C h a p t e r

9

Solid Problem Eight

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

1. Sketch of Solid Problem Eight
2. Starting a 3D Part Drawing
3. Creating a New Layer in an AutoCAD Drawing
4. Modifying How the UCS Icon is Displayed
5. Constructing a Solid Perimeter Using the Line Command
6. Adding Circles to the Detail
7. Moving a Circle Inside the Perimeter
8. Using the Copy Command to Create Another Circle
9. Using the Offset Command to Create a New Line
10. Using the Arc Command to Add a Circular Segment
11. Creating Polylines Using the Edit Polyline Tool
12. Drawing Solid Boxes
13. Extruding a Solids from a Closed Polyline and Circles
14. Drawing a Solid Cylinder
15. Moving Solids into Position on the Master Solid
16. Subtracting 3D Solids from the Master Solid
17. Aligning Two Solids
18. Union the Solids
19. Add a Bevel Edge Using the Chamfer Command
20. Hide the Solid
21. Moving the Solid to the Origin Point
22. Saving the Solid Problem

Sketch of Solid Problem Eight

_____

You will start the eighth solids problem with a rough sketch showing the dimensions of each detail. This part is not symmetrical across any of the axis. In this exercise, a strategy you will continue to use is to draw and extrude closed Polylines that contain the most complex shape of the component and then use simple geometric solids to subtract detail. In problem Eight, the vertical solid contains an arc drawn into the shape. You will place a two circle into that shape. After extruding the Polyline and Circular extruded solids, subtract the cylinders. The second horizontal shape will have a Box and Cylinder detail subtracted. You will learn another new 3D tool, the Chamfer command, where you will place a 45 degree broken edge on the solid finishing the problem as shown in Figure 9.1.
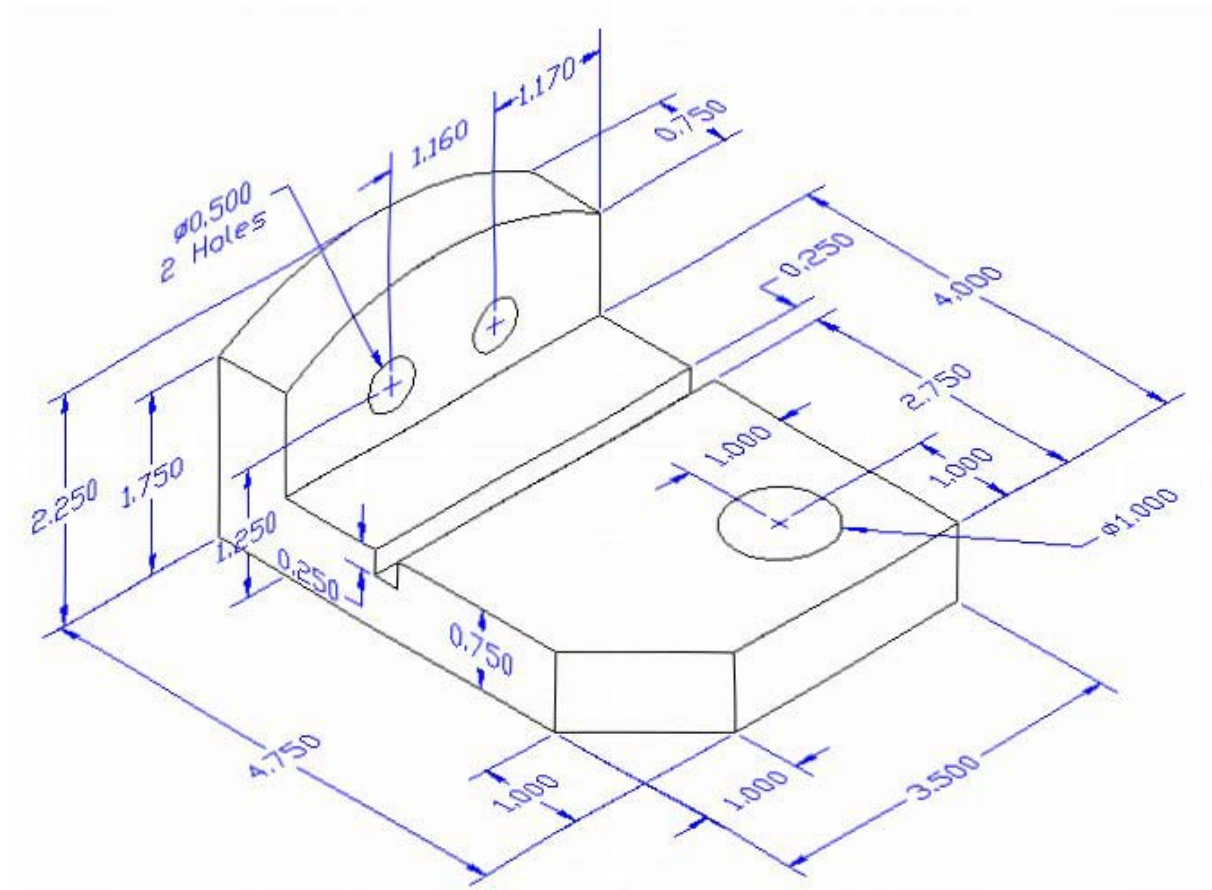

Figure 9.1 – Problem Eight Sketch

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 Eight, you will begin the 3D drawing by selecting the Mechanical template. In Figure 9.2, 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.
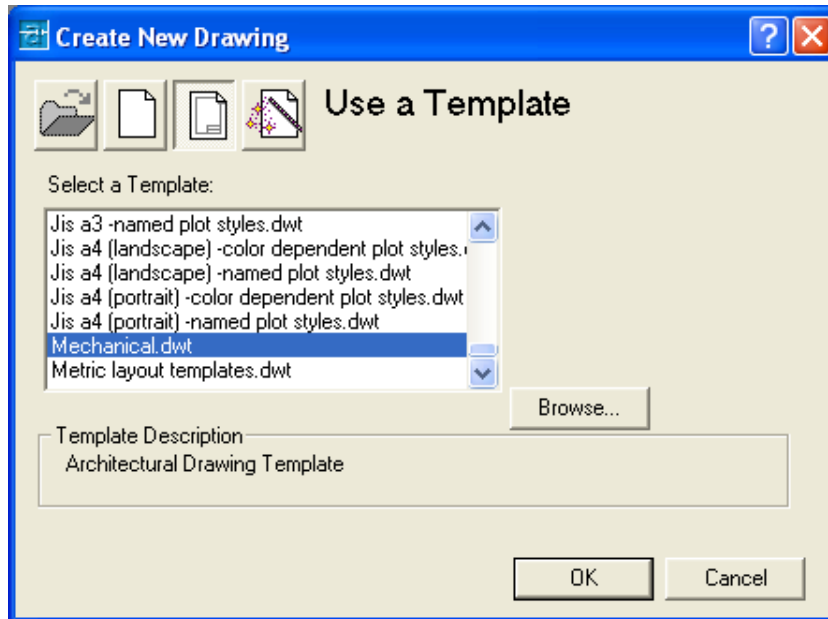

