# C h a p t e r

# 8

# 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

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You 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 solids, you will 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.
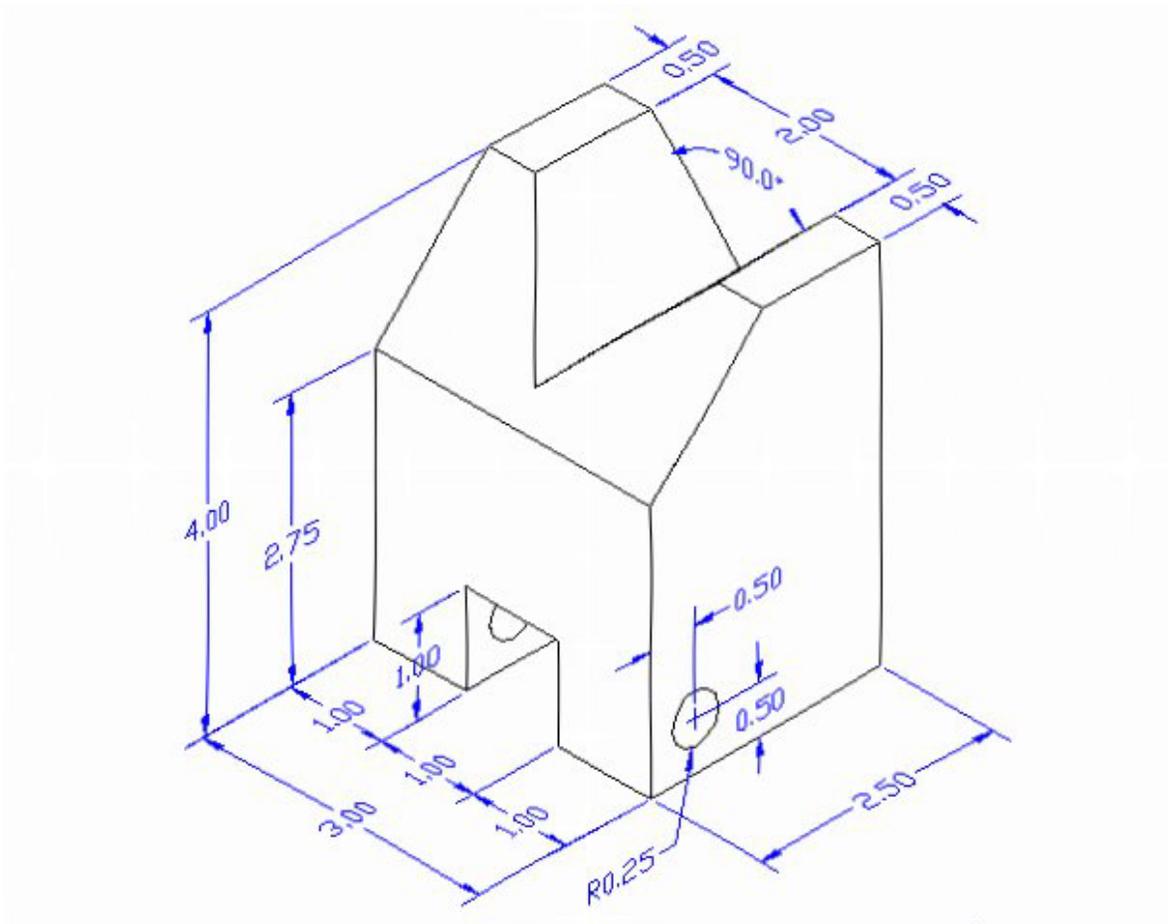


**Figure 8.1 – Problem Seven Sketch**

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

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In problem seven, you will begin the 3D drawing by selecting the Mechanical template. In Figure 8.2, the window "Select template" is loaded by selecting the New tool on the Standard Toolbar. 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.
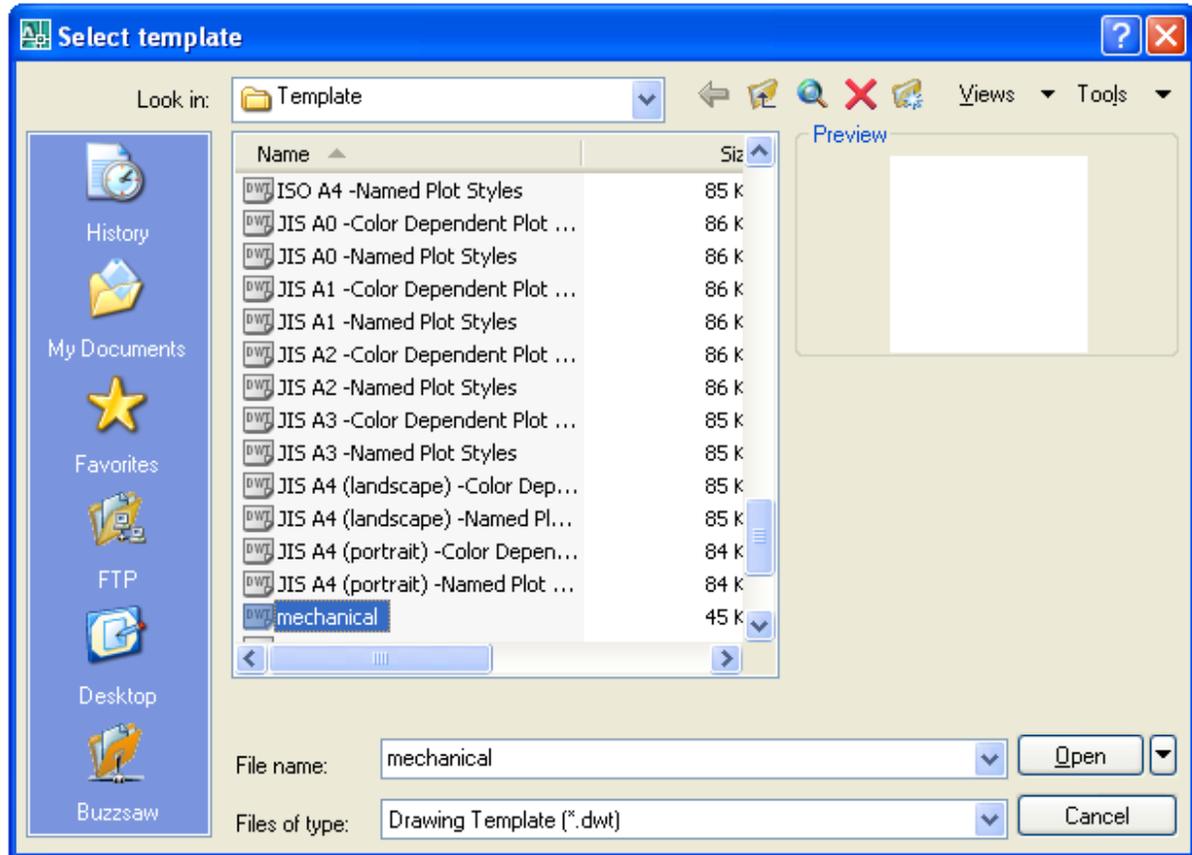


