

Solid Problem Seven

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

- 1. Sketch of Solid Problem Seven**
- 2. Starting a 3D Part Drawing**
- 3. Modifying How the UCS Icon is Displayed**
- 4. Drawing a Closed Polyline Shape to be Extruded**
- 5. Adding Circles to the Detail**
- 6. Moving the Circle Inside the Polyline**
- 7. Extruding Solids from a Closed Polyline and Circle**
- 8. Subtracting a 3D Solid from Another Solid**
- 9. Rotating a 3D Solid Using the Rotate3D Command**
- 10. Drawing a Solid Box**
- 11. Moving a 3D Solid**
- 12. Constructing a Solid Perimeter Using the Line Command**
- 13. Creating Polylines Using the Edit Polyline Tool**
- 14. Extruding the Solid from a Closed Polyline**
- 15. Aligning a Solid to the Master Solid**
- 16. Subtracting 3D Solids from the Master Solid**
- 17. Hide the Solid**
- 18. Moving Solid to the Origin Point**
- 19. Saving the Solid Problem**

Sketch of Solid Problem Seven

You will start the seventh solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across one line, which you can call the x-axis. In this exercise, a strategy you will continue to use is to draw closed Polylines that contains the most complex shape of the component and then use simple geometric solids to subtract detail. In problem seven, the master solid contains a chamfer drawn into the Polyline shape. You will place a circle off the edge of the main shape. After extruding the Polyline and Circular extruded solids, subtract the cylinder. You will finish by subtracting a Box and a Wedge shape from the master solid to end with the product as shown in Figure 8.1.

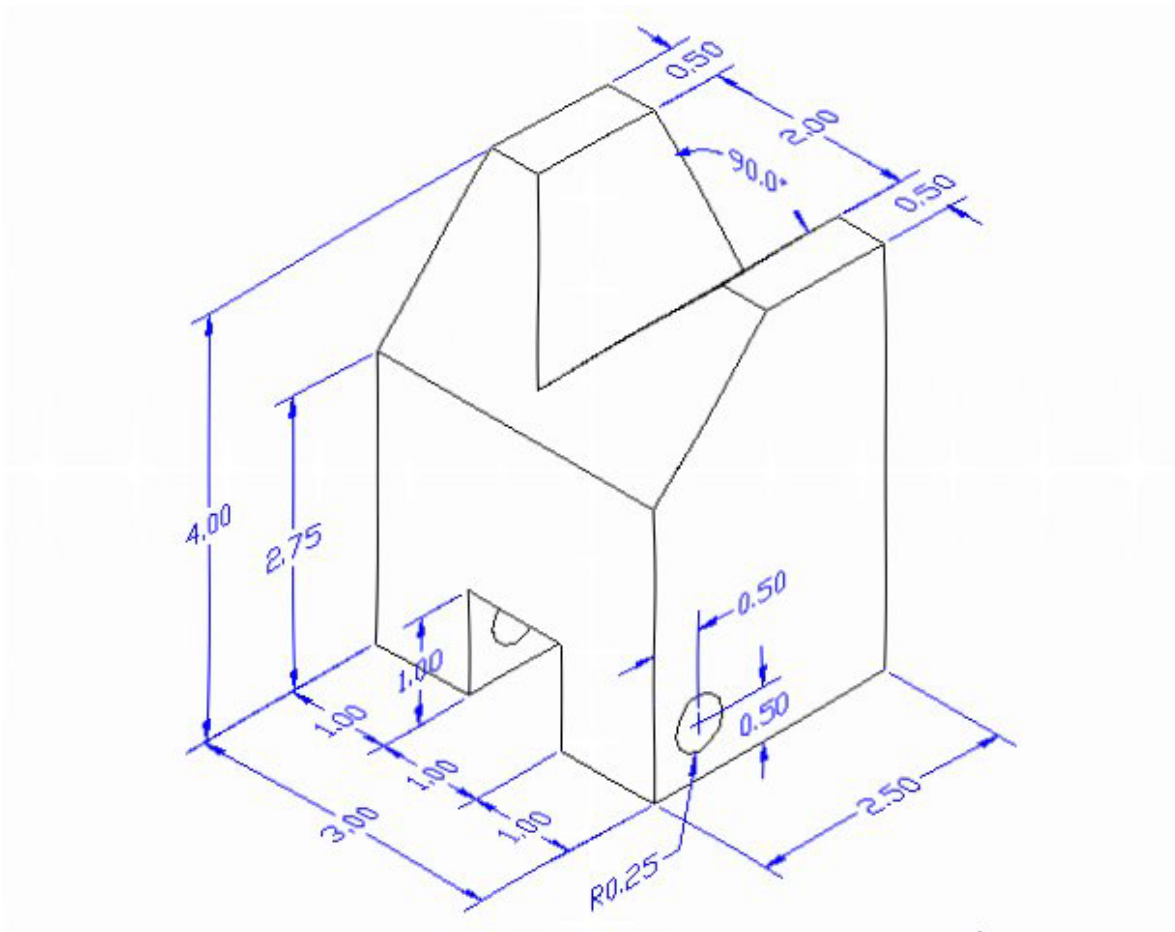


Figure 8.1 – Problem Seven Sketch

Remember, when you receive a sketch from a professional, you need to practice to quickly identify the different shapes and decide how to proceed with the solid exercise. You can see rather quickly that problem has different contours so separate each extrusion or solid that is made and subtract them from the master. This problem will help you address solid parts where there are different complexities and sizes. All ten drills in the Fundamentals of 3D Drawing textbook will give you those repetitive maneuvers, which will make you a successful in the 3D modeling.

Starting a 3D Part Drawing

In problem seven, you will begin the 3D drawing by selecting the Mechanical template. In Figure 8.2, the window “Create New Drawing” is loaded by selecting the New tool on the Standard Toolbar. Select the “Use the Template” (third button) at the top of the Create New Drawing window. A list of templates is loaded in the Select a Template list box. Scroll down the list and select Mechanical.dwt. A new drawing file will open that contains the system variables and layer definitions which you will use later in Paper Space to finish the drawing.