Figure 9.2 – Starting the Drawing Using the Metric Template

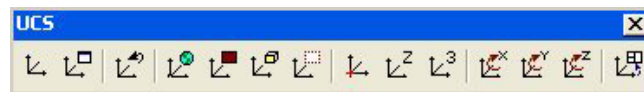# Modifying How the UCS Icon is Displayed

_____

Figure 9.3 - 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 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 9.4 and press the OK button to have the system variable change. Now you can start to create the solid.
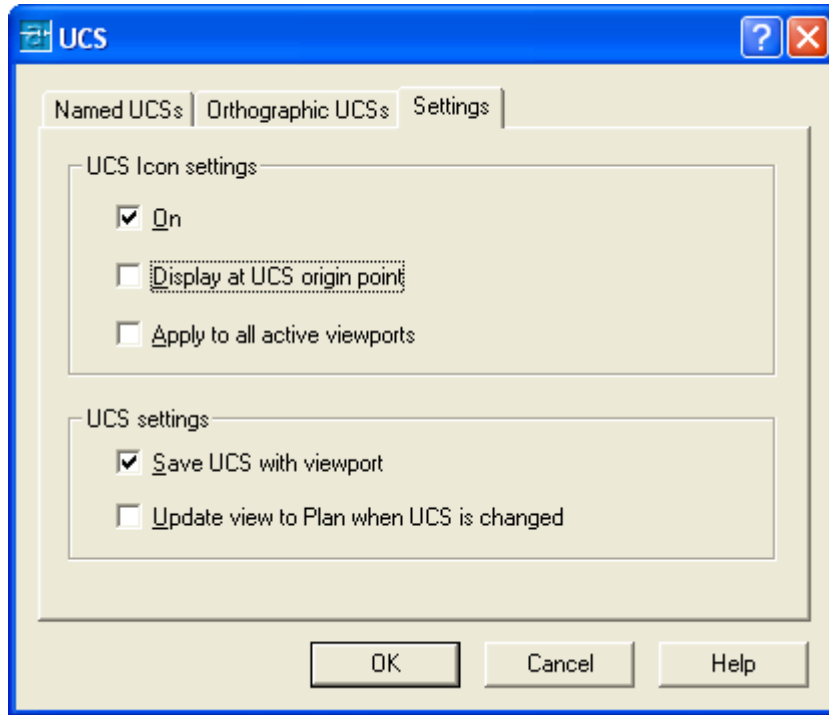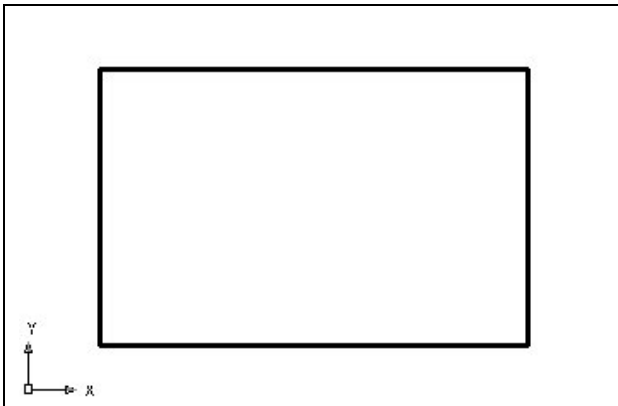
Figure 9.4 - The UCS Window, Settings Tab

Constructing a Solid Perimeter Using the Line Command

_____

First, you need to construct a 3.5 by 2.25 rectangle by selecting the Line tool from the Draw toolbar. With your mouse, select any point at the lower right side of the graphical display.

Even while in the Line command, continue with the Ortho mode On to keep the lines perfectly horizontal or vertical when drawing. You then move the line on the graphic screen to the right and type "**3.5**" and **Enter** at the keyboard. To draw the vertical line, move the mouse upward, dragging the line and type "**2.25**" and **Enter**. Drawing the third line, you move the mouse to the left, dragging the line and type "**3.5**" and **Enter**. To close the rectangle, you just type "**C**" and **Enter**.

Figure 9.5 – Drawing a Perimeter

Notice that you did not draw a Polyline, since you will need to add some additional details to the perimeter. In the next few steps, you will add and move a circle into position. Then you will copy the circle. You will add a construction line, which will allow you to place a 3-point arc in on the perimeter. After two lines are erased, you can edit the lines and turn the circular segment and three lines into a Polyline.