**Figure 8.2 – Starting the Drawing Using the Metric Template**

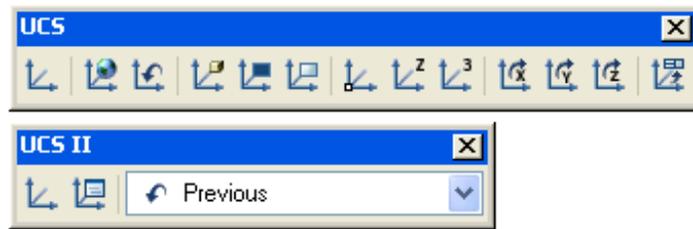# Modifying How the UCS Icon is Displayed

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**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 addition 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 II toolbar (Figure 8.3) and the UCS window will appear the graphical display. Pick the Settings tab to change the UCS Icon settings. 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.
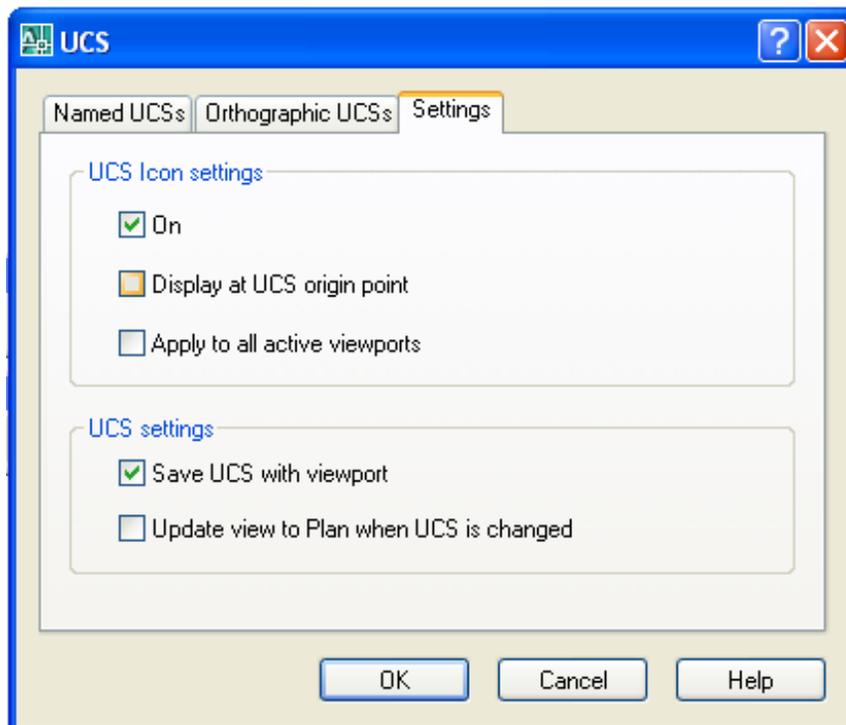


**Figure 8.4 - The UCS Window, Settings Tab**

# Drawing a Closed Polyline Shape to be Extruded

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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.