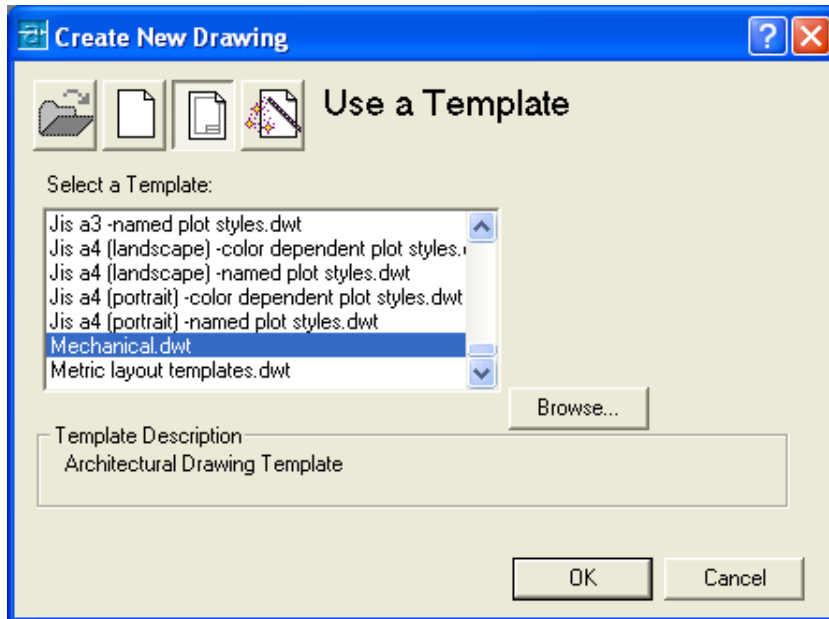


Figure 8.2 – Starting the Drawing Using the Metric Template

Modifying How the UCS Icon is Displayed

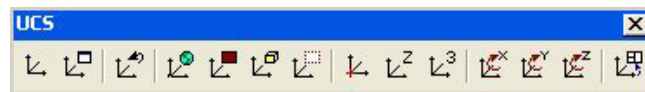


Figure 8.3 - The UCS Toolbar

When working in AutoCAD, you may notice the UCS icon moving on the graphical display showing the precise location of the origin. Many computer aided design operators do not appreciate having additional UCS lines in with their drawing or solid, so you can set the system to show the UCS in the lower left hand corner of the display.

Select the Display UCS Dialog tool on the UCS toolbar and the UCS window will appear the graphical display. Pick the Settings tab to make the UCS Icon settings change. Remove the check from “Display at UCS origin point” as shown in Figure 8.4 and press the OK button to have the system variable change. Now you can start to create the solid.

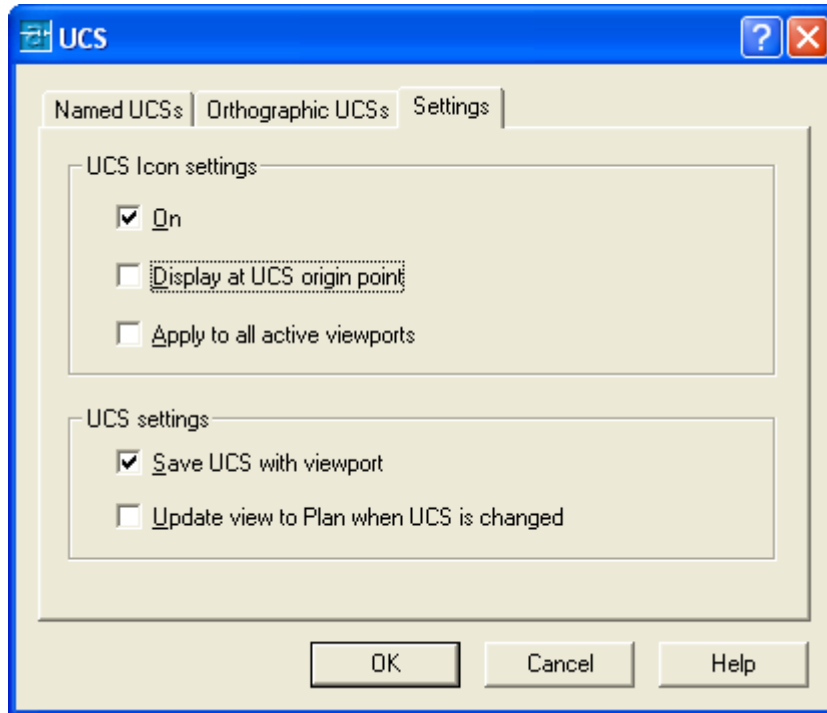



Figure 8.4 - The UCS Window, Settings Tab

Drawing a Closed Polyline Shape to be Extruded

On the Draw Toolbar, select the Polyline tool and specify a start point in the upper left hand corner of the graphical display. In Figure 8.5, you can see the actual continuous line of data you need to specify at the command line. 

```
Command: _pline
Specify start point:
Current line-width is 0.00
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: <Ortho on> 2.75
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 2.5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 4.0
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 1.25
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: c
```

Figure 8.5 – The Polyline Text from the Command Line

With the Ortho mode “on”, draw a **2.75** unit line downward, a **2.5** line to the right, a **4.0** line upward, a **1.25** line to the left and then type “**C**” to close the Polyline. A closed entity will appear in the graphical display. Select the Zoom Extents tool on the Standard toolbar to have the box fill the monitor, so you can proceed to the next step in the drawing process. In Figure 8.5 you can see the step-by-step progression of building the Polyline with Figure 8.6 showing the final two-dimensional entity.

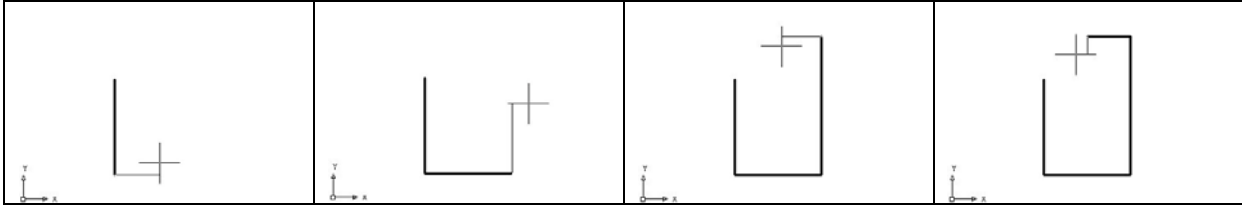
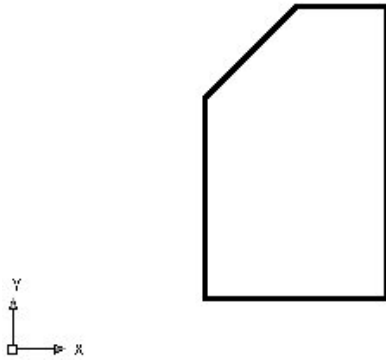


Figure 8.6 – Drawing a Series of Lines Using the Polyline Command



As you gain experience in drawing shapes in the AutoCAD software application, notice that you can start defining the perimeter anywhere on the graphical display and not just in the lower left hand corner. In this case, by describing the vertical and horizontal lines first, the last diagonal line is left to last. When you close the perimeter, the mathematical formulation of the angled line is left to the computer. Whenever the computer can automatically draw an entity, the more complex line can be left to the automatic feature.

