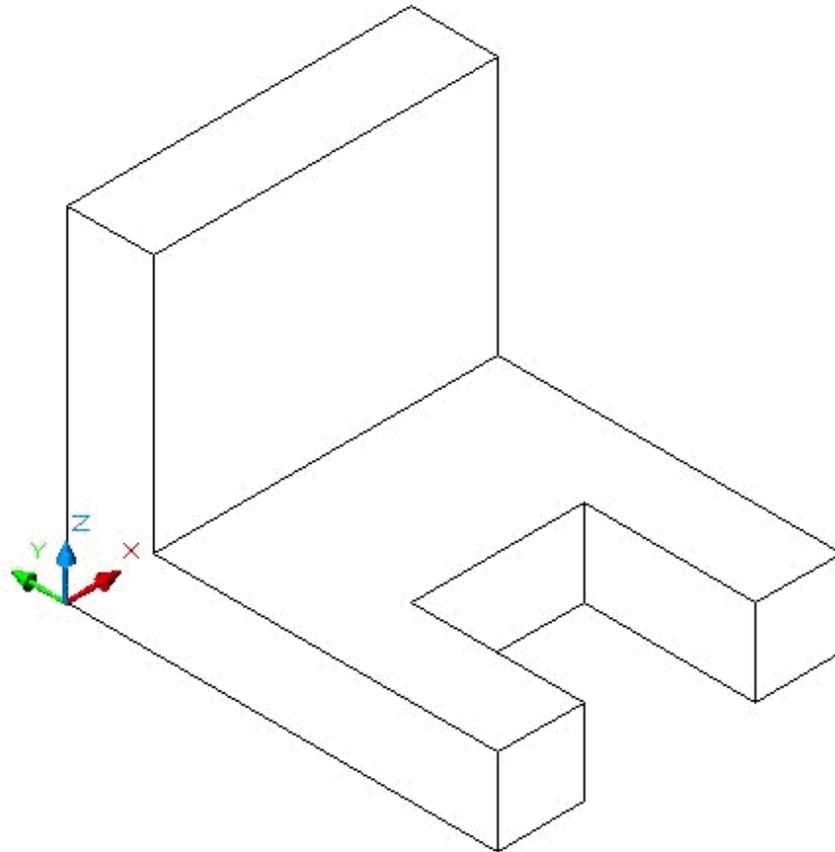


Figure 15.2 – Open Solids Problem 1

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 use the Tenview program to generate the orthographic and isometric views in seconds and leave Model Space.

Load Tenview.LSP

To obtain a copy of the LISP routine named Tenview, we need to visit the World Class CAD website at www.worldclasscad.com. At the World Class CAD home page, click on the Engineering Programming hyperlink which will take us to that web page. Then click on the link that will take us to the Visual AutoLISP online textbook. As shown in Figure 15.3, in the top center of the page, we will see the phrase “Download LISP Routines”. By selecting this link, we are able to view and download LISP freeware for our use.

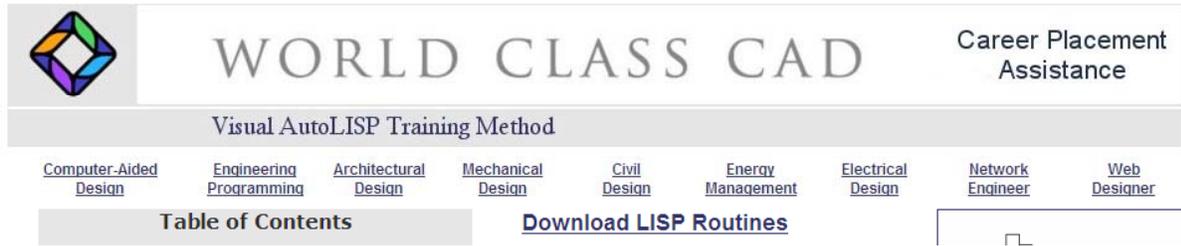


Figure 15.3 – Select Download LISP Routines on the Visual AutoLISP Web Page

Across the top of the page are letters of the alphabet that, once selected take us to programs that begin with that character. Select that letter T and we will view the Tenview hyperlink as shown in Figure 15.4.



Figure 15.4 – Select Tenview.LSP on the LISP Routines Web Page

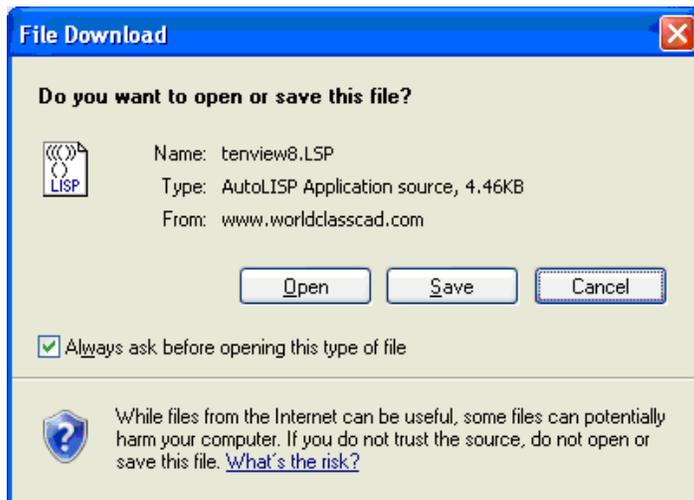


Figure 15.5 – Save the Tenview.LSP Program

Select Tenview.LSP and the File Download window appears. Push the Save command button and save the Tenview file to a folder on our desktop called “LISP Routines”. The next step is to return back to the AutoCAD program, select Tools, and Load Application as shown in Figure 15.6 to open the Load / Unload Application window.

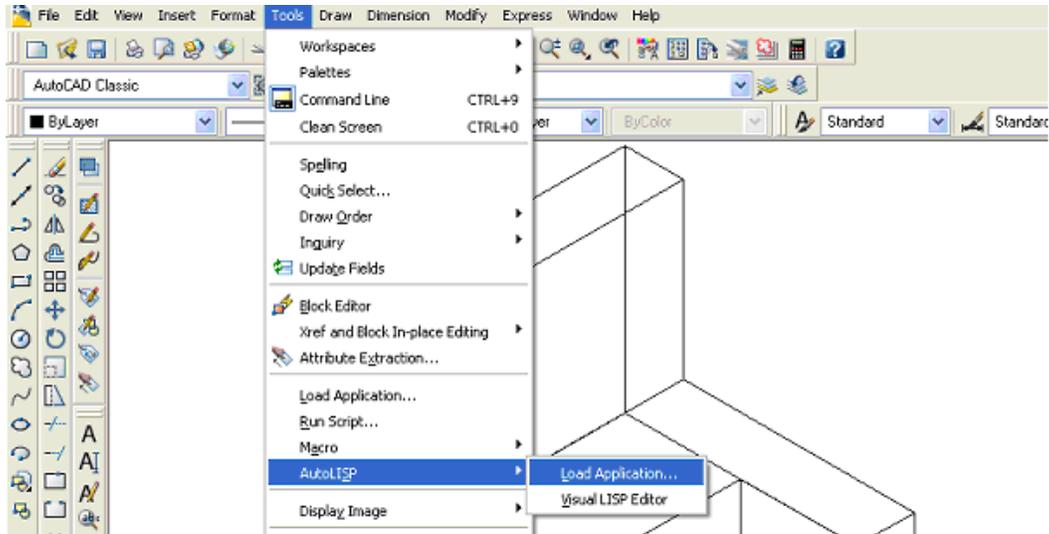


Figure 15.6 – Load Application

Make sure the **Look in** list box is displaying the LISP Routines folder and then select the program “Tenview8” and press the Load button. At the bottom – left corner of the Load / Unload Applications window, we will see a small text display that was blank initially but now displays the text, “Tenview8.LSP successfully loaded.” (Figure 15.7)