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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.
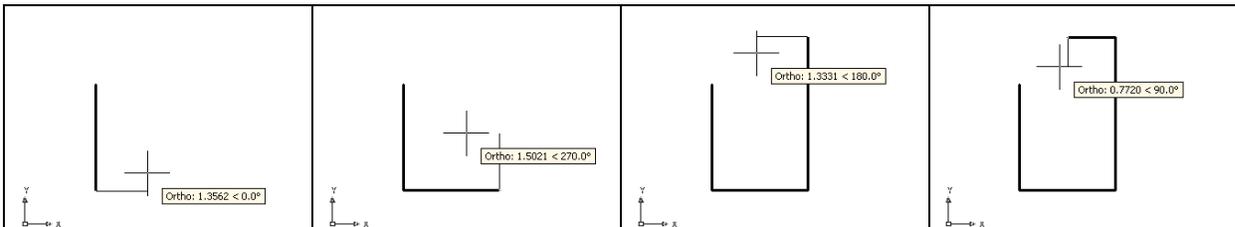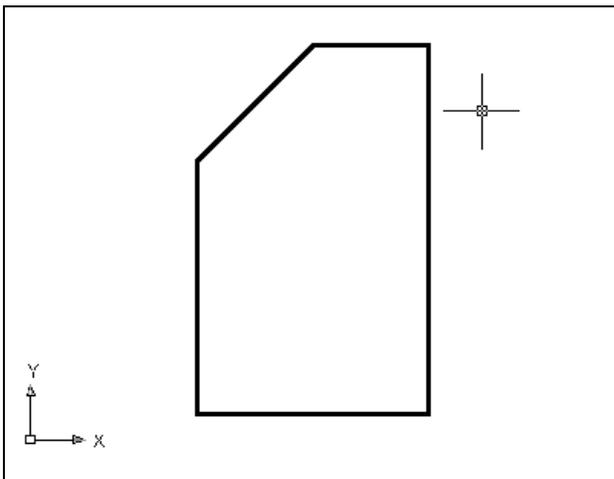


**Figure 8.6  – Drawing a Series of Lines Using the Polyline Command**



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 remains. 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 comutations can be left to the automatic feature.

**Figure 8.7 – The Finished Polyline**

# Adding Circles to the Detail

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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 **0.5** as the diameter and the circle will appear on the corner as shown in Figure 8.8.
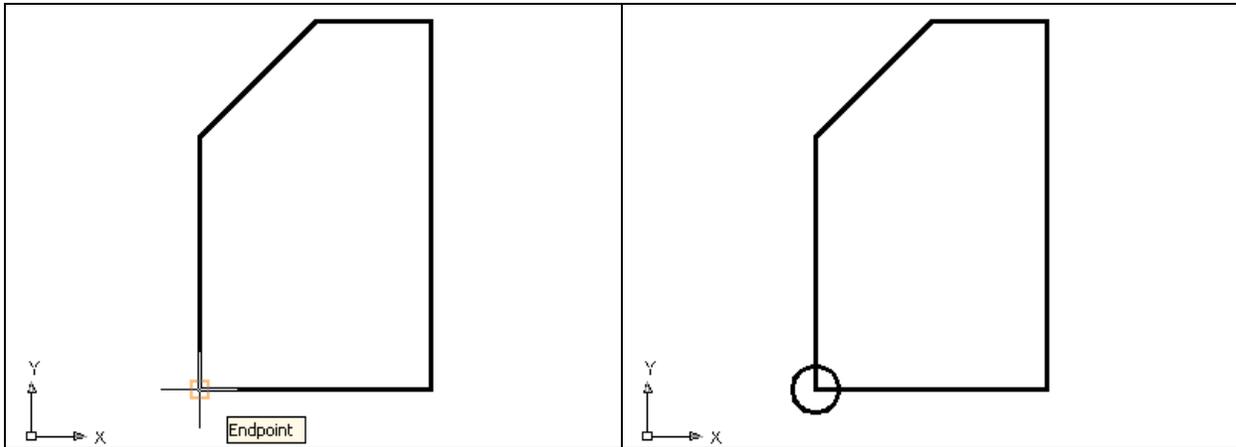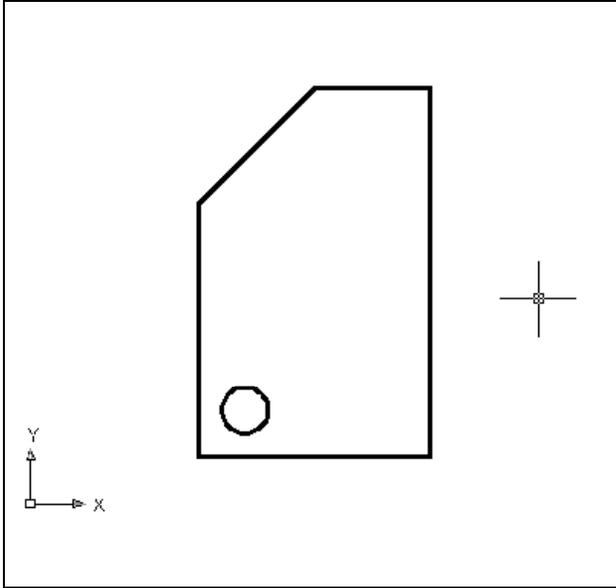


**Figure 8.8 – Adding Circles For The Solid**    **Figure 8.9 – Adding Circles For The Solid**

# Moving a Circle Inside the Polyline

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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.