Figure 8.7 – The Finished Polyline

Adding Circles to the Detail

Next, select the Circle command on the Draw toolbar. Place the mouse aperture at the lower left Endpoint as shown in Figure 8.8 and an Endpoint Osnap symbol (square) will appear on the end of the line. Select the center point of the circle by selecting with the left mouse button. Then type “D” when the command line prompts you to “specify radius of circle” and press **Enter**. Type the **0.5** diameter and the circle will appear on the corner as shown in Figure 8.8.

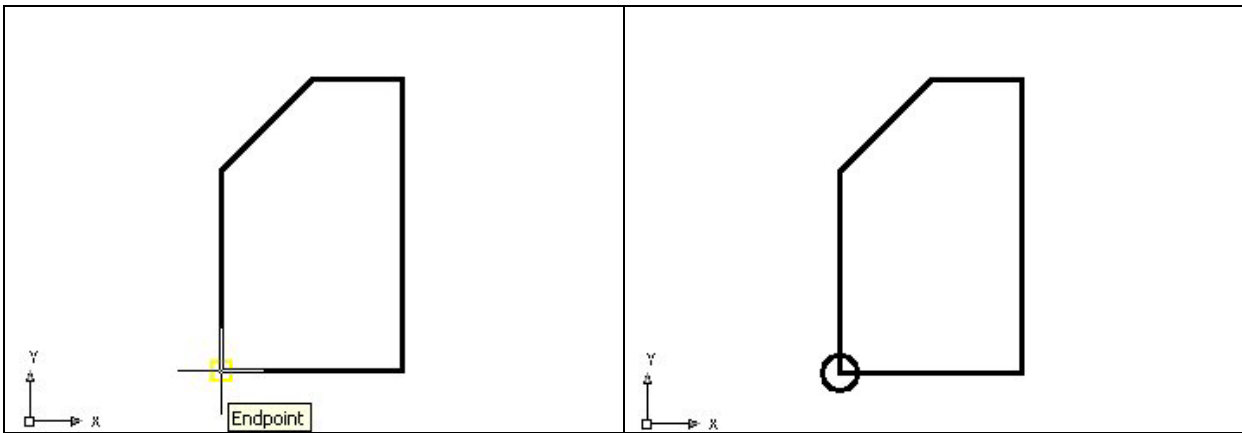
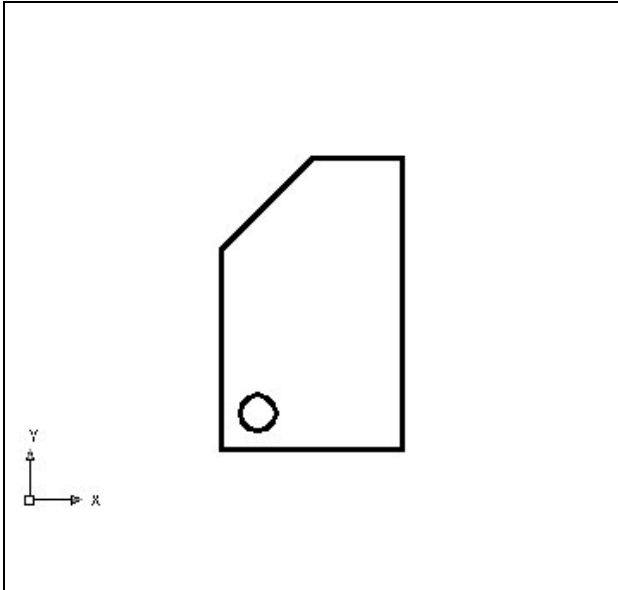


Figure 8.8 – Adding Circles For The Solid

Figure 8.9 – Adding Circles For The Solid

Moving a Circle Inside the Polyline

Before extruding both the Polyline and the Circle, you will need to position the circle at the precise location. Use the Move tool on the Modify toolbar to accomplish the task.



To move the circle, you select the Move tool on the Modify toolbar. Once you choose the Move command, the prompt will be “Select Objects”. Pick the circle and the command line will respond with “1 found”, so just press Enter to go the second part of the command. Next, you need to “specify base point or displacement” on the graphical display. Pick your base point anywhere on the display. Type **Enter** to proceed to the “second point of displacement” as input **@0.5,0.5** which will move the entity a distance of 0.5 in the X-axis, 0.5 in the Y-axis and when leaving the Z coordinate off, the default is 0 in the Z-axis. The drawing will appear as shown in Figure 8.10.

Figure 8.10 – Positioning the Circle

Extruding Solid from a Closed Polyline and Circle

Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Solids toolbar. You now have two shapes that are single entities.