Adding Circles to the Detail

Next, select the Circle command on the Draw toolbar. Place the mouse aperture at the upper left Endpoint as shown in Figure 9.6 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 9.7
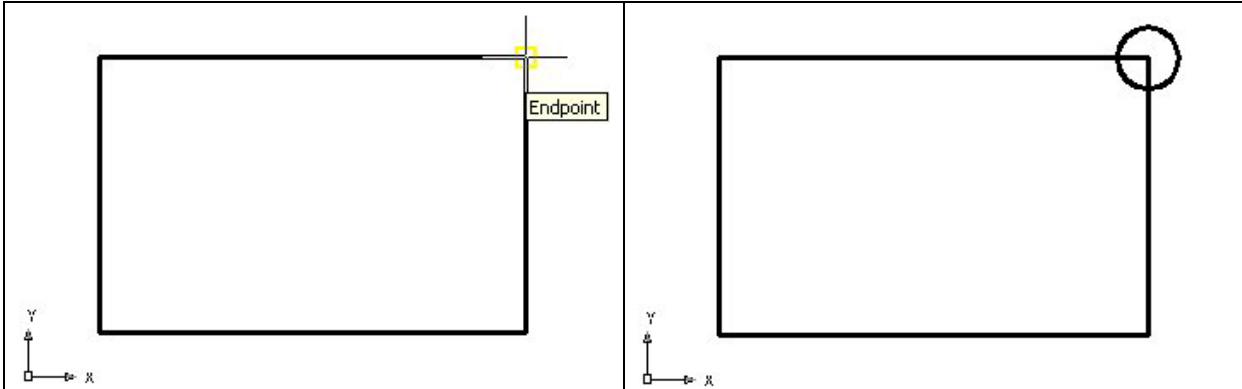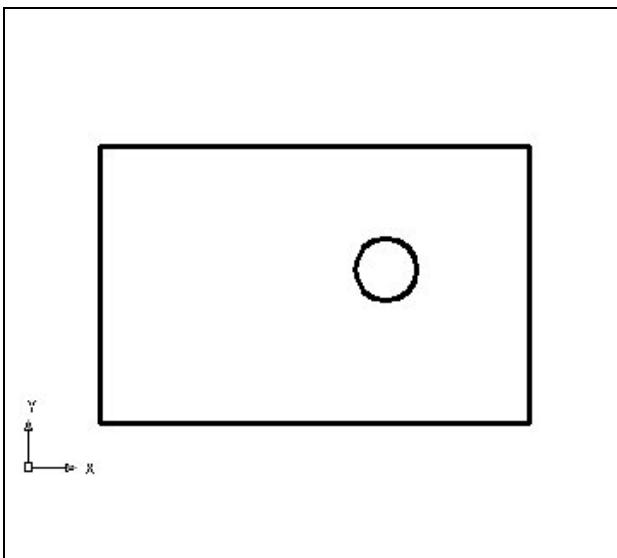
Figure 9.6 – Adding Circles For The Solid   **Figure 9.7 – Adding Circles For The Solid**

Moving a Circle Inside the Perimeter

Next, 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.
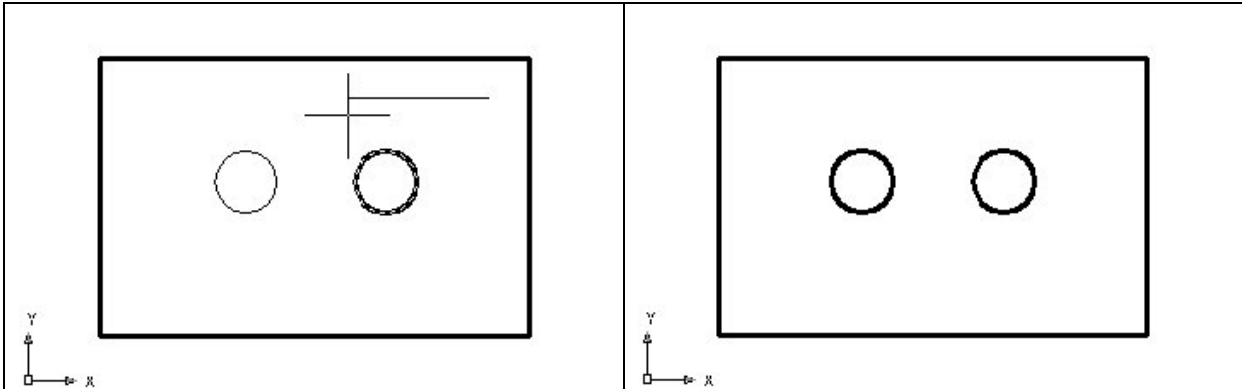
Once you choose the Move command, the prompt will be "Select Objects". Pick the circle and the command line will respond with "**1 found**", so just press Enter to go the second part of the command. Next, you need to "specify base point or displacement" on the graphical display. Pick your base point anywhere on the display. Type **Enter** to proceed to the "second point of displacement" as input **@-1.0,-1.17** which will move the entity a negative distance of 1.0 on the X-axis, a negative 1.17 in the Y-axis and when leaving the Z coordinate off, the default is 0 in the Z-axis. The circle will be as shown in Figure 9.8.

Figure 9.8 – Positioning the Circle

Use the Copy Command to Create Another Circle

_____

The Copy command is actually a three-dimensional function in the modern AutoCAD application software program. Like the Move tool, you can use the Copy function to relocate any entity to another point (X,Y,Z) in Model Space.


Figure 9.9 – Copying the First Circle   **Figure 9.10 – Positioning the Circle**

The Copy command is separated into two parts, the first being the selection process and the second being the function of the command itself. To copy an AutoCAD entity such as the circle, you select the Copy tool on the Modify toolbar. Once you choose the Copy command, the prompt will be "Select Objects". You can do this by placing the pick box that now has replaced the aperture cursor on the 0.5 diameter circle and select the entity. 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 on any blank space on the graphical display, not on top of any entity, because the Object Snaps can interfere with the process. For the "second point of displacement", push the circle to the left as shown in Figure 9.9. Type 1.16 while orienting the circle to the left and the second circle appears now 1.16 to the left as shown in Figure 9.10.

Using the Offset Command to Create a New Line

_____
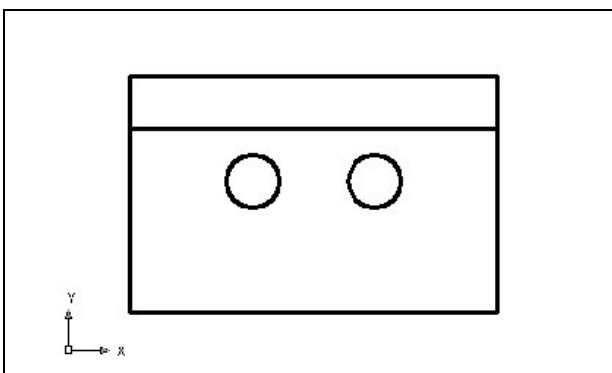
To create a new line 1.75 above the bottom line you will use the Offset command, which will allow you to create a line without selecting points to define the starting and endpoints. Choose the Offset tool on the Modify toolbar. After picking the Offset tool, the command line will prompt you for the offset distance which you will type **1.75** and **Enter.**

Figure 9.11 – Offset the Bottom Line, 1.75

Next, select the line you wish to offset which is the bottom horizontal line. After selecting the line, pick the direction of movement, which is above the bottom horizontal line. A new line will appear 1.75 above the bottom line as shown in Figure 9.11. You need to **Enter** to leave the Offset command after placing the new line.