**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.
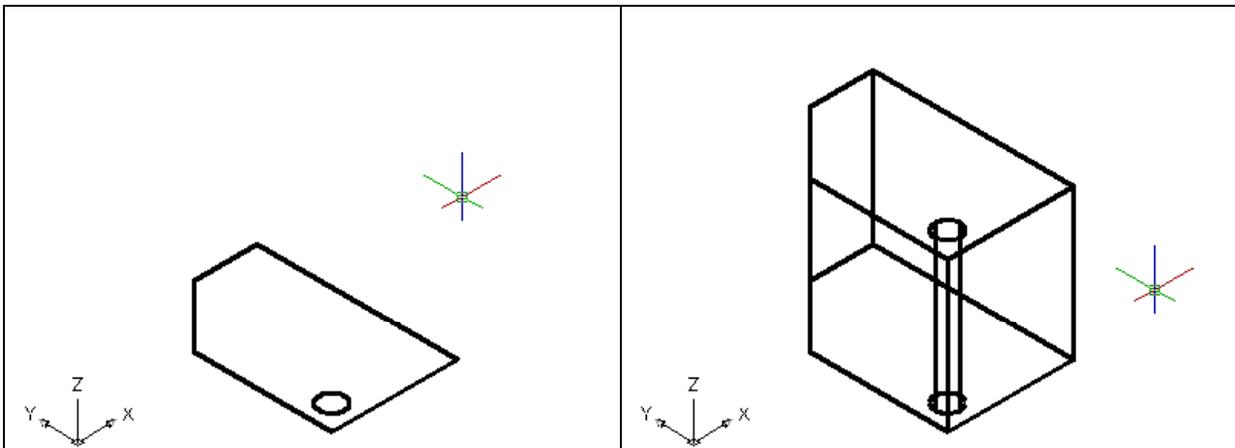
To move the circle, select the Move tool on the Modify toolbar. Once you choose the Move command, the prompt will be "Select Objects". Pick the circle and the command line will respond with "**1 found**", so just press Enter to go the second part of the command. Next, you need to "specify base point 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 8.11 – 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 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

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



**Figure 8.13 – Subtracting The Cylinder**
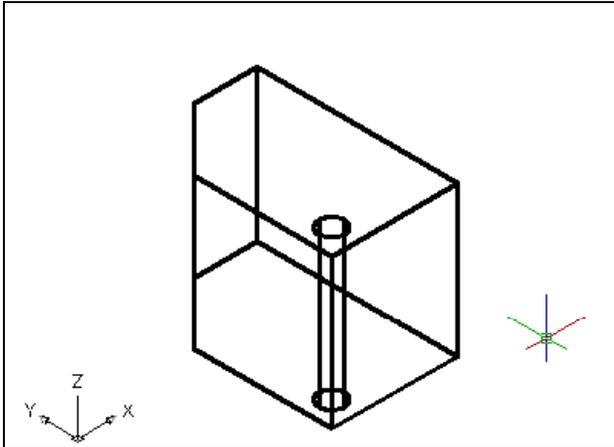
AutoCAD will prompt you to "Select solids and regions to subtract from". Select the larger master solid and the system will respond with "1 found". Hit **Enter** to proceed to the next step, which is to "Select solids and regions to subtract". Pick the cylinder and the software application will counter with "1 found". Hit **Enter** and a region exactly the shape of the cylinder is missing from the block as shown in Figure 8.13. Notice that you cannot see any difference in the solid when you subtract the cylinder.

## Rotating a 3D Solid Using the Rotate3D Command
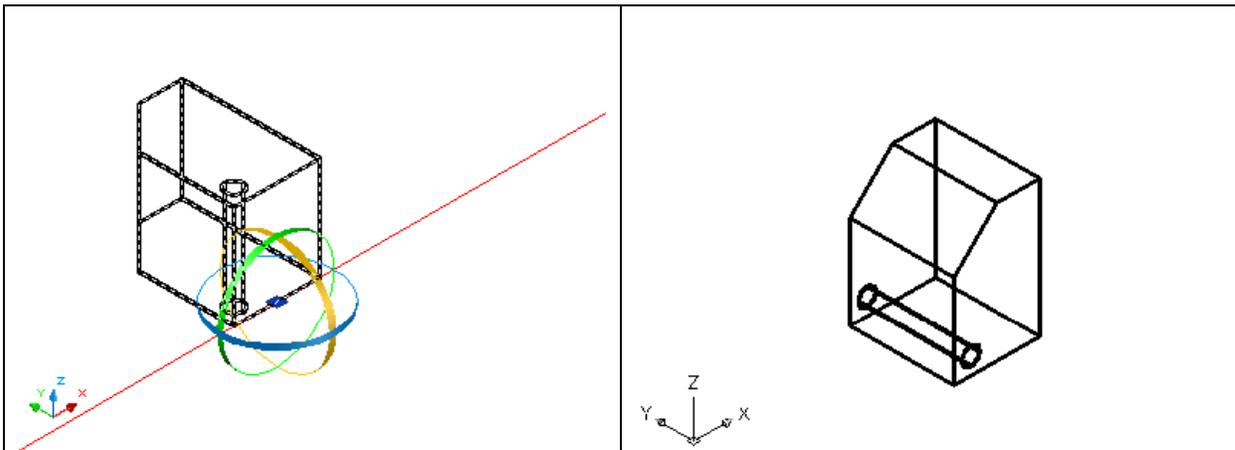
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**Figure 8.14 – Selecting the X Axis**     **Figure 8.15 – The Rotated Solid**

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 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 select the x-axis orbit in the sphere as shown in Figure 8.14 and then with the Ortho mode still "on" pick a second point drawing a line perpendicular to the x-axis. 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 to "Specify Length" where the number you type will designate the **X** coordinate. The second prompt is "Specify Width" where the number you type will designate will designate the **Y** coordinate. The third prompt is "Specify Height" where the number you type will designate will designate the **Z** coordinate.
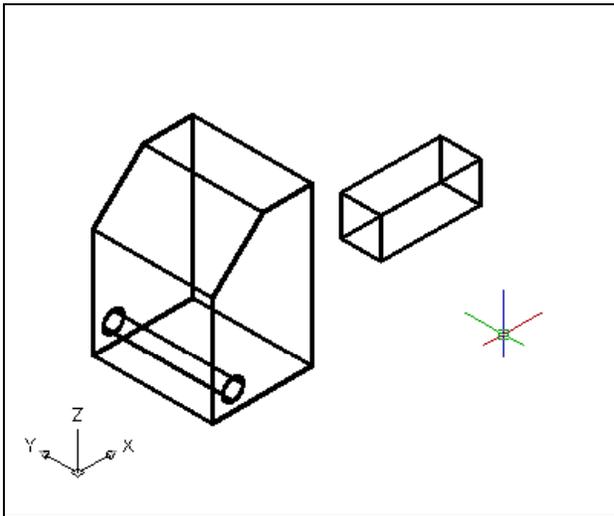