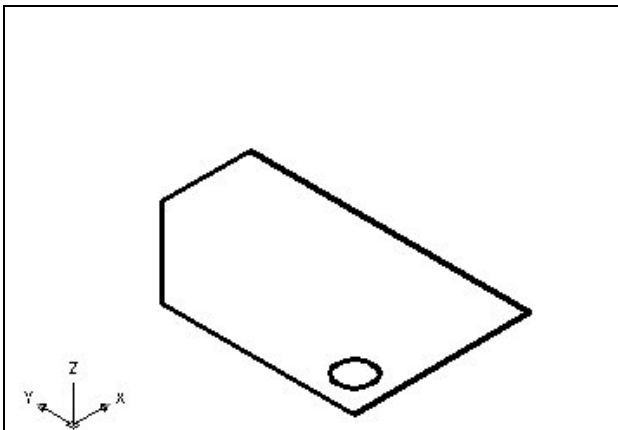


Figure 8.11 – Southwest View

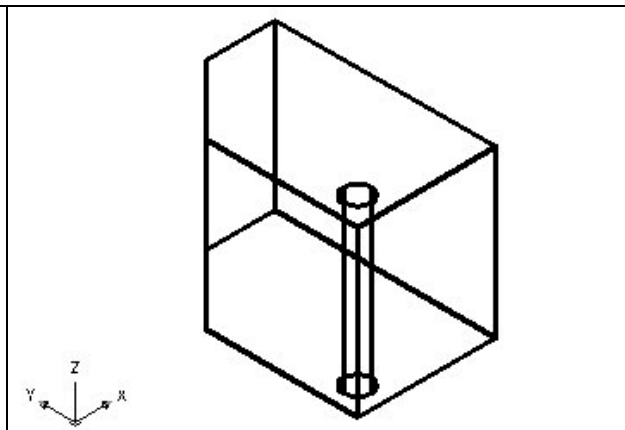


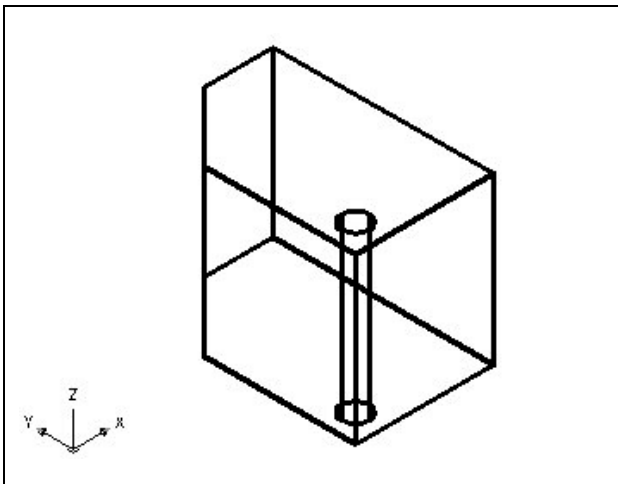
Figure 8.12 – Extruding Two Entities

On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on

the XY plane and the vertical Z-axis will rise upward as shown in Figure 8.11. Select the Extrude tool on the Solids toolbar and the command line on the AutoCAD application will prompt you to “**select objects**”. Pick to the upper right of the graphical display and going to the left, you will see the dotted selection window signifying the “crossing” option in selecting entities. The command line will return with “**2 found**”. Press **Enter** to specify the height or path of the solid. You will type “**3.0**” for the height of the extrusion and for the angle of taper; you will **Enter** to accept the zero degree default. Two complex 3D solid will appear in the graphical display as shown in Figure 8.12.

Subtracting a 3D Solid from Another Solid

To remove the cylinders from the master solid, select the Subtract tool on the Solids Editing toolbar.



Then the AutoCAD program will prompt you with “Select solids and regions to subtract from”. Select the larger master solid and the system will respond with “1 found”. You need to **Enter** to proceed to the next step, which is “Select solids and regions to subtract”. Pick the three cylinders and the software application will counter with “1 found”. **Enter** and a region exactly the shape of the two cylinders is missing from the larger one as shown in Figure 8.13. Notice that you cannot see any difference in the solid when you subtract the cylinders.

Figure 8.13 – Extruding Two Entities

Rotating a 3D Solid Using the Rotate3D Command

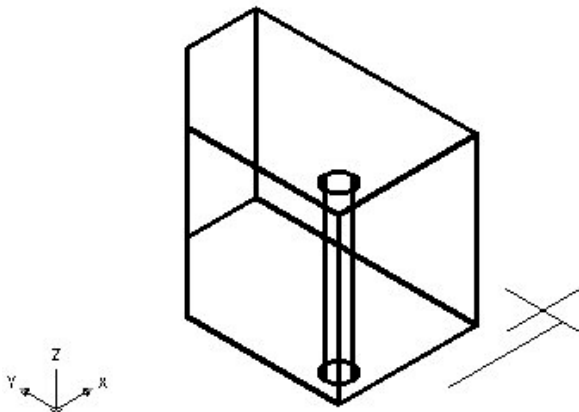


Figure 8.14 – Extruding Two Entities

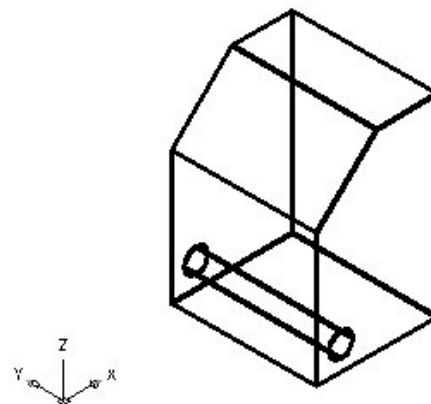
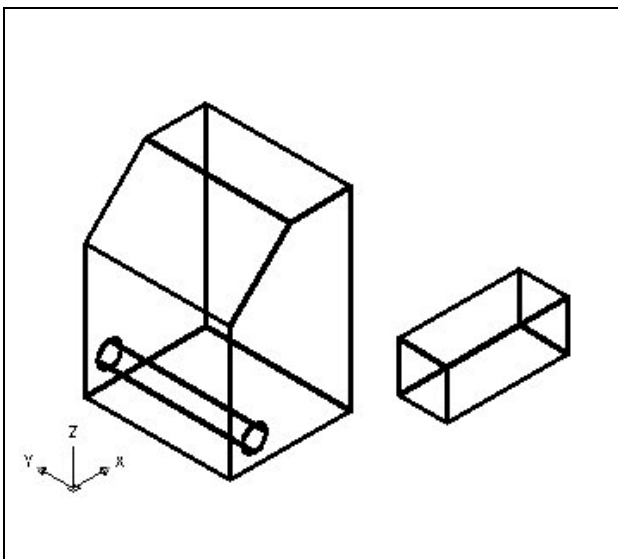


Figure 8.15 – Extruding Two Entities

You can rotate the solid to the left on the X-axis using the Rotate3D command. Select the Modify on the Menu Bar, then 3D Operations and pick Rotate 3D from the list of four commands. At the command, “select objects”, pick the solid on the graphical display and **Enter** to proceed to the second part of the function. You are going to rotate the solid part on the x-axis, so pick any point on the display and then with the Ortho mode still “on” pick a second point drawing a line on the x-axis as shown in Figure 8.14. Now you type **-90** to rotate the solid as shown in Figure 8.15.

Drawing a Solid Box

The technique you will use in this exercise will utilize the Length option of the Box command. Type **L** after specifying the corner on the graphical display and the system will ask three questions. The first prompt is “Specify Length” where when you type a number and Enter will designate the **X** coordinate. The second prompt is “Specify Width” where when you type a number and Enter will designate the **Y** coordinate. The third prompt is “Specify Height” where when you type a number and Enter will designate the **Z** coordinate.



Select the Box tool on the Solids toolbar. At the command line, a prompt asks you to specify the corner of the solid and for this box, specify the any point to the right of the solid. Then, type **L** after specifying the starting point and the system will ask you three questions. The first prompt is “Specify Length” where then you will type **2.5** and Enter for the X coordinate. The second prompt is “Specify Width” where you will type a **1.0** for the y coordinate. The third prompt is “Specify Height” where when you type **1.0** and Enter for the **Z** coordinate. The second box will appear as shown in Figure 8.16.

Figure 8.16 – Extruding Two Entities

Moving a 3D Solid

The Move command is actually a three-dimensional function in the modern AutoCAD application software program. The way you can determine whether a command is three dimensional in nature is to attempt to use the function outside the Z-plane, that is the Z coordinate is not 0. You can use move to relocate any entity to another point (X,Y,Z) in Model Space.