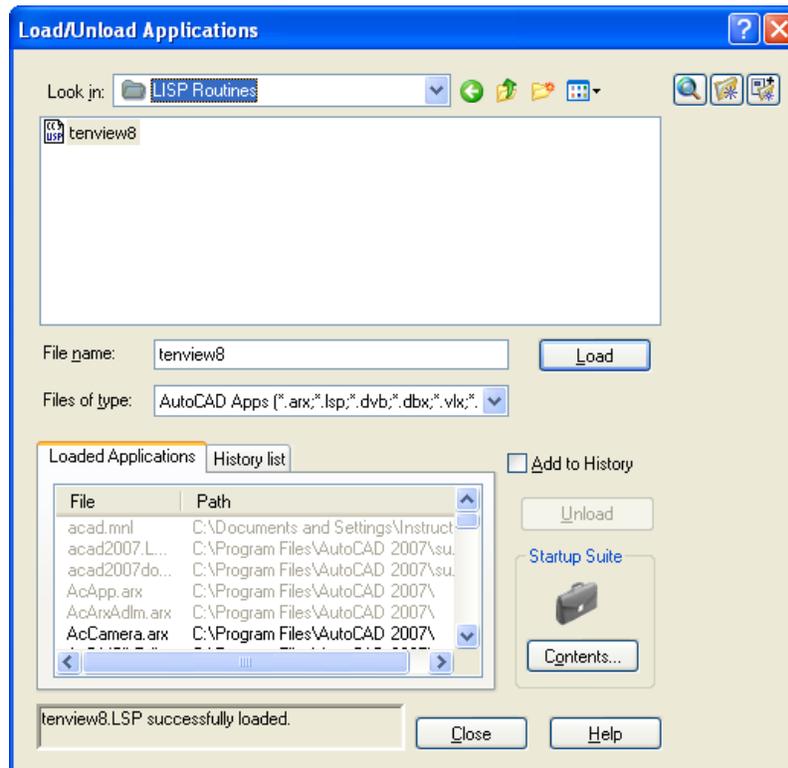


Figure 15.7 – Select Tenview8.LSP to Load to Solids Problem 1

After noting that the program is loaded, press the Close button and now an AutoCAD message window appears in the middle of the graphics display of the AutoCAD program stating:

“Tenview.LSP – Copyright © 1998,1999,2001 by Charles Robbins. Type Tenview to start”

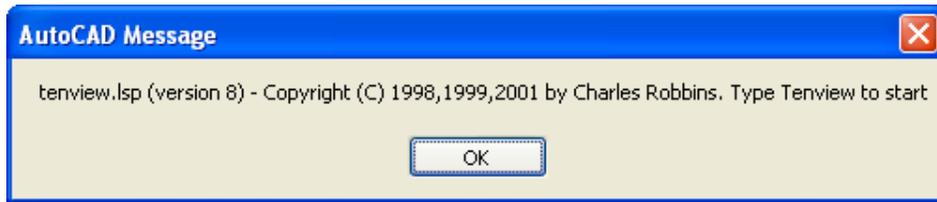


Figure 15.8 – AutoCAD Message at the Beginning of the Tenview Program

Run the Tenview Program

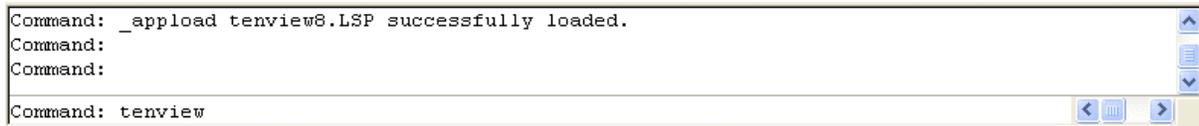
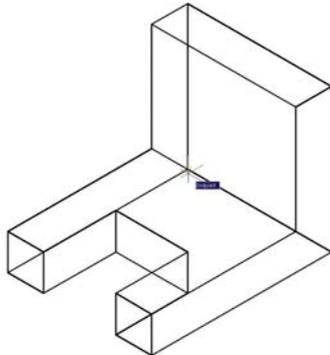
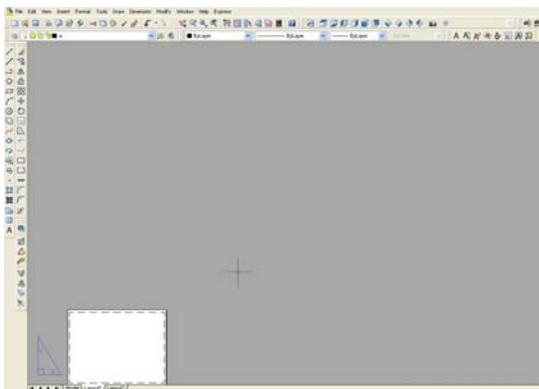


Figure 15.9 – Type Tenview at the Command Line



Press the OK button on the AutoCAD message if we agree with the information and follow the instructions by typing **Tenview** at the command line as shown in Figure 15.9. The message “Pick your insertion point” appears on the command line and the user should select a point at the lower left hand corner of the solid part as shown in Figure 15.10. If we select another point on the part, this is okay.

Figure 15.10 – Select Insertion Point



The Tenview.LSP program will run in a few seconds, so we 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. So now, we have to make changes to Layout 1 for the B sized border to fit.

Figure 15.11 – Layout1 in Paperspace
Setting the Page Size and Paper Space

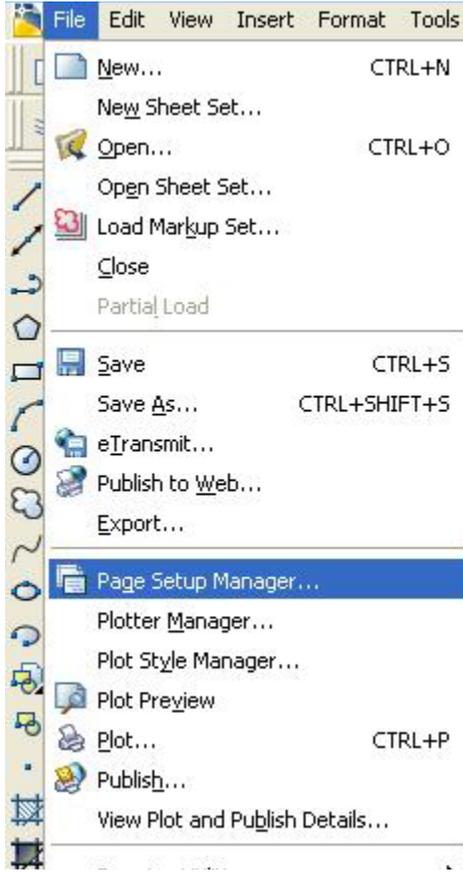


Figure 15.12 – File Manager

Select File and then Page Setup Manager as shown in Figure 15.12. Select Layout 1 and click Modify as shown in Figure 15.13 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 professional groups, there are black and white machines for check prints and color devices for finished drawings.