Select the Box tool on the Solids toolbar. At the command line, a prompt asks you to specify the corner of the solid and for this box, specify the any point to the right of the solid. Type **L** after specifying the starting point and the system will ask you three questions. The first prompt is "Specify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

**Figure 8.16 – A New Box**

# 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.
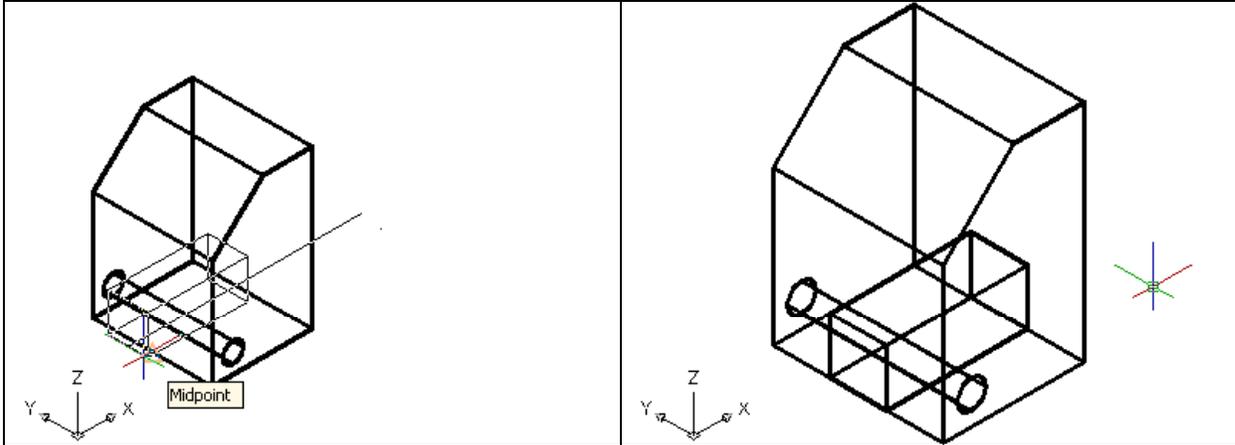
**Figure 8.17 – Moving the Box**          **Figure 8.18 – The Moved Box**

To move the box, select the Move tool on the Modify toolbar. Once you choose the Move command, the prompt will be to "Select Objects". Pick the solid box and the command line will respond with "1 found", so 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 by 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

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You need to create a right triangle with a 2-inch hypotenuse, so 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.
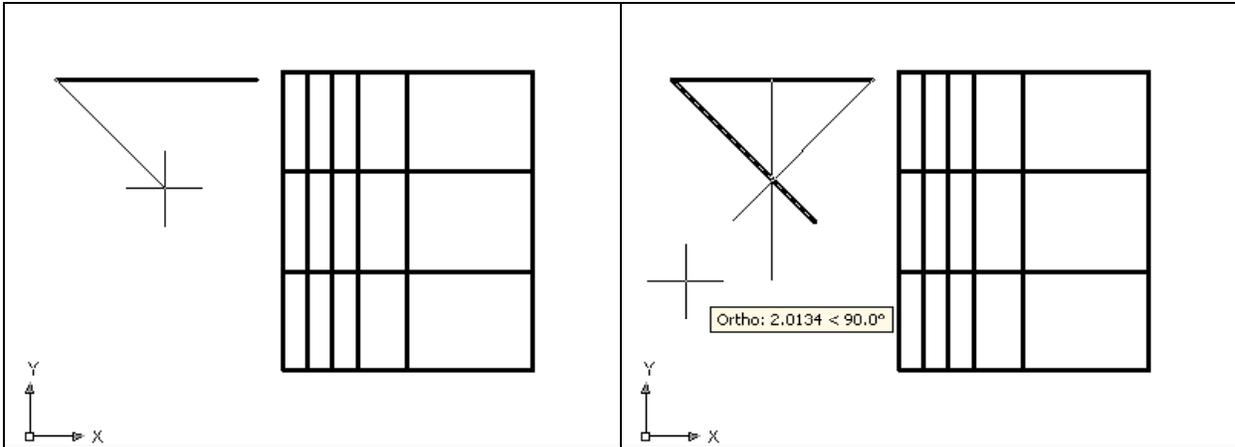
**Figure 8.19 – Drawing the Line**          **Figure 8.20 – Mirroring a Line**