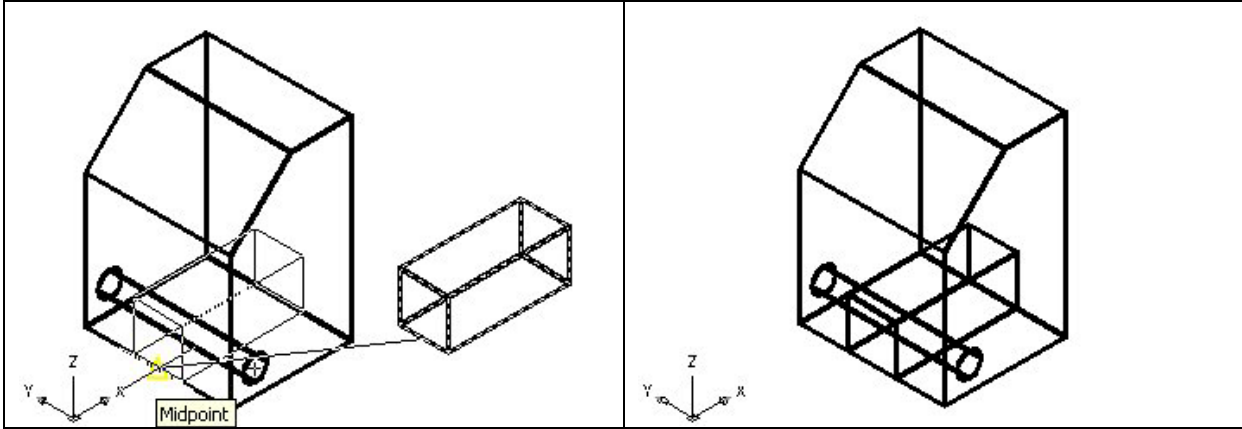


Figure 8.17 – Moving the Box

Figure 8.18 – The Moved Box

To move the box, you select the Move tool on the Modify toolbar. Once you choose the Move command, the prompt will be “Select Objects”. Pick the solid box and the command line will respond with “1 found”, so just press **Enter** to go the second part of the command. Next, you need to “specify base point or displacement” on the graphical display. Pick your base point with the Midpoint Osnap selecting the Midpoint of the box as shown in Figure 8.17. The “second point of displacement” is the center of the bottom left of the master solid. Pick the second point using the Midpoint Osnap and the solid box will move as shown in Figure 8.18.

Constructing a Solid Perimeter Using the Line Command

You need to create a right triangle with a 2-inch hypotenuse. You may want to be in the top orthographic view to draw the three-sided shape. After changing your view, select the Line tool from the Draw toolbar.



Again, you will allow the computer to do the mathematical work to create the triangle. Pick the starting point to the left of the solid and with the Ortho “On”, draw a line 2.0 to the left. Now you will draw a line using polar displacement. Type **@2.0<-45** and a line will as shown in Figure 8.19. **Enter** to exit the Line command.

As before using relative coordinates, the “@” symbol moves the entity from the previous point. The 2.0 is the distance required to move and the “-45” is the angle in degrees from the base of “0” degrees. In AutoCAD, the zero degree base angle is in the right or positive X-axis. The “-45” angle draws the line 45 degrees to the bottom of the 2.0 line.

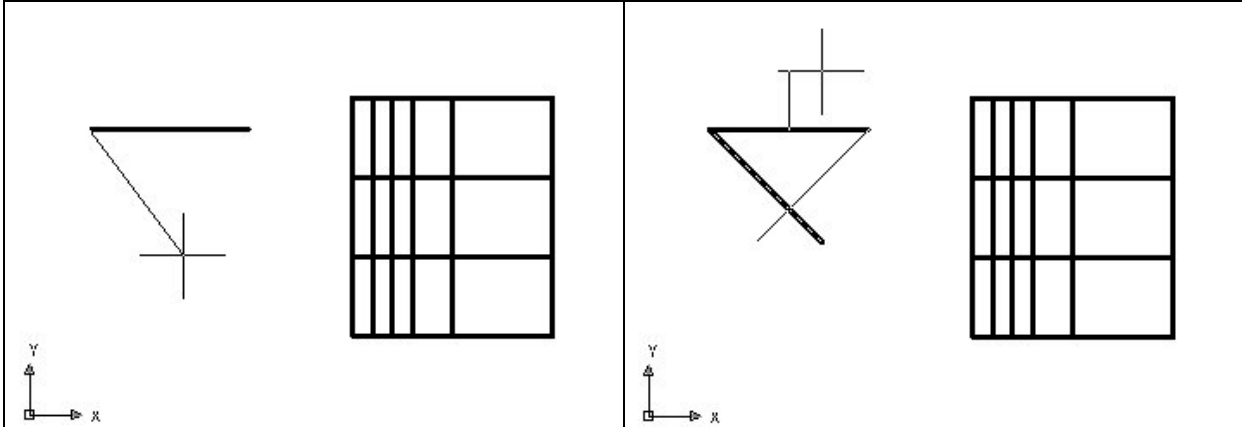


Figure 8.19 – Drawing the Line

Figure 8.20 – Mirroring a Line

You will use the Mirror tool on the Modify toolbar to copy the 45-degree line across the midpoint of the 2.0 line. At the prompt to select objects, pick the angled line and then **Enter** proceed to the second half of the Mirror command. “Specify first point of mirror line:” by picking the Midpoint of the line and “Specify second point of mirror line:” by picking just above the first point with the Ortho “On” as shown in Figure 8.20. The default to delete the source objects is no, so at the prompt “Delete source objects? [Yes/No] <N>:” just **Enter** and the mirrored and duplicated entities from the top half will appear on the bottom half of the top orthographic view. (See Figure 8.17)

Pick the Fillet tool on the Modify toolbar, type “**R**” for radius, “**0**” and **Enter**. Select the two angled lines near their midpoints and the intersection of the 45-degree triangle will become a zero radius. You can either use the Fillet or Trim tool to trim the lines as shown in Figure 8.22.