## Using the Arc Command to Add a Circular Segment

_____

This problem has a few new functions to learn and you will need to become skilled at creating a circular segment or arc in the drawing occasionally. Prior to the introduction of the Fillet command, AutoCAD application software users would use the Arc command to make their arcs on the drawing. The Fillet command will join two lines and place the circular segment at the precise radius without much construction problems. Therefore, computer-aided designers select the Fillet tool rather than the Arc command. You can spend hours learning the many variations of the Arc command, but today you will learn how to draw a simple three point circular segment.
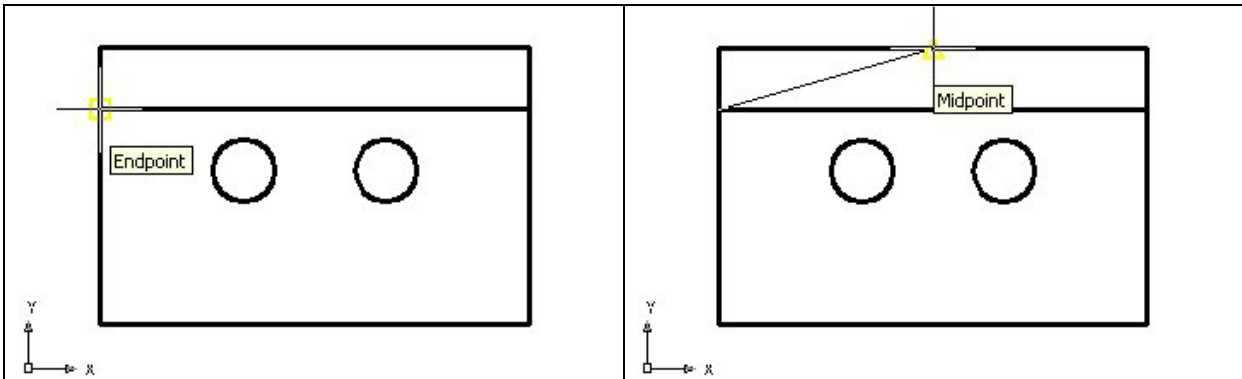
Figure 9.12 – The First Point of the Arc   Figure 9.13 – The Second Point of the Arc

Pick the Arc tool on the Draw toolbar. As shown in Figure 9.12, pick the Endpoint on the offset line just drawn. Next, using the Midpoint object snap, select the center of the top line as shown in Figure 9.13. After the second selection, you can see a light tracing of the arc on the graphical display, and the last pick will be the Endpoint on the right side of the offset line as shown in Figure 9.14. Go ahead and use the Erase tool on the Modify toolbar to remove the offset and top lines as shown in Figure 9.15.
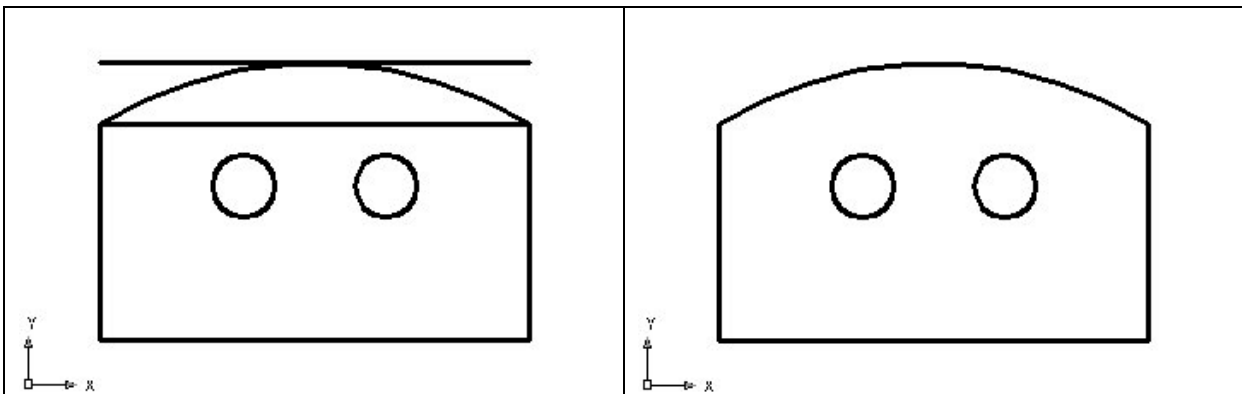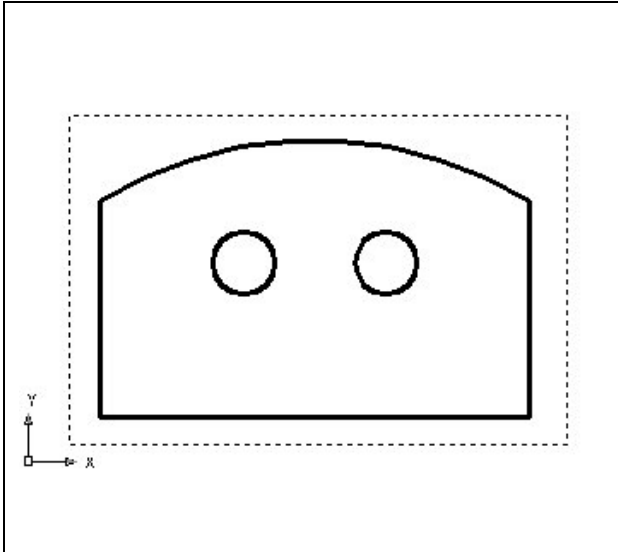

Figure 9.14 – The Third Point of the Arc   Figure 9.15 – Erase Two Lines

Creating Polylines Using the Edit Polyline Tool

_____

To merge the three lines and arc segments together you need to use the Edit Polyline tool.

Select the Edit Polyline tool on the Modify II toolbar and the command line on the AutoCAD application will prompt you to "**select objects**". Pick to the upper right of the graphical display as shown in Figure 9.16 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 9.16 – Polyline Edit the Perimeter

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.

Drawing Solid Boxes

_____

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 **3.5** and Enter for the X coordinate. The second prompt is "Specify Width" where you will type a **4.75** for the **y** coordinate. The third prompt is "Specify Height" where when you type **0.75** and Enter for the **Z** coordinate. The second box will appear as shown in Figure 9.17.