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 hit **Enter** to 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 below 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>:" hit **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.
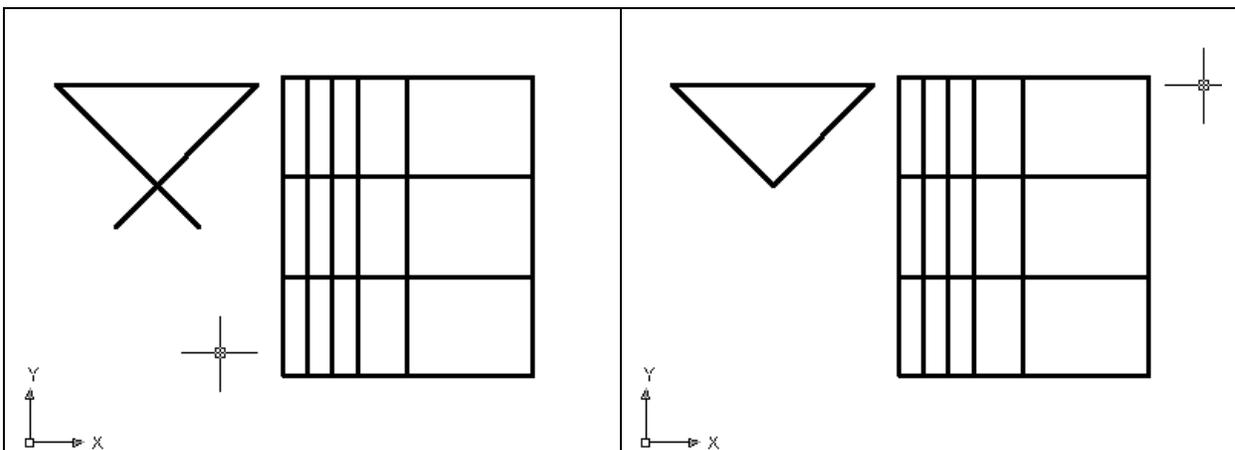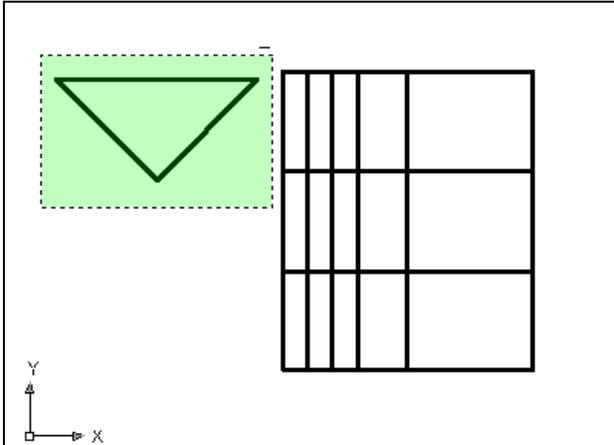


**Figure 8.21 – Fillet the Lines, Radius 0**     **Figure 8.22 – The Finished Triangle**

# Creating Polylines Using the Edit Polyline Tool

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



Type "M" and Enter for Multiple, and then 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" and Enter 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

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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.
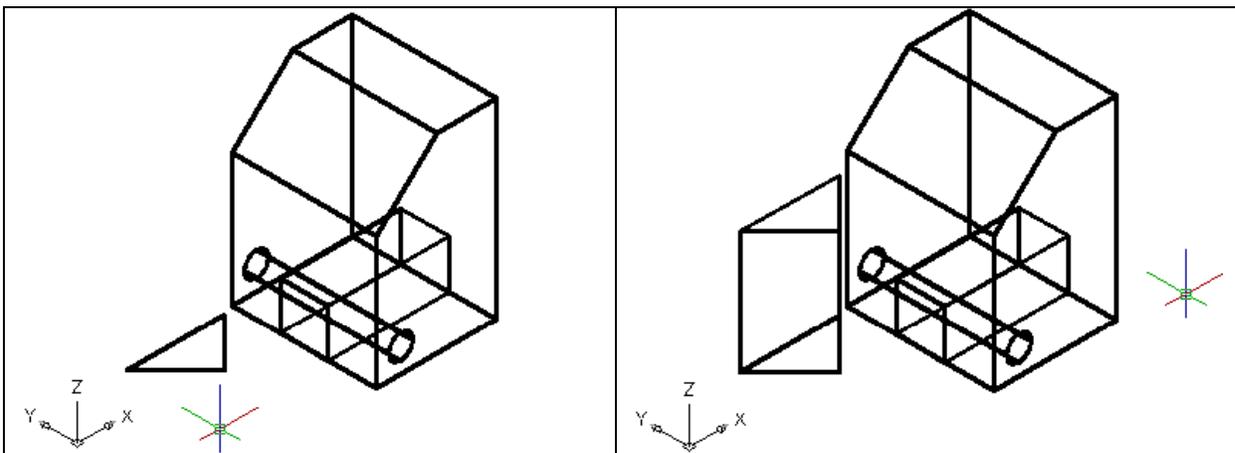


**Figure 8.24– SW Isometric View**          **Figure 8.25 – Extruding the Triangle**

In the Southwest Isometric view, select the Extrude tool on the Solids toolbar. The command line 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. Type "**2.5**" for the height of the extrusion and for the angle of taper, hit **Enter** to accept the zero degree default. A more complex 3D solid will appear in the graphical display as shown in Figure 8.25.
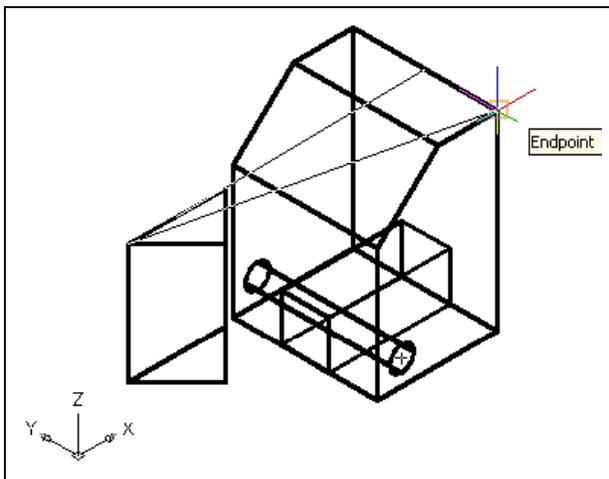
# Aligning a Solid to the Master Solid