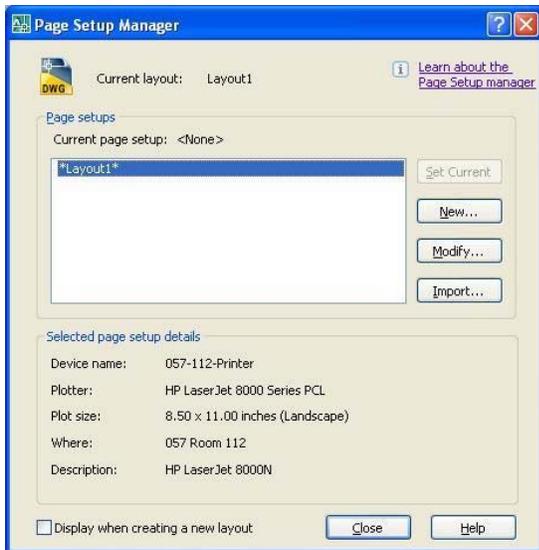


Figure 15.13 – Page Setup Manager

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.

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.

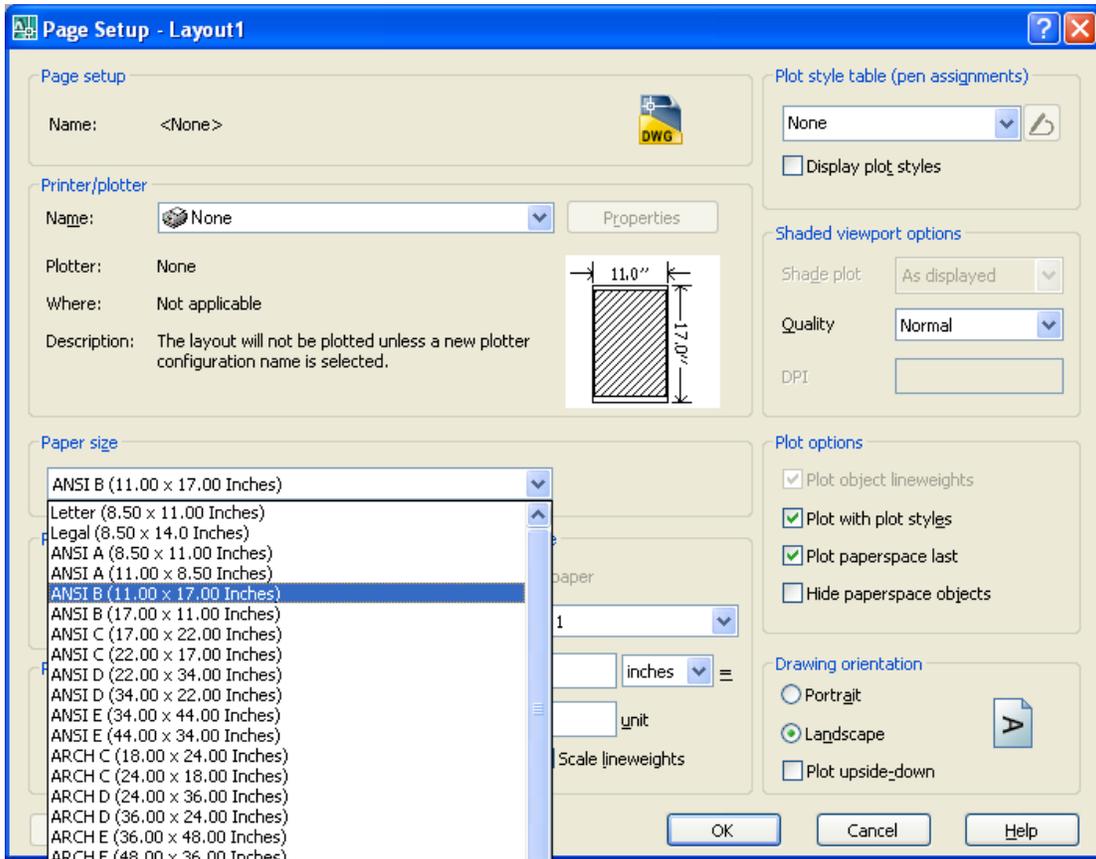


Figure 15.14 – Page Setup – Layout 1

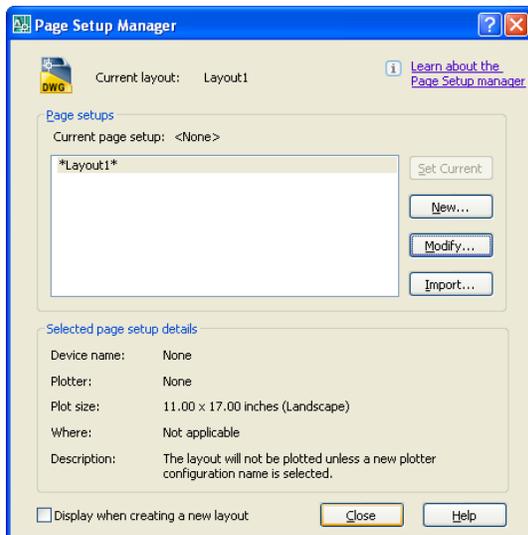


Figure 15.15 – Completed Page Setup Manager

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 15.16. The B-size drawing sheet is shown.



Figure 15.16 – B Size Sheet in Layout View

Insert the B Sized Border with Insert Block

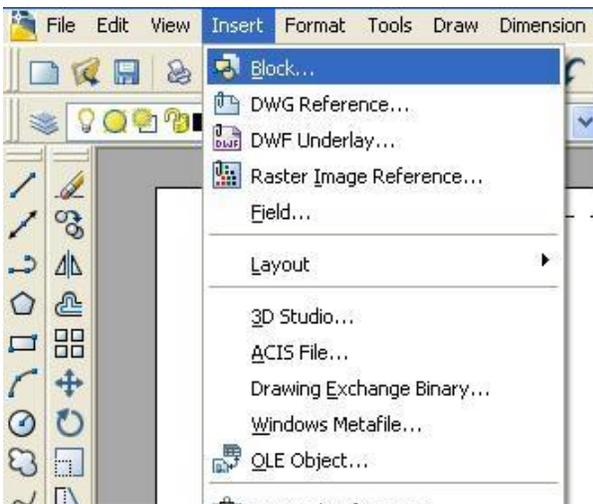


Figure 15.17 – Insert Block

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 15.17. The Insert window (Figure 15.18) 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 15.19). Keep the Specify On-screen checkbox unchecked and indicate that the Uniform Scale by checking the box (Figure 15.20).