Enter to repeat the Box command. Pick the lower top surface of the solid box as your starting point. 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 **3.5** and Enter for the X coordinate. The second prompt is "Specify Width" where you will type a **0.25** for the **y** coordinate. The third prompt is "Specify Height" where when you type **-0.25** and **Enter** for the **Z** coordinate. The second box will appear as shown in Figure 9.18.
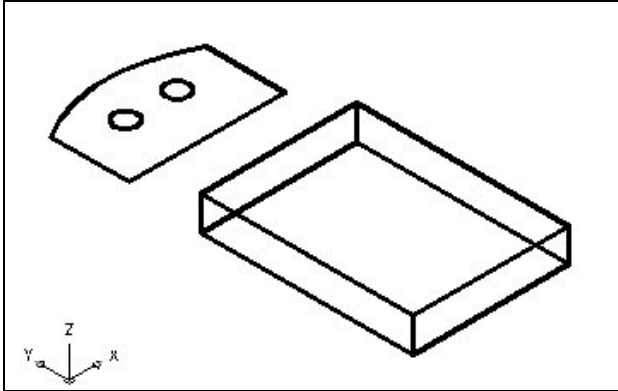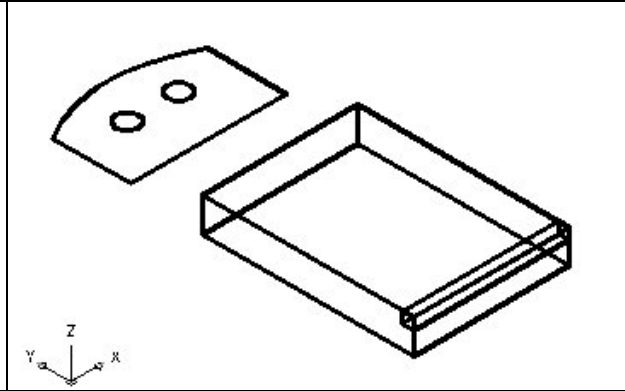

Figure 9.17 – Draw a Solid Box   **Figure 9.18 – Draw a Second Solid Box**

Extruding a Solid from Closed Polylines and Circles

_____

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.

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 "**3 found**" as shown in Figure 9.19. Press **Enter** to specify the height or path of the solid. You will type "**0.75**" for the height of the extrusion and for the angle of taper; you will **Enter** to accept the zero degree default. Three 3D solids will appear in the graphical display as shown in Figure 9.20.
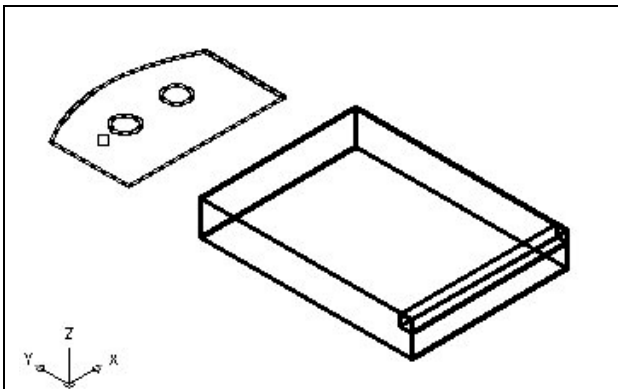

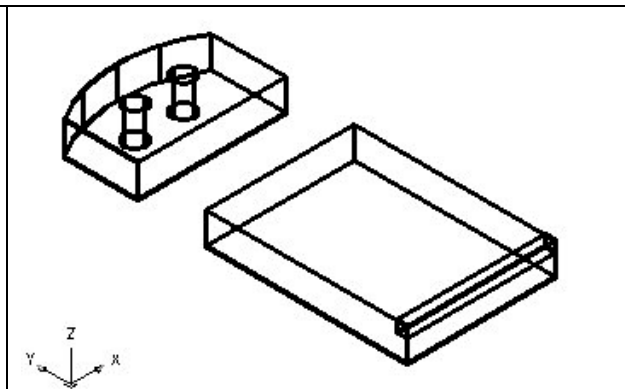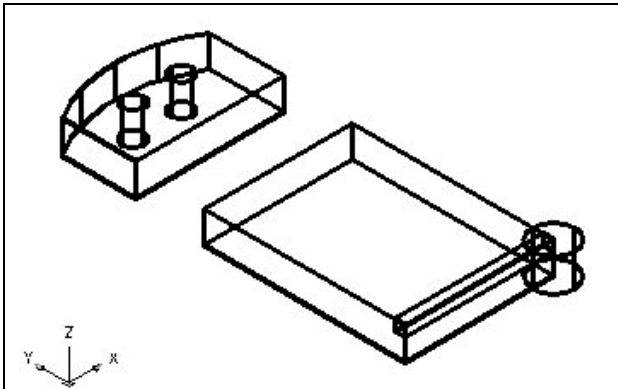**Figure 9.19 – Select Polyline and Circles**   **Figure 9.20 – Extruding Two Entities**

Drawing a Solid Cylinder

_____

You need to add a 1.0 diameter hole in the master solid, so you need to use the Cylinder command to create the feature.

On the Solids toolbar pick the Cylinder tool. Specify the center point for the base of the cylinder as the lower right Endpoint on the solid box. You need to type "**D**" for diameter and **Enter** to input the sketch's measurement of **1.0**. **Enter** to the next prompt, which is "specify the height of the cylinder". Key in **0.75** for the elevation of the can shaped solid and **Enter**. The cylinder will appears in the graphical display as shown in Figure 9.21.

Figure 9.21 – Adding a Solid Cylinder

Moving 3D Solids into Position on the Master 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.
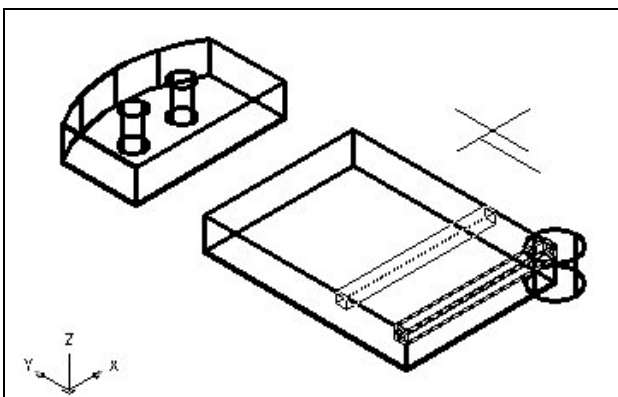

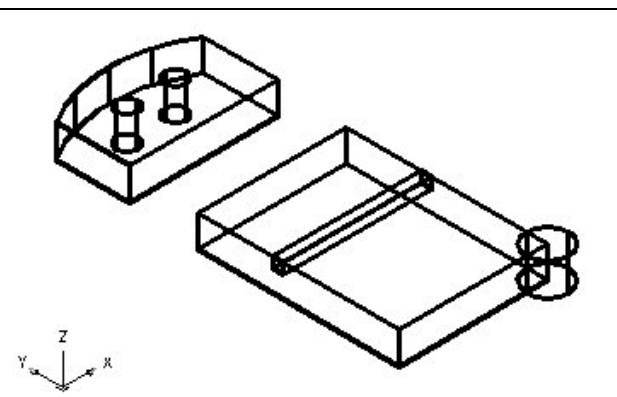**Figure 9.22 – Moving the Box**   **Figure 9.23 – Moving the Cylinder**

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" anywhere on the graphical display. For the "second point of displacement", you need to type 2.75 while orienting the solid box to the left as shown in Figure 9.22.