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Select 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.
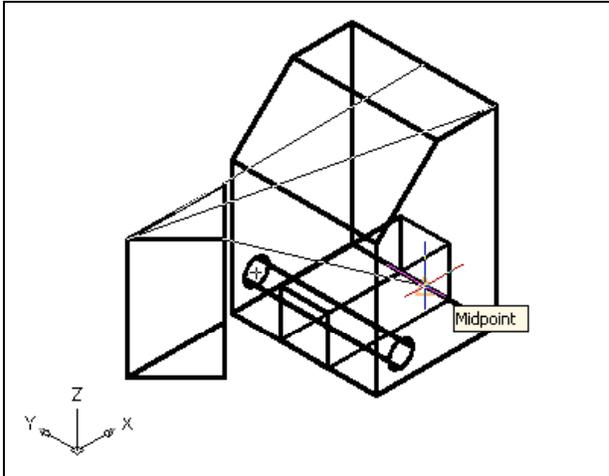


**Figure 8.26 – Align Point One**
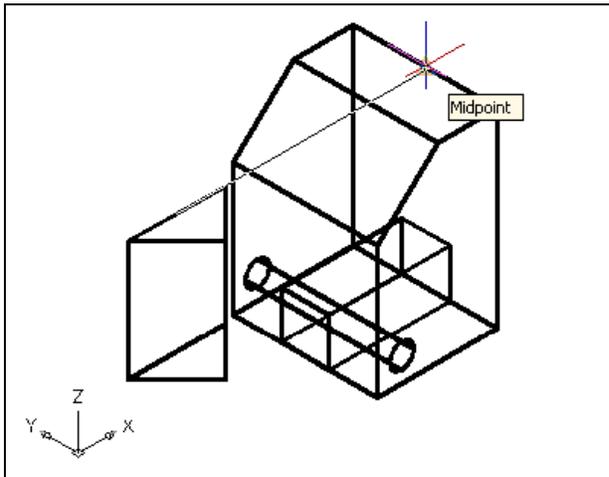
The first alignment point brings the two points selected together. Of the three choices for alignment, you can only guarantee that the first group will unite exactly. Unless the solids are exactly the same size this will always be true. This makes the first selection the most important, since you will 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 – Align Point Two**

The second set of points 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 – Align Point Three

The last group of points you will select will 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

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



Figure 8.29 – The Aligned Wedge
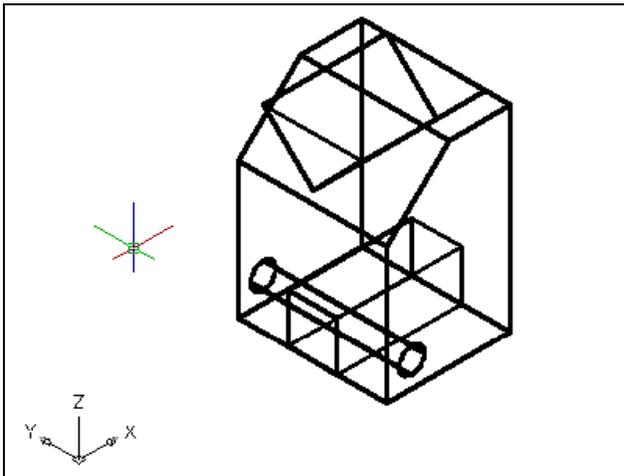
The AutoCAD program will prompt you with "**Select solids and regions to subtract from**". Select the larger master solid and the system will respond with "**1 found**". You need to hit Enter to proceed to the next step, which is to "**Select solids and regions to subtract**". Pick the box and triangular solid and the software application will counter with "**2 found**". Hit Enter and a region exactly the shape of the box and wedge disappears from the larger one. You will want to use the Hide command to see your results better.

# Hide the 3D Solid

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In Figure 8.29, 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.
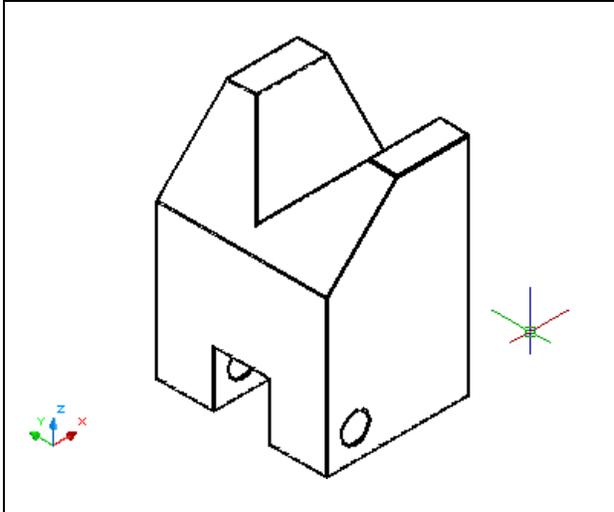
Select the Hidden tool on the Render toolbar. This is a very simple command, because moments after picking the tool, the tessellation lines representing the back of the part are removed from view as shown in Figure 8.30. If you wish to return the graphical display to the previous setup where all the tessellation lines are seen, then choose the 3D Wireframe Visual Style on the Visual Styles toolbar. The solid part 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