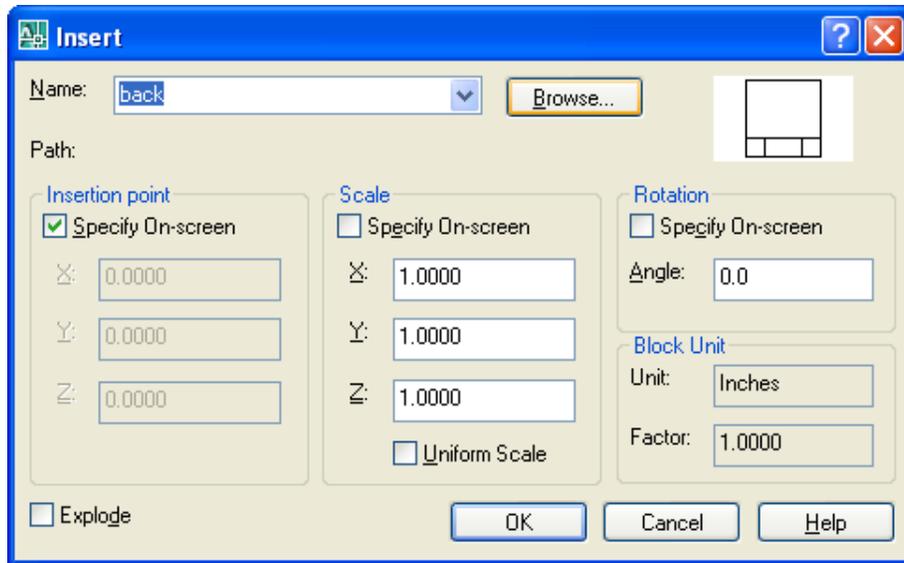


Figure 15.18 – Insert Window

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 15.21 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 15.21.