To move the solid cylinder, **Enter** to repeat the Move command. Pick the solid cylinder and the command line will respond with "1 found", so just press **Enter**. Next, you need to "specify base point or displacement" anywhere on the graphical display. For the "second point of displacement", you need to type in the relative coordinate of "**@-1.0,-1.0**" to move the cylinder 1.0 in the negative X direction and 1.0 in the negative Y direction as shown in Figure 9.23.

Subtracting 3D Solids from the Master Solids

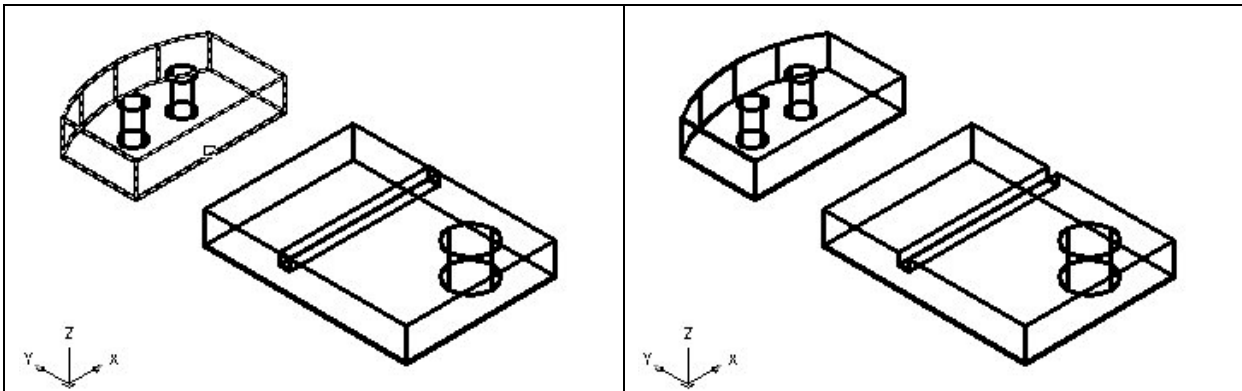To remove the cylinders from the master solid, select the Subtract tool on the Solids Editing toolbar.

**Figure 9.24 – Subtract Solids**   **Figure 9.25 – Subtracting Solids**

Then the AutoCAD program will prompt you with "Select solids and regions to subtract from". Select the left 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 two cylinders 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 9.24.

Press **Enter** to repeat the Subtract command. Select the right 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 cylinder and the box 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 9.24.

Aligning Two Solids

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.
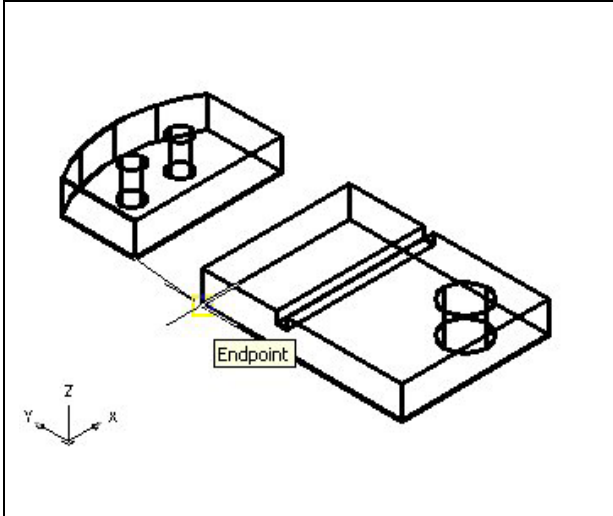
Figure 9.26 – Align First Points

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 9.26, the first point of alignment is the Endpoint on the bottom of the arched solid. The destination is the Endpoint on the bottom left tessellation line of the master solid. A temporary line appears on the display showing the connection.
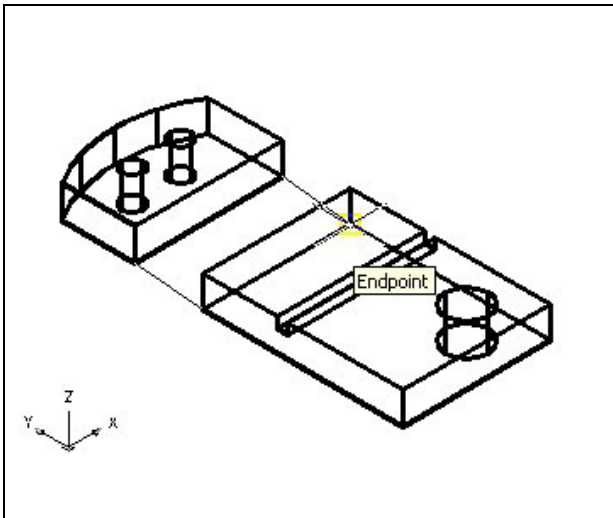
Figure 9.27 – Align Second Points

The second set of point will signify the two dimensional line of alignment. You can see in Figure 9.27 that the first pick is the lower right corner of the arched solid. The second selection is the lower top corner of the master solid. The sides are the same measurement, but they can be different in length to have these two tessellation lines 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.
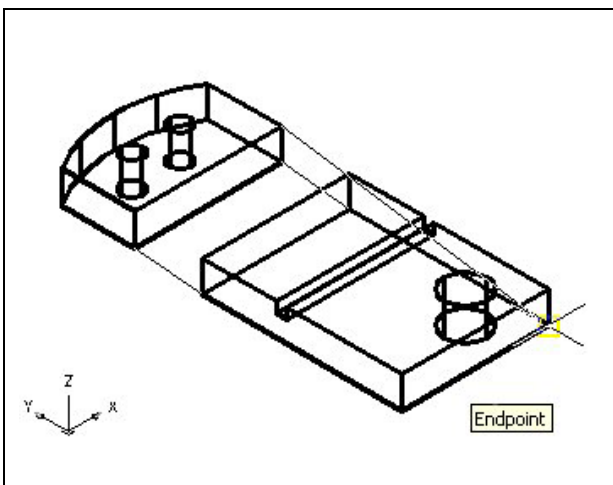
Figure 9.28 – Align Third Points

The last group of points you will select is to align the planes of both solids. You can see in Figure 9.28 that the first pick is the upper right corner of the arched solid. The second selection is the lower right Endpoint of the solid box. 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.