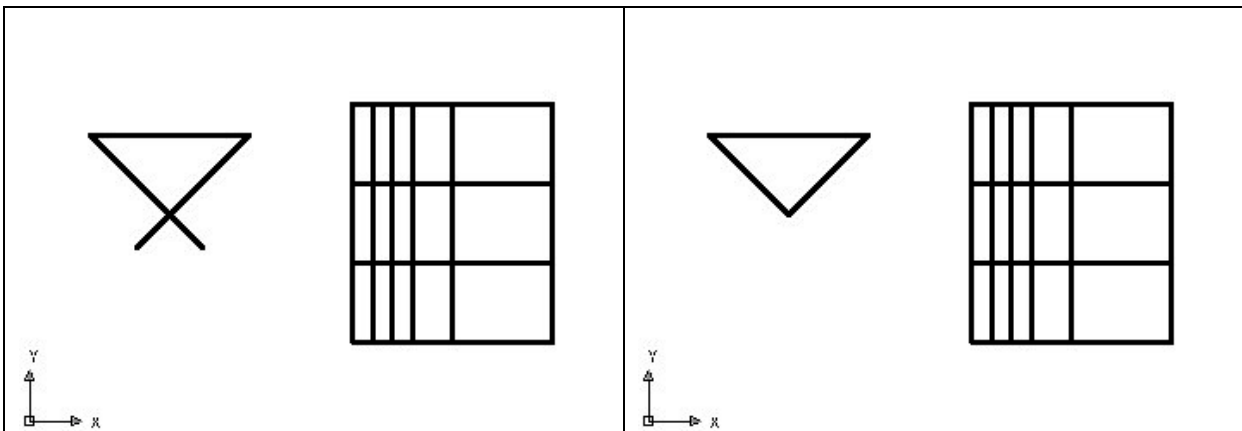


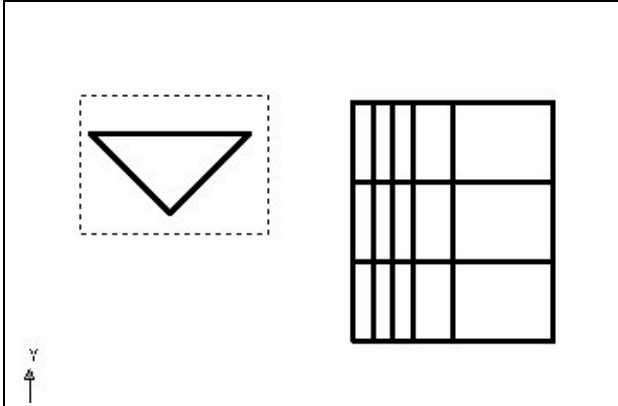
Figure 8.21 – Fillet the Lines, Radius 0

Figure 8.22 – The Finished Triangle

Creating Polylines Using the Edit Polyline Tool

To merge the three lines segment together you need to use the Edit Polyline tool. Select the Edit Polyline tool on the Modify II toolbar, and then select the objects.





Pick to the upper right of the graphical display as shown in Figure 8.23 and going to the left, you will see the dotted selection window signifying the “crossing” option in selecting entities. Pick again to make the crossing selection. The command line will return with “Do you wish to turn the line into a Polyline <y>”. Enter to convert a line to a Polyline and at the next prompt, type “J” for join. Reselect the lines and all adjoining lines will become a Closed Polyline.

Figure 8.23 – Polyline Edit the Right Side

Extruding a Solid from Closed Polyline

Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Solids toolbar. You can extrude multiple Polylines and Circles at a time with the Extrude command.

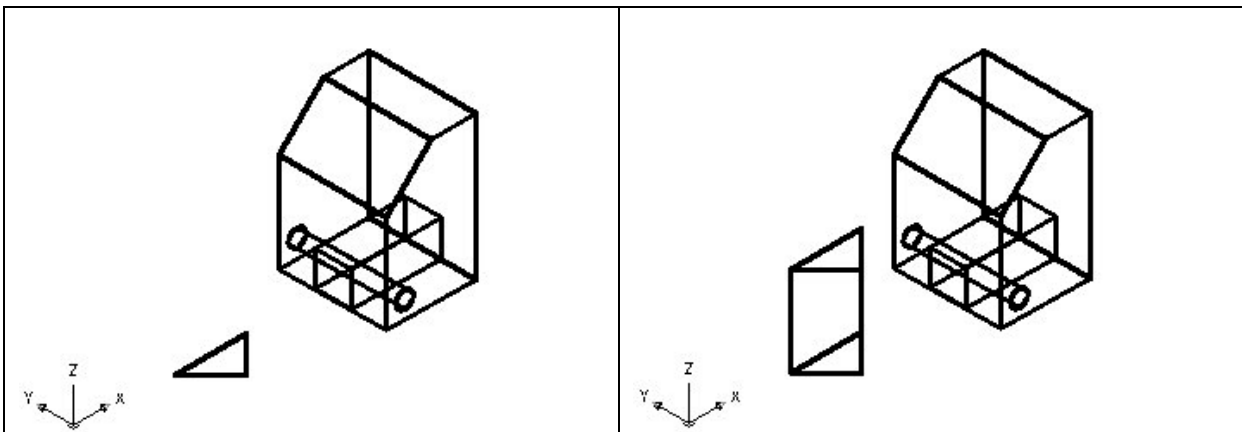


Figure 8.24– Polyline Edit the Right Side

Figure 8.25 – Polyline Edit the Right Side

Select the Extrude tool on the Solids toolbar and the command line on the AutoCAD application will prompt you to “select objects”. Pick the triangle and the command line will return with “1 found”. Press **Enter** to specify the height or path of the solid. You will type “2.5” for the height of the extrusion and for the angle of taper; you will **Enter** to accept the zero degree default. A more complex 3D solid will appear in the graphical display as shown in Figure 8.25.

Aligning a Solid to the Master Solid

Select the Modify on the Menu Bar, then 3D Operations and pick Align from the list of four commands. At the command, “select objects”, pick the solid on the graphical display and **Enter** to proceed to the second part of the function.

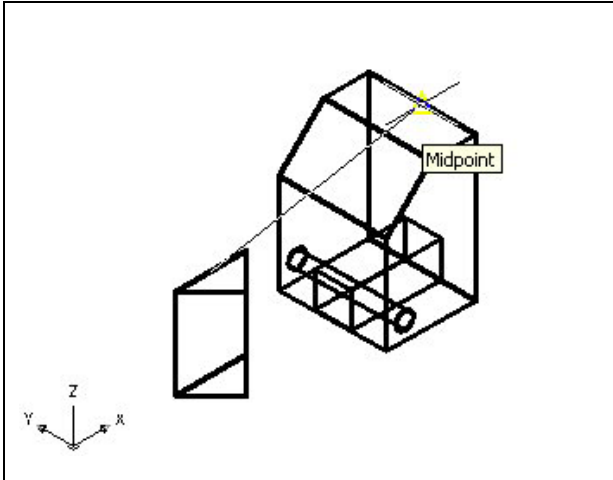


Figure 8.26 – Polyline Edit the Right Side

The first alignment point brings the two points selected together. Of the three choices for alignment, you can only guarantee that the first group will unite exactly. Unless the solids are exactly the same size this will always be true. This makes the first selection the most important, since you will only want to run the command once. In Figure 8.26, the first point of alignment is the Midpoint on the top of the triangle. The destination is the Midpoint on the upper back tessellation line of the master solid. A temporary line appears on the display showing the connection.