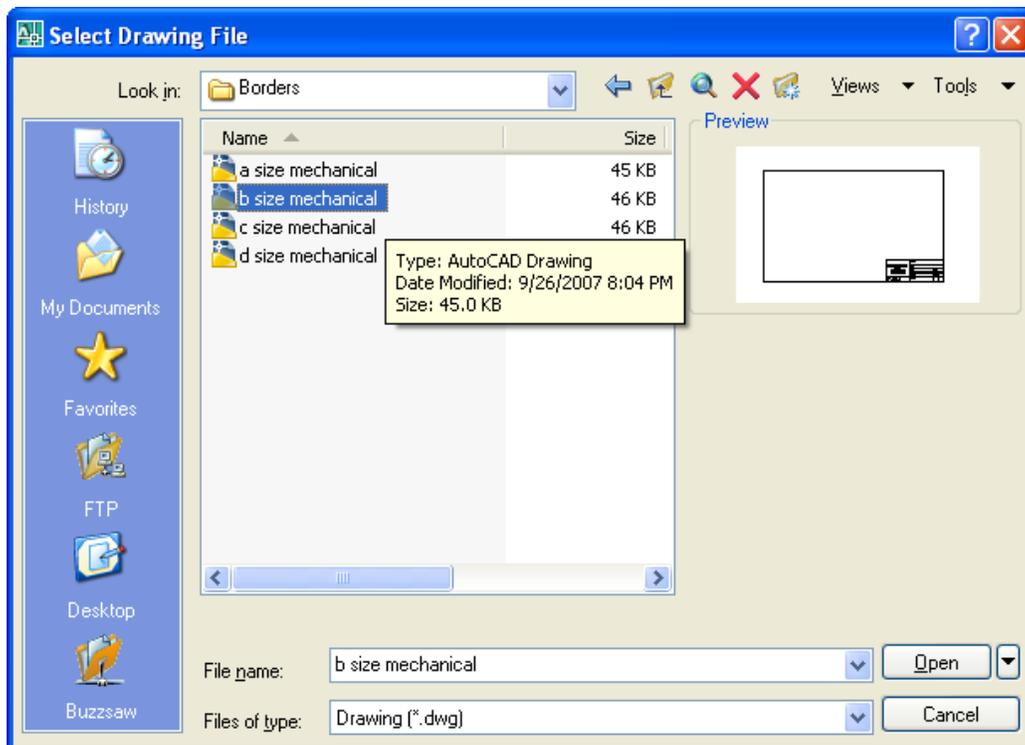


Figure 15.19 – Select Drawing File Window

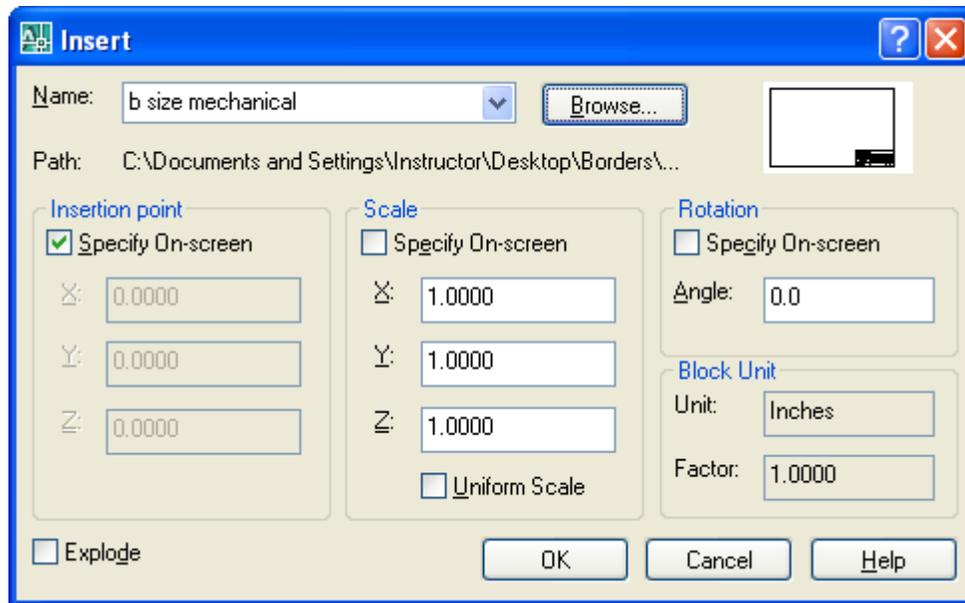


Figure 15.20 – B Size Mechanical

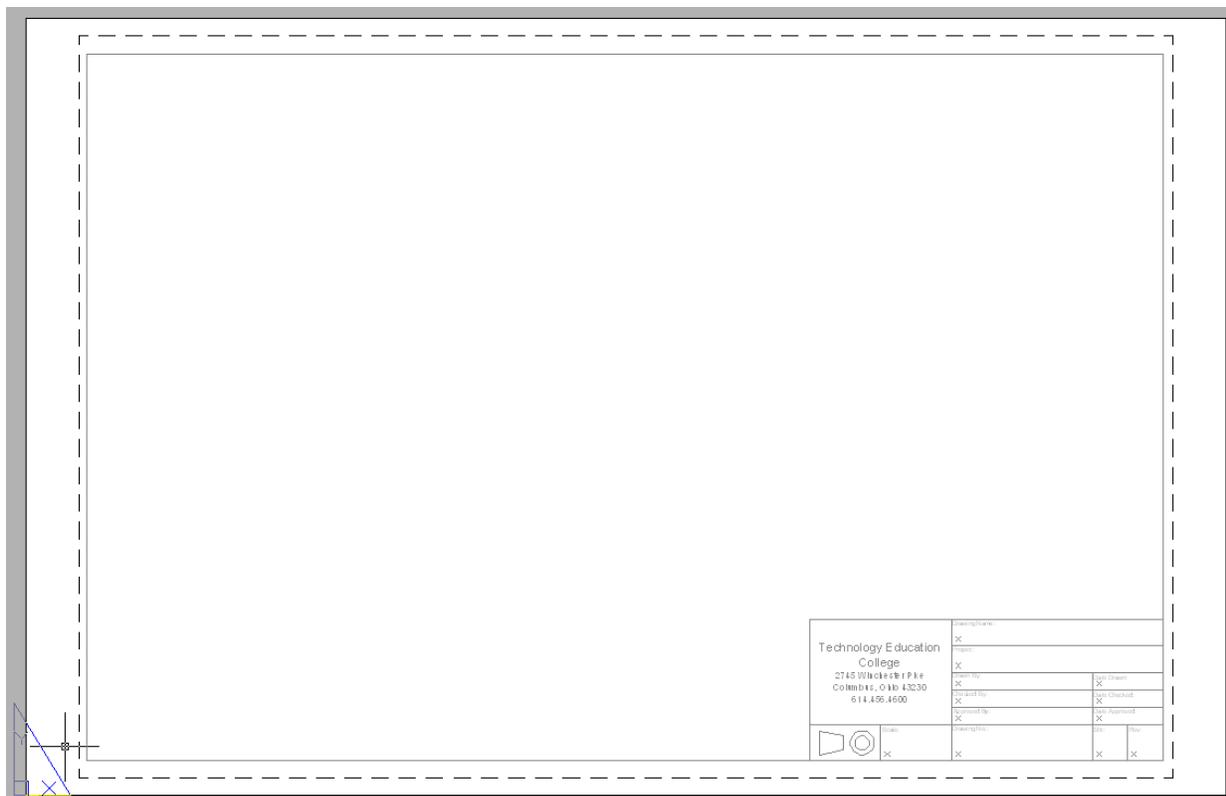


Figure 15.21 – B Size Mechanical in Layout1

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.

Insert the Orthographic Views and Move Them into Position

In the lower left hand corner of the drawing border, we will insert the Front view. Select the Insert tool on the Draw toolbar. The Insert window (Figure 15.22) will appear in the graphical display. This time, just select in the Name list box with the mouse to view the entire inventory of blocks in the local library. Keep the Specify On-screen checkbox unchecked and indicate that the Uniform Scale by checking the box.

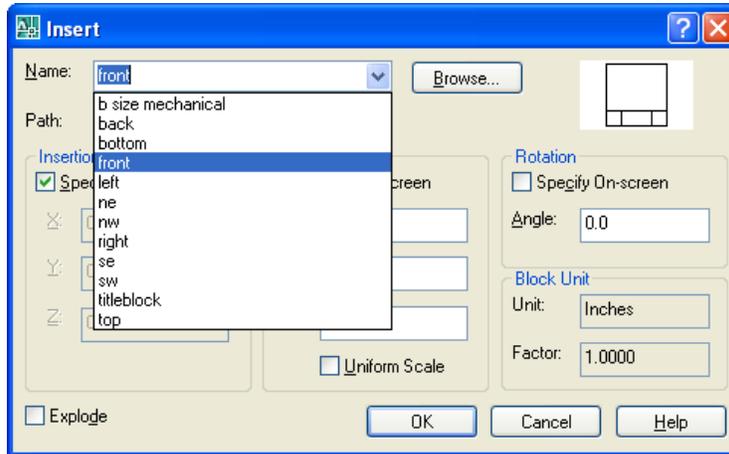


Figure 15.22 – B Size Mechanical in Layout1

After picking the OK button on the Insert window, pick the insertion point for the Front view as shown in Figure 15.23.

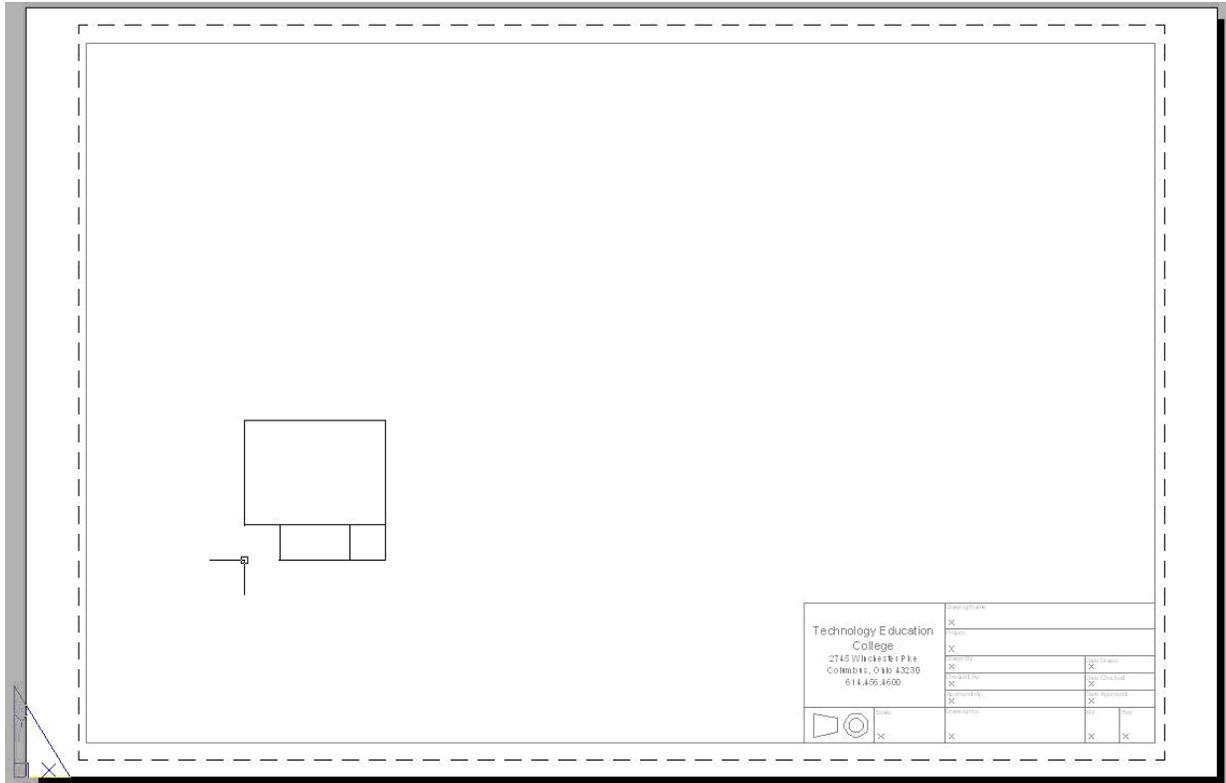


Figure 15.23 – Insert the Front View in Drawing

Now, select the Insert tool on the Draw toolbar to insert the Right side view. The Insert window (Figure 15.24) will appear in the graphical display. In the Name list box, we pick the Right view. Keep the Specify On-screen checkbox unchecked and indicate that the Uniform Scale by checking the box.