Union 3D Solids

_____

For the next modification of a 3D solid shape in AutoCAD, you need to learn the second Boolean command, Union. With the Union command, you will learn how to join one shape or more shapes to each other.
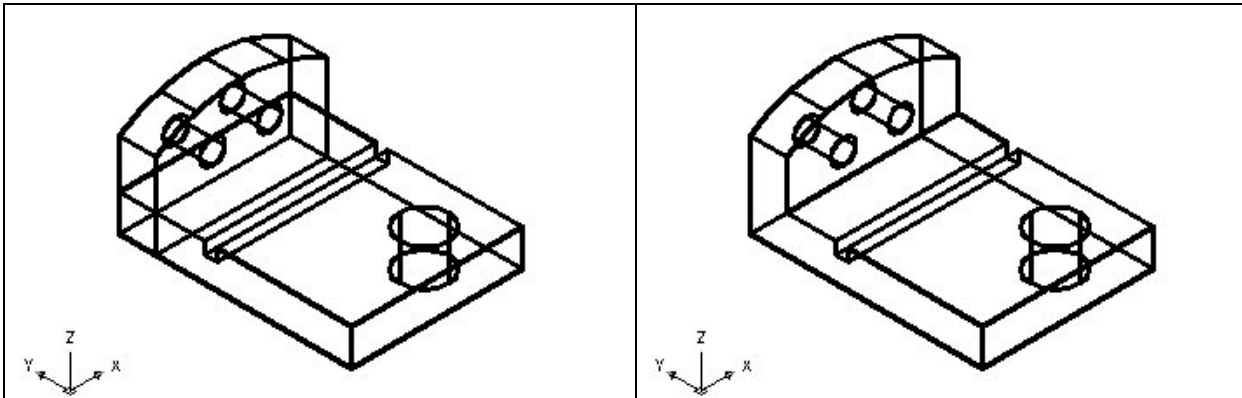

Figure 9.29 – Before Joining Two Solids   **Figure 9.30 – After Joining Two Solids**

To add the two solids together, select the Union tool on the Solids Editing toolbar. Then the AutoCAD program will prompt you with "Select objects". Select both shapes and the system will respond with "2 found". Enter and now all the solids become as one entity as shown in Figure 9.30. You can see in the graphical display the difference in that the tessellation lines outlining the two solids and now the overlapping tessellation lines are gone. To test your solid, select the entity with one pick of the mouse and the entire solid will become highlights.

Add a Bevel Edge Using the Chamfer Command

_____

To add a beveled edge to a corner, you can use the Chamfer command to create the feature rather than adding a triangle and subtracting the detail.
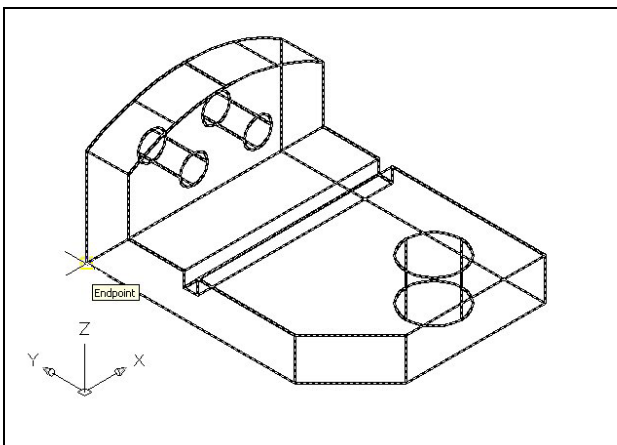

Pick the Chamfer tool on the Modify toolbar and select the lower vertical corner on the edge of the box. You now type "**D**" for distance and "**1.00**" for the offset distance. **Enter** and type "**D**" for distance and "**1.00**" for the second offset distance. Enter and select the vertical tessellation line. Enter and the 1.0 by 45 degree chamfer will appear on the edge of the master solid as shown in Figure 9.31. You will commonly use this technique to improve the process of fitting closely dimensioned parts together.

Figure 9.31 – Chamfer the Edge

Hide the 3D Solid

In Figure 9.31, 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.
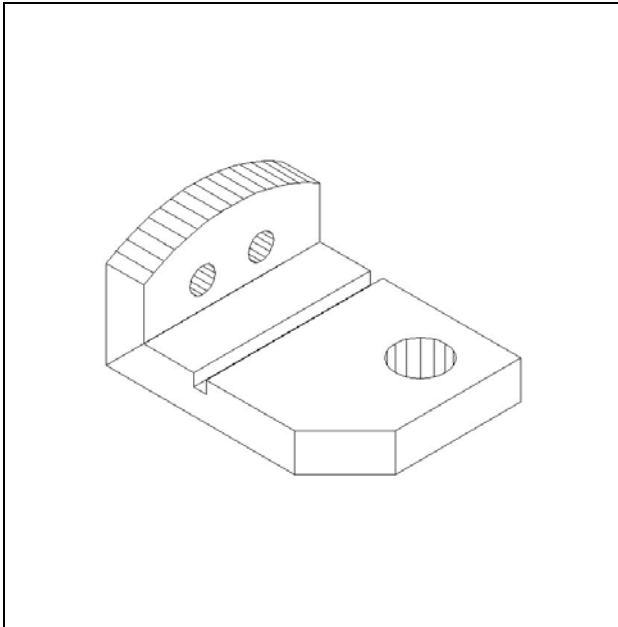
Select the Hidden tool on the Shade toolbar. This is a very simple command, because moments after picking the tool, the tessellation lines representing the back of the part are removed from view. Notice in Figure 9.32 that the UCS icon changes from the thin lined and arrowed icon to a colorful image. 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 9.32 – Hide the Solid