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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.**
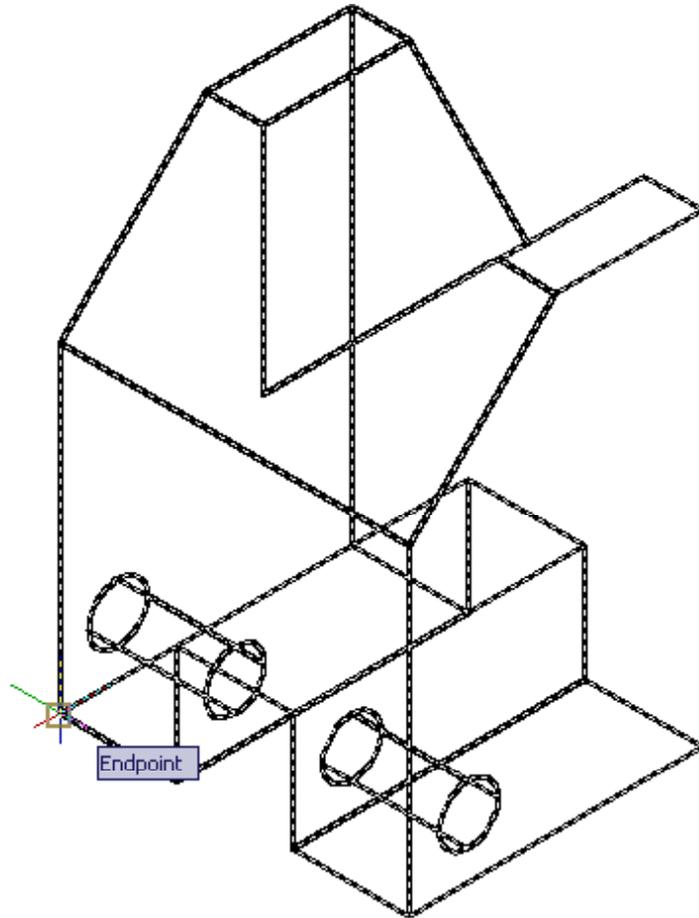
**Figure 8.31 – Moving the Finished Solid to the Origin**

## Saving the Solid Problem

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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)
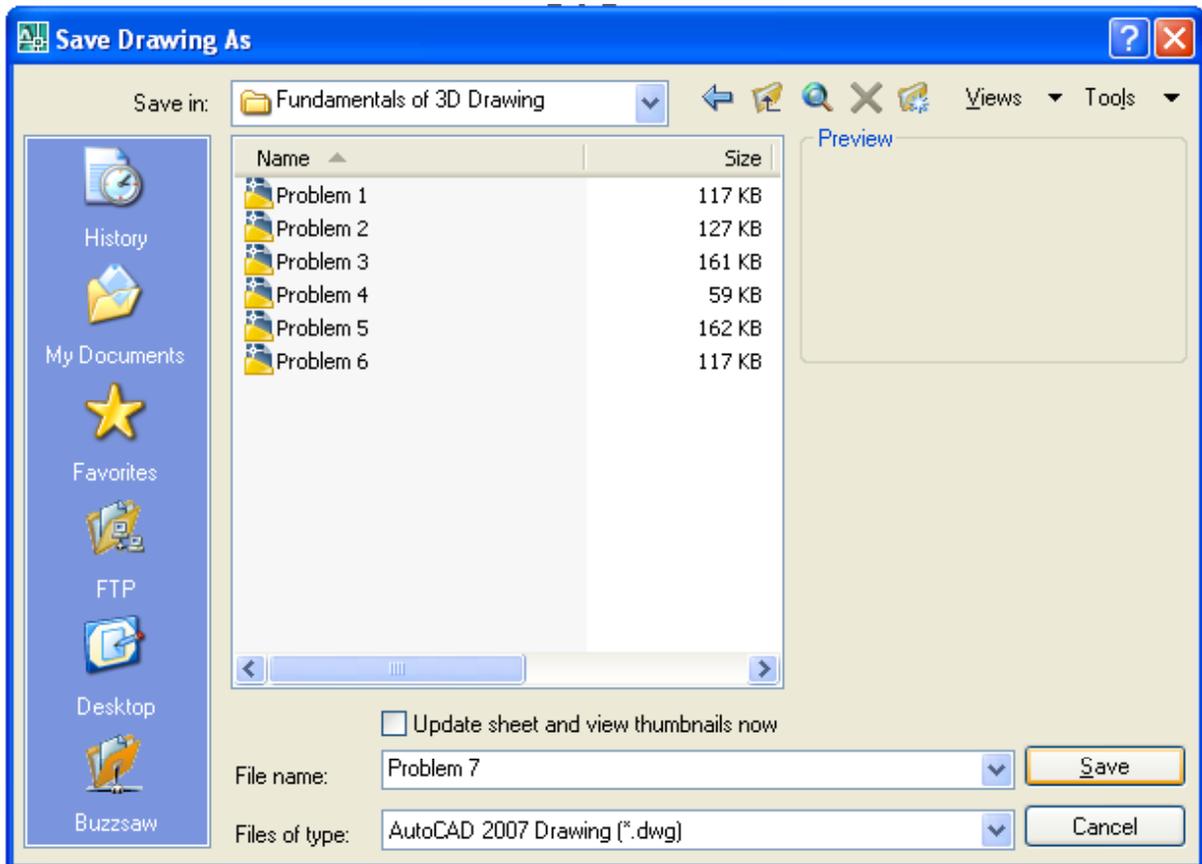
**Figure 8.32 – Saving Problem Seven**

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