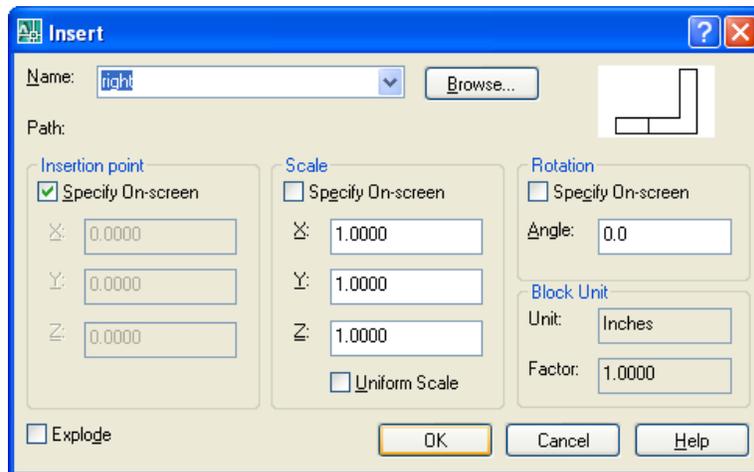
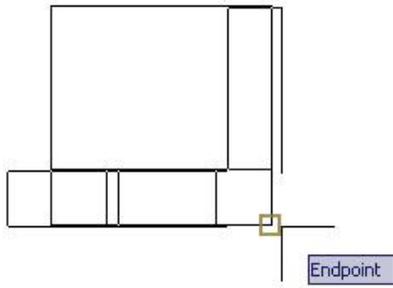


Figure 15.24 – Insert the Right View in Drawing



After picking the Right view, press the OK button on the Insert window, pick the insertion point for the Front view as shown in Figure 15.25. We choose a point on the base of the front view to align the orthographic views. Next, we move the Right view to its position as shown Figure 15.26. How do we know how far to separate the views? Mostly experience will help with this decision and keep enough space to place a dimension on the drawing without causing confusing to which view we are measuring.

Figure 15.25 – Insert the Right View

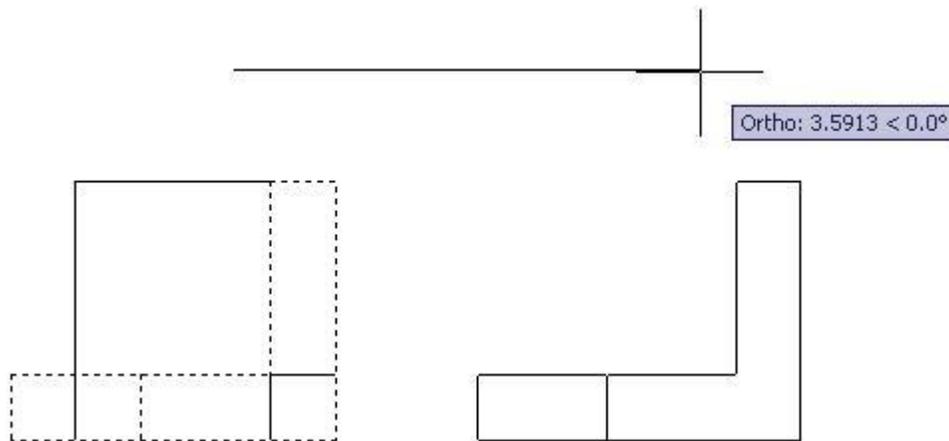


Figure 15.26 – Move the Right View into Position

At this moment, select the Insert tool on the Draw toolbar to insert the Top view. The Insert window (Figure 15.27) will appear in the graphical display. In the Name list box, we pick the Top view. Keep the Specify On-screen checkbox unchecked and indicate that the Uniform Scale by checking the box.

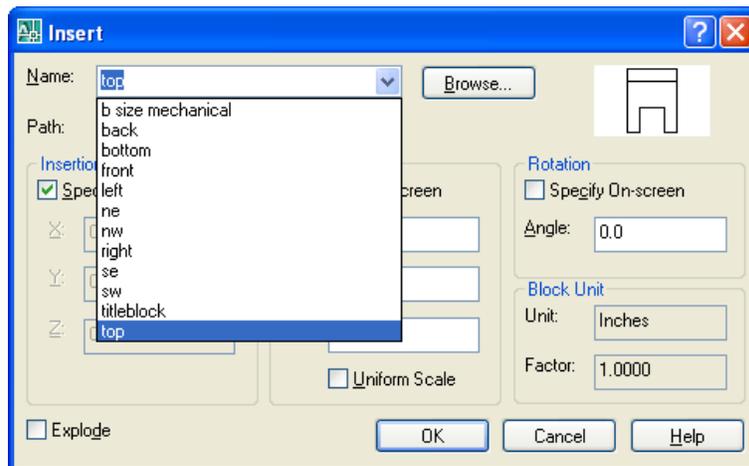
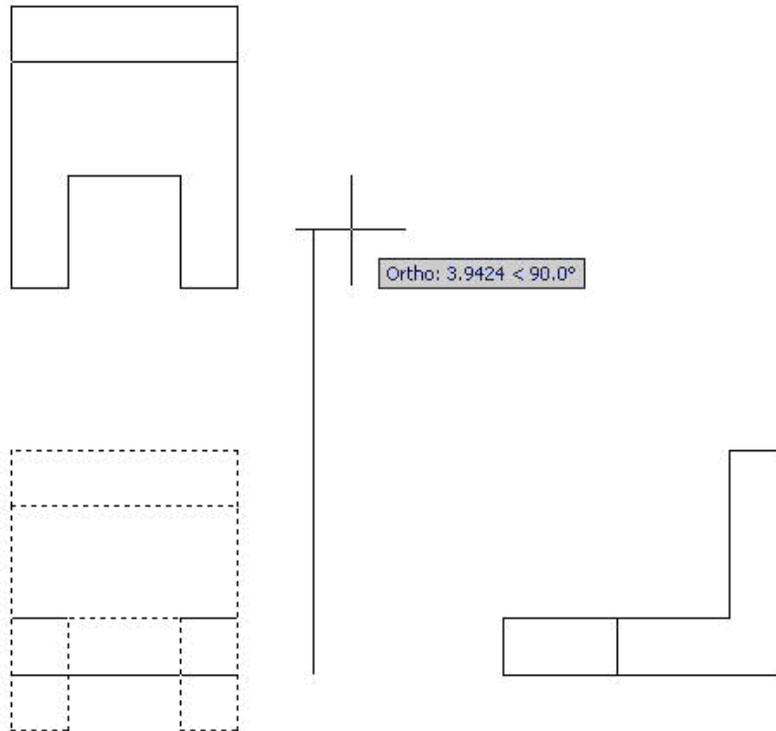


Figure 15.27 – Insert the Top View

After picking the Top view, press the OK button on the Insert window, pick the insertion point on the Front view. We choose a point on the base of the front view to align the orthographic views. Next, we move the Top view to its position as shown Figure 15.28.



**Figure 15.28 – Move the Top View
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 Tenview, this view is practically free. We will place the southwest isometric view in the upper right corner of the drawing.

Select the Insert tool on the Draw toolbar to insert the SW view. The Insert window (Figure 15.29) will appear in the graphical display. In the Name list box, we pick the SW view. Keep the Specify On-screen checkbox unchecked and indicate that the Uniform Scale by checking the box.