Moving the Solid to the Origin Point

To move problem Eight 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 9.33 at the lower left hand side of the problem eight, which will become the problem eight insertion point into an assembly drawing. At the "second point of displacement", type 0,0,0 and Enter. Problem Eight will move to the new origin point. There is more than one way to do many actions in AutoCAD, but the 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.
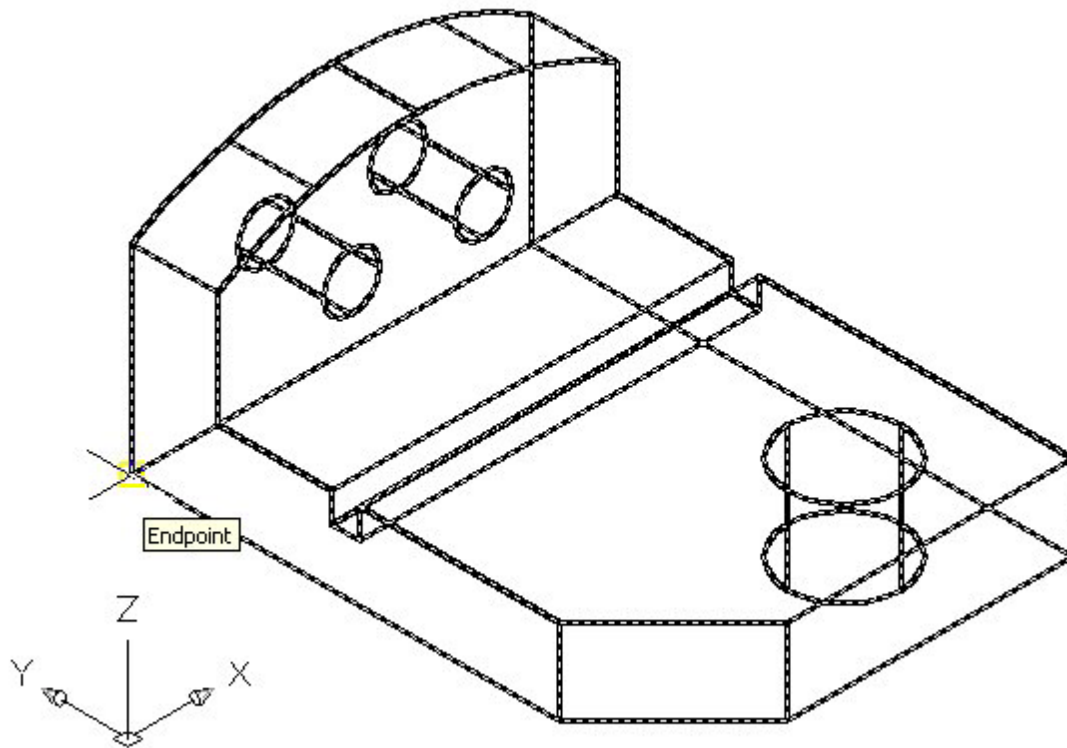
Figure 9.33 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Problem 8 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 8" and press the Save button to save the drawing. (See Figure 9.34)
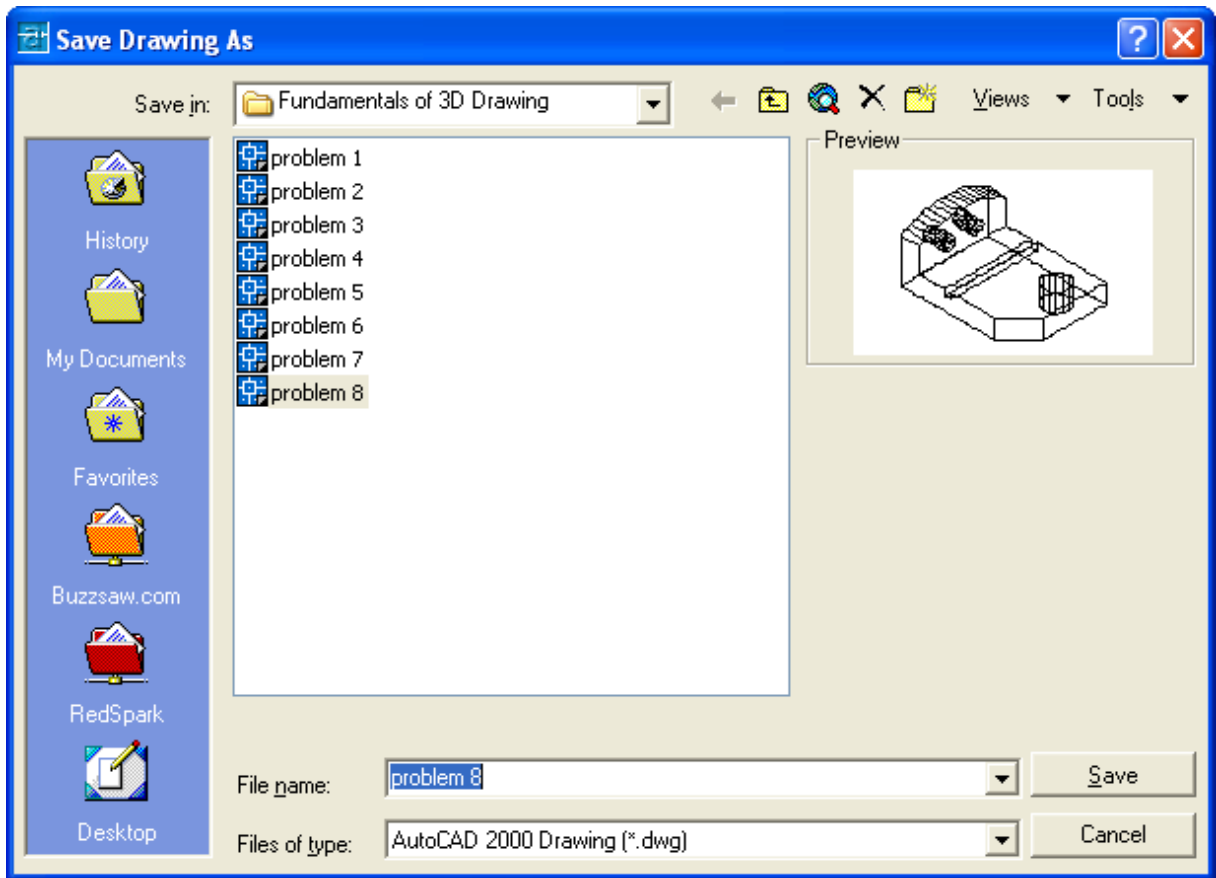
Figure 9.34 – Saving Problem Eight

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