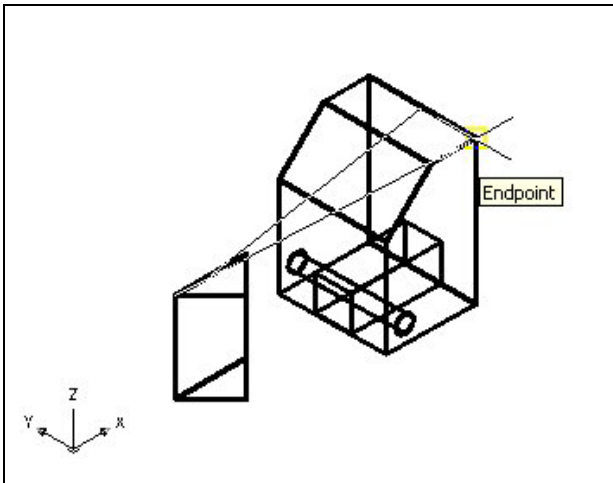


Figure 8.27 – Polyline Edit the Right Side

The second set of point will signify the two dimensional line of alignment. You can see in Figure 8.27 that the first pick is the upper left corner of the triangular solid. The second selection is the upper right corner of the master solid. The sides are different in height but these two tessellation lines will unite. The two exact points defining this line will be the first and second point selections. If you were doing a two-dimension drawing, this would be your last set of picks, and by Entering through the last prompts, you would make the 2D alignment.

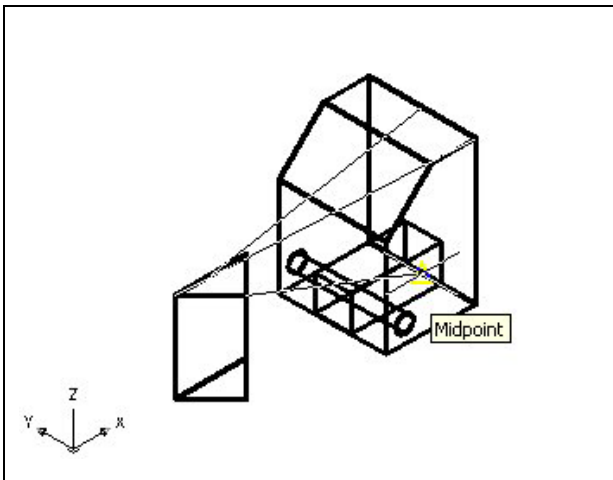
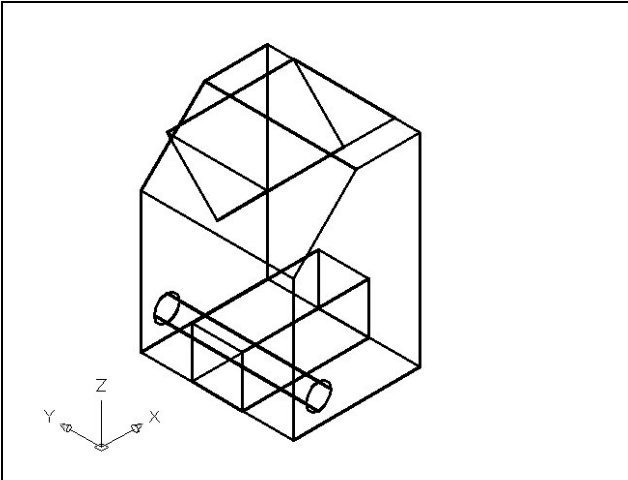


Figure 8.28 – Polyline Edit the Right Side

The last group of points you will select is to align the planes of both solids. You can see in Figure 8.28 that the first pick is the lower right corner of the triangular solid. The second selection is the lower rear Midpoint of the master solid. As in each set of associations, a temporary line appears on the graphical display showing the relationship. This last two picks will force an arrangement between the two planes that you are showing by the selection of three points. You may remember from previous math classes that any three points will represent a plane.

Subtracting 3D Solids from the Master Solid

To remove the two solid details from the master solid, select the Subtract tool on the Solids Editing toolbar.

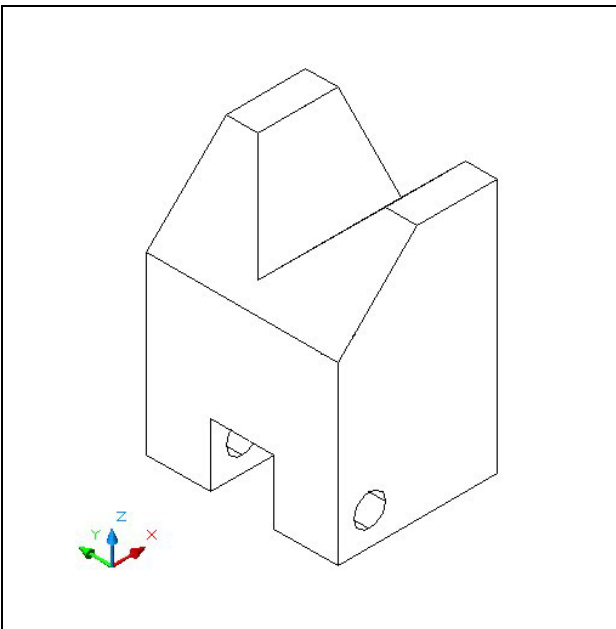


Then the AutoCAD program will prompt you with “**Select solids and regions to subtract from**”. Select the larger master solid and the system will respond with “**1 found**”. You need to Enter to proceed to the next step, which is “**Select solids and regions to subtract**”. Pick the box and triangular solid and the software application will counter with “**2 found**”. Enter and a region exactly the shape of the two cylinders is missing from the larger one as shown in Figure 8.29. You will want to use the Hide command to see your results better.

Figure 8.29 – Subtract the Solids

Hide the 3D Solid

In Figure 8.25, you will notice that the tessellation lines representing the solid’s outline on the far edge of the part are visible. In a real world solid part, you would not be able to see through solid unless the material was transparent. You can use the Hide command to remove the hidden lines from your graphical display.



Select the Hidden tool on the Shade toolbar. This is a very simple command, because moments after picking the tool, the tessellation lines representing the back of the part are removed from view. Notice in Figure 8.30 that the UCS icon changes from the thin lined and arrowed icon to a colorful image. If you wish to return the graphical display to the previous setup where all the tessellation lines are seen, then choose the first tool on the Shade toolbar, called 2D Wireframe. By choosing the 2D wireframe tool, the solid part and the UCS icon will return to the previous appearance. This command does not alter the mass of the solid or any of the dimensions, but is available for the computer aided designer as a viewing option.

Figure 8.30 – Hide the Solid

Moving the Solid to the Origin Point

To move problem seven to the drawing origin of 0,0,0 on the x, y and z axis, pick the Move tool on the Modify toolbar and select the 3D solid and Enter. At the prompt, “**specify base point or displacement**”, select the endpoint shown in Figure 8.31 at the lower left hand side of the problem seven, which will become the problem seven insertion point into an assembly drawing. At the “second point of displacement”, type **0,0,0** and **Enter**. Problem seven will move to the new origin point. There is more than one way to do many actions in AutoCAD, but the end result of having a part at the origin is important when you wish to insert the solid into an External Referenced Assembly drawing.

