

Solid Problem Eight

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

- Sketch of Solid Problem Eight
- Starting a 3D Part Drawing
- Creating a New Layer in an AutoCAD Drawing
- Modifying How the UCS Icon is Displayed
- Constructing a Solid Perimeter Using the Line Command
- Adding Circles to the Detail
- Moving a Circle Inside the Perimeter
- Using the Copy Command to Create Another Circle
- Using the Offset Command to Create a New Line
- Using the Arc Command to Add a Circular Segment
- Creating Polylines Using the Edit Polyline Tool
- Drawing Solid Boxes
- Extruding a Solids from a Closed Polyline and Circles
- Drawing a Solid Cylinder
- Moving Solids into Position on the Master Solid
- Subtracting 3D Solids from the Master Solid
- Aligning Two Solids
- Union the Solids
- Add a Bevel Edge Using the Chamfer Command
- Hide the Solid
- Moving the Solid to the Origin Point
- 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 two circles 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.

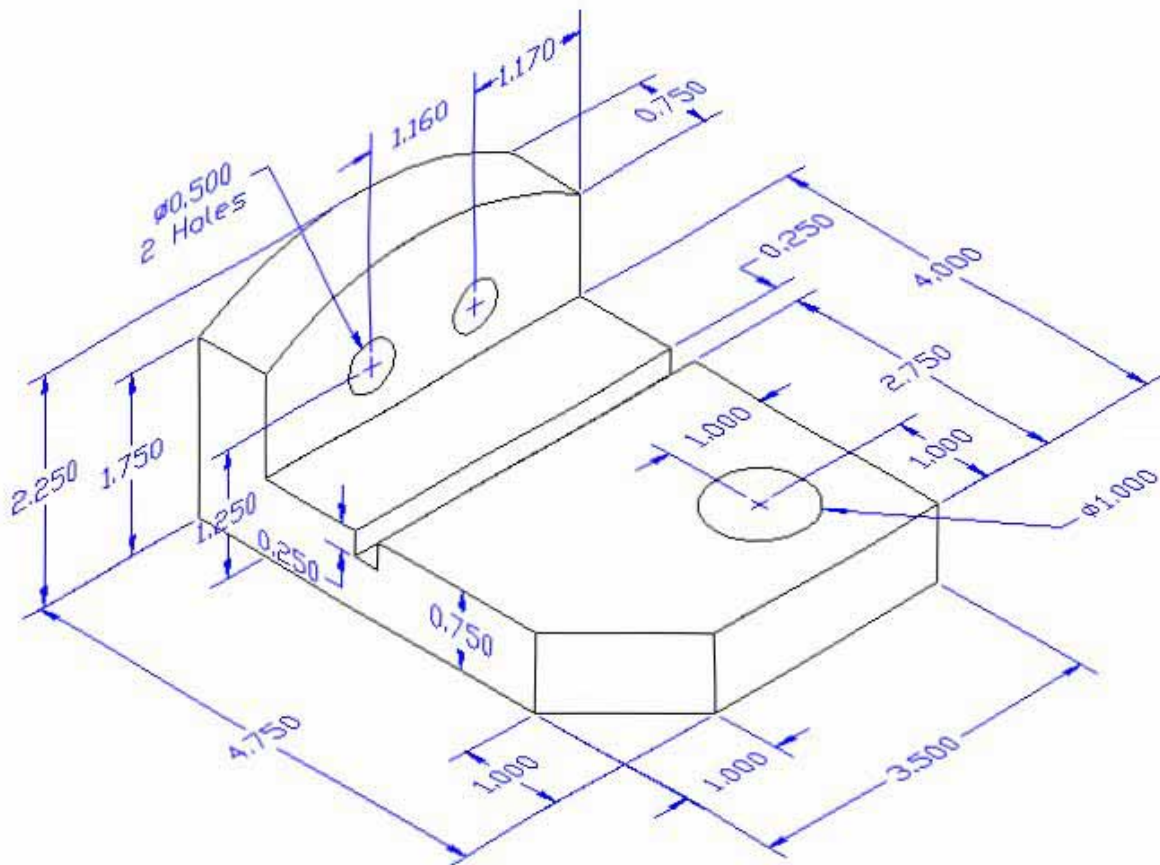


Figure 9.1 – Problem Eight Sketch

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 “Select template” is loaded by selecting the New tool on the Quick Access 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.

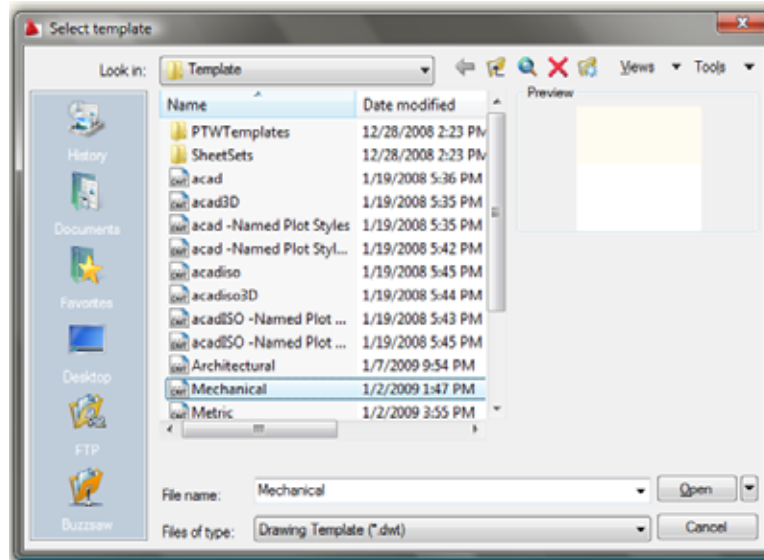


Figure 9.2 – Starting the Drawing Using the Metric Template

Modifying How the UCS Icon is Displayed