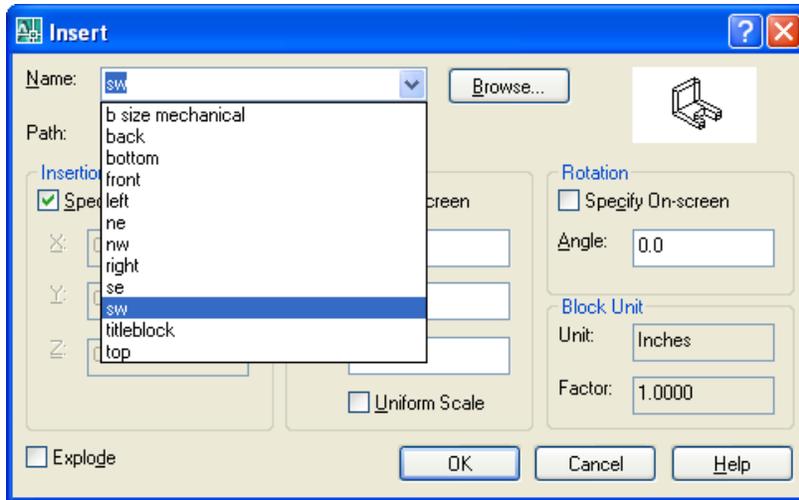


Figure 15.29 – Insert the Southwest Isometric View

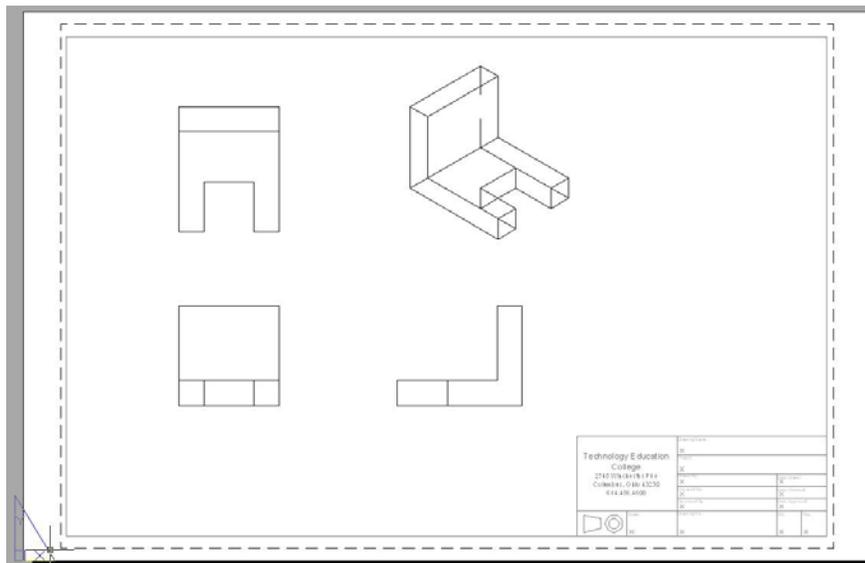
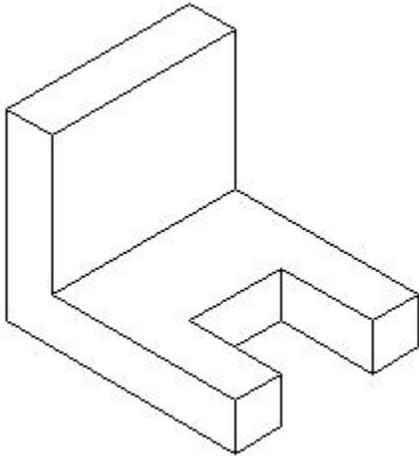


Figure 15.30 – Place the Southwest Isometric View into the Drawing



After picking the SW view, press the OK button on the Insert window, and pick the insertion point anywhere in the upper right corner of the drawing as shown in Figure 15.30. We will select the Explode tool on the Modify Toolbar and remove the hidden tessellation lines with the Erase command. Now we can easily view the 3D solid.

Figure 15.31 – Remove the Hidden Lines

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.

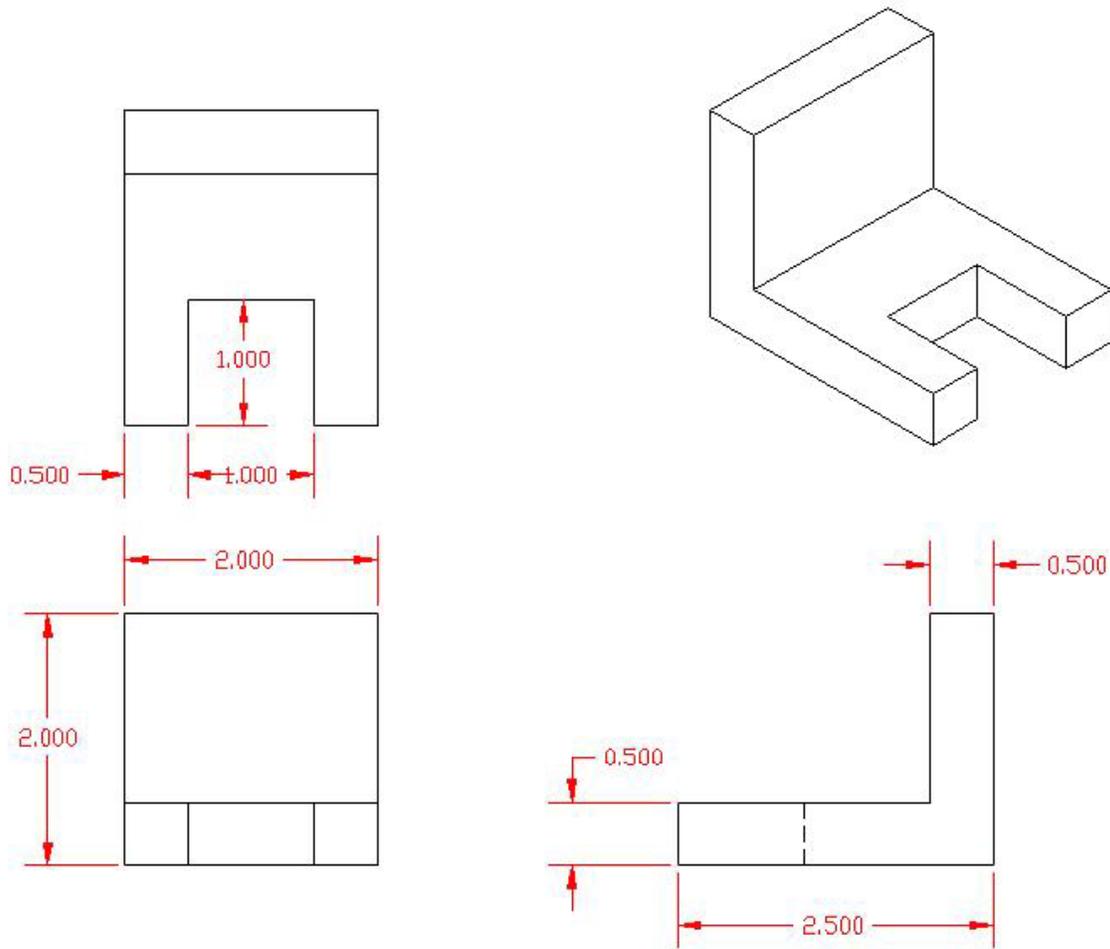


Figure 15.32 – Dimensioning the Solids Problem

In Figure 15.32, we see the dimensions in position that will satisfy a project manager.