*** World Class CAD Challenge 03-07 * - Close this drawing file. Create a New file and draw the solid using the techniques in this chapter. Move the finished solid to the origin of the drawing. Complete the task in less than 5 minutes. Continue this drill four times, each time completing the drawing under 5 minutes to maintain your World Class ranking.**

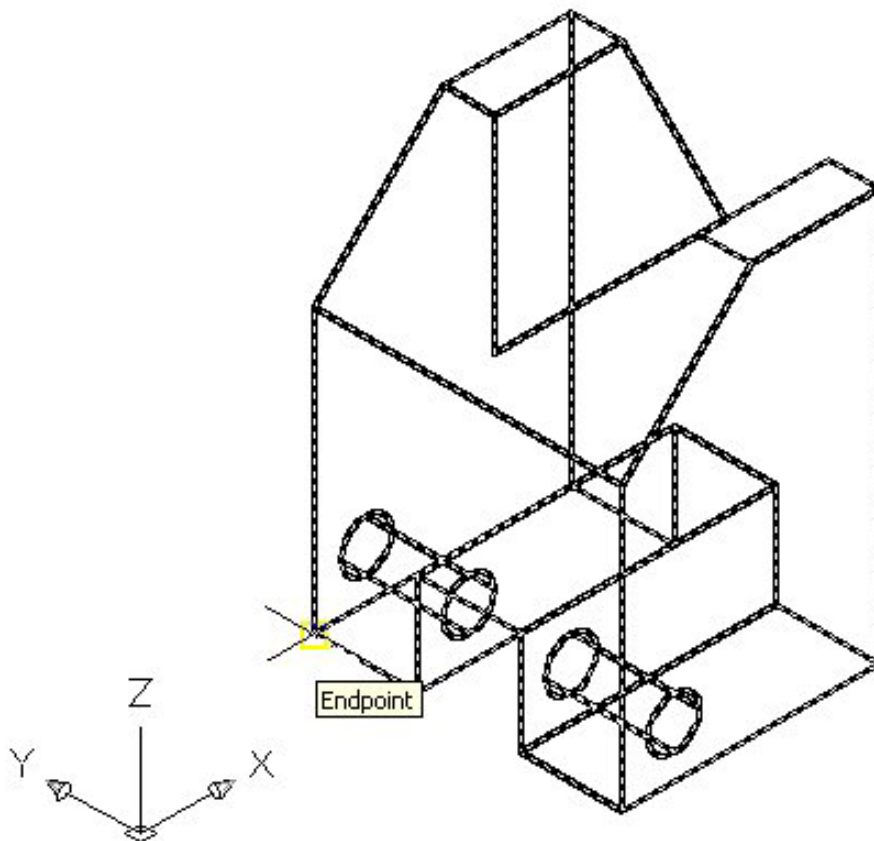


Figure 8.31 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Problem 7 in your Fundamentals of 3D Drawing Folder, select the Save tool on the Standard toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. At the File Name textbox, type “Problem 7” and press the Save button to save the drawing. (See Figure 8.32)

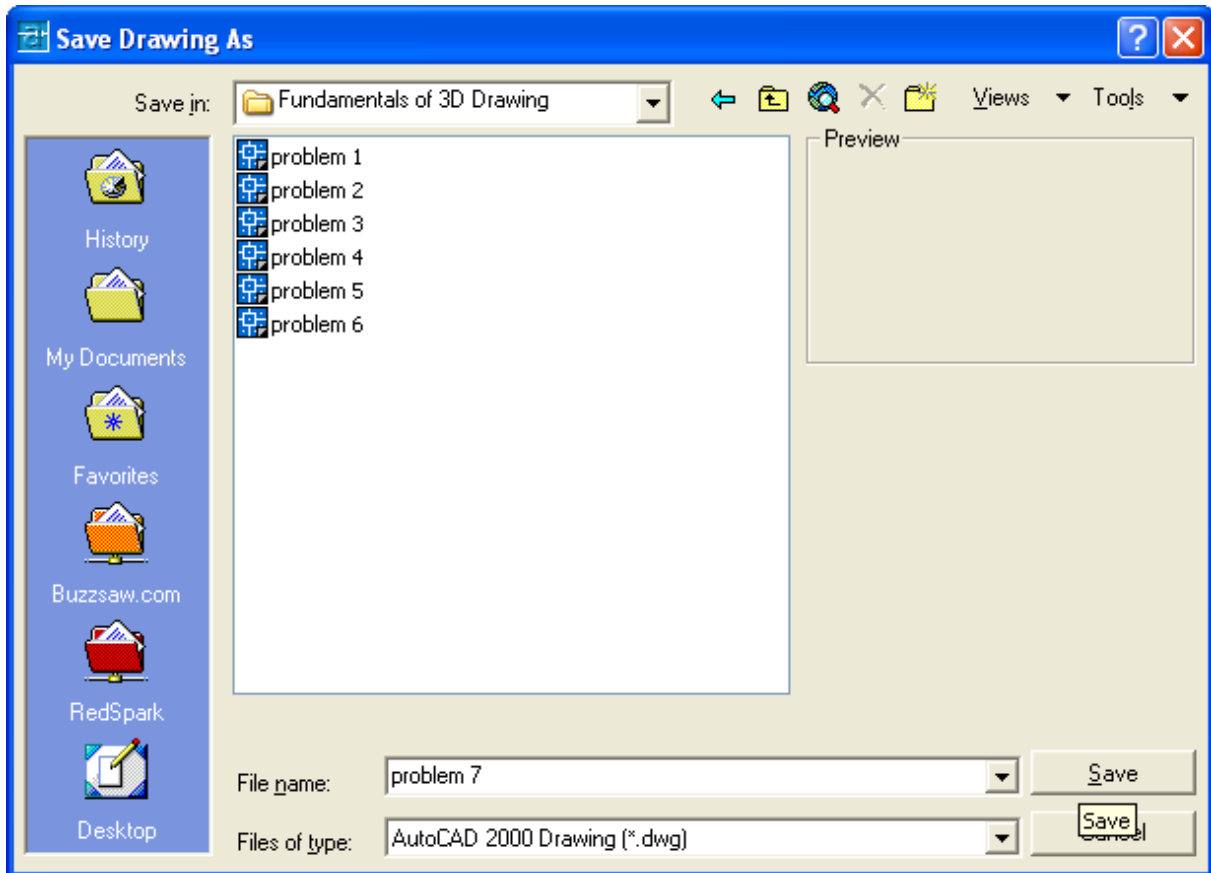


Figure 8.32 – Saving Problem Seven

*** World Class CAD Challenge * - Report your best times to World Class CAD at www.worldclasscad.com to obtain your world class ranking.**