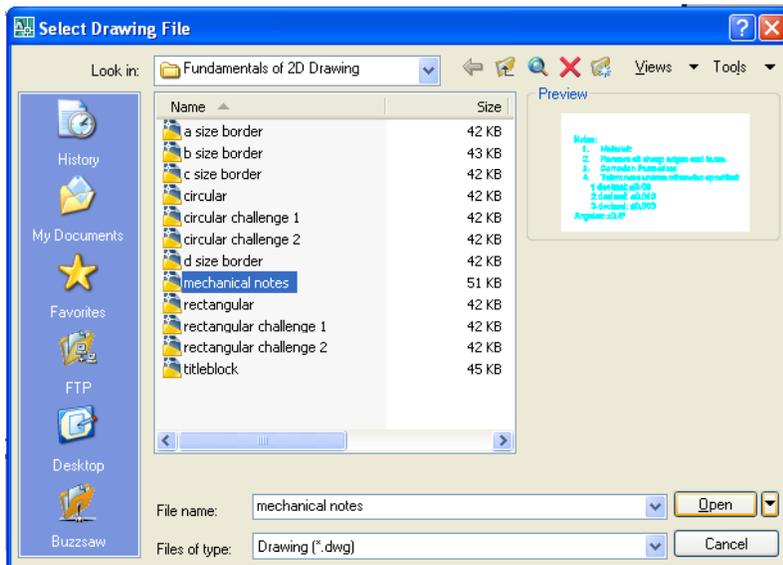
Complete the Title Block

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.

 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 15.33 – 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 15.34. 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 15.34 – Inserting the Metric Mechanical Notes

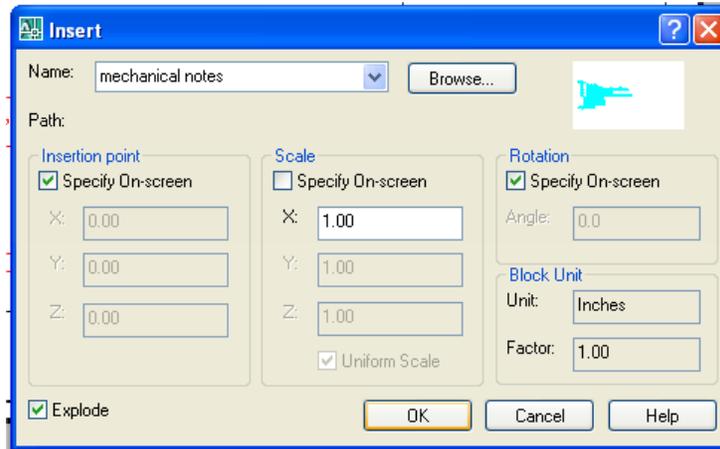


Figure 15.35 – Check Explode in the Insert Window

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 15.36)

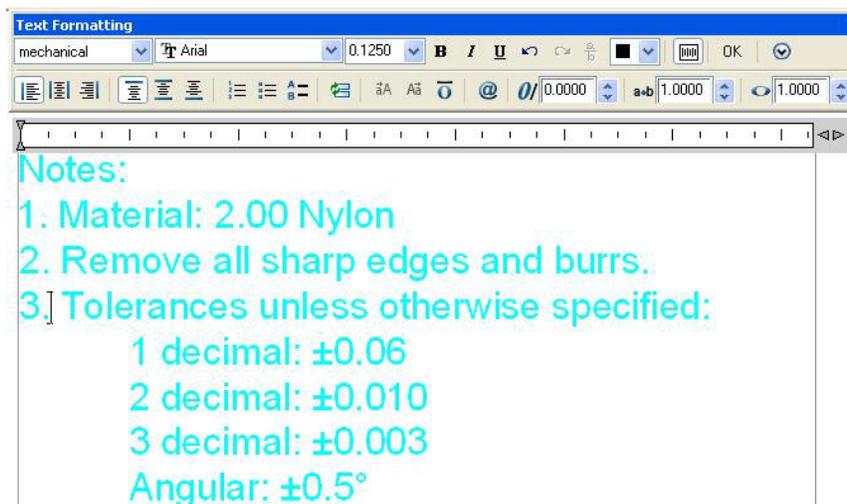


Figure 15.36 – Modifying the Notes by Adding Material and Corrosion Details

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 15.37 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.

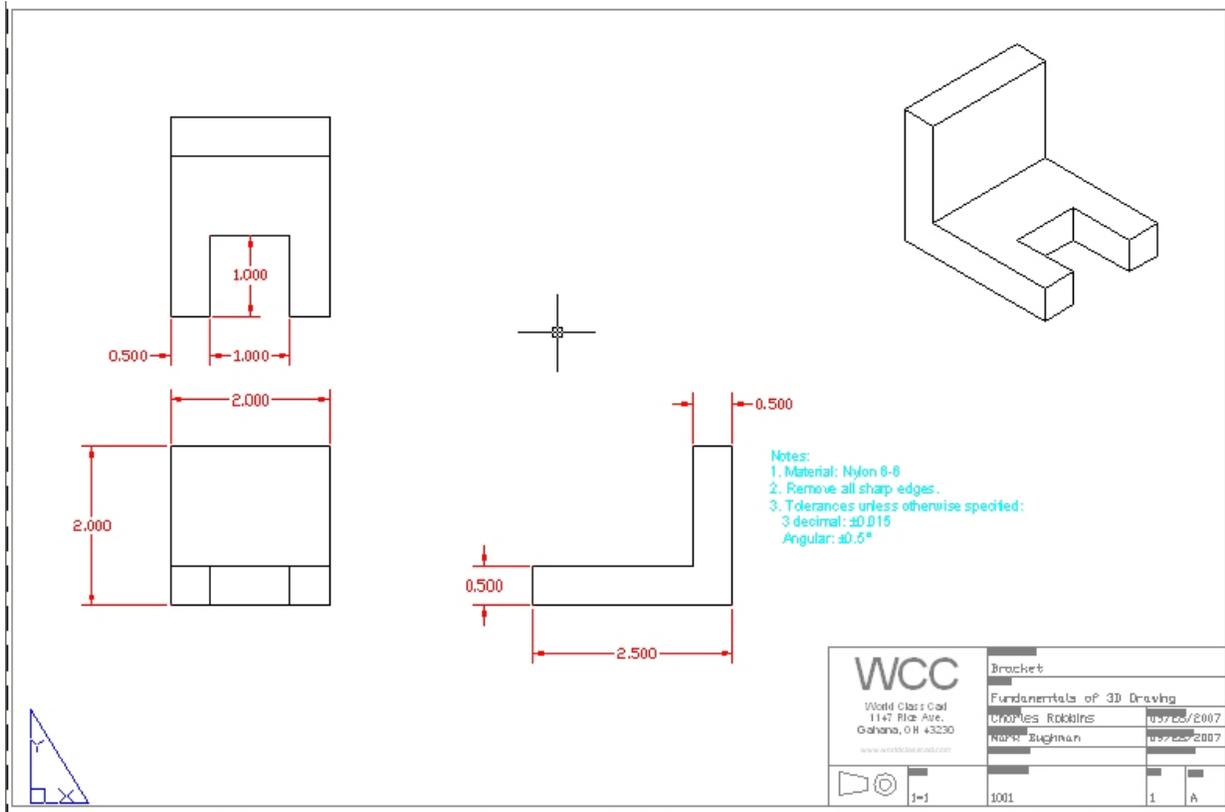


Figure 15.37 – The Finished Solids Problem 1 Drawing

*** 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, 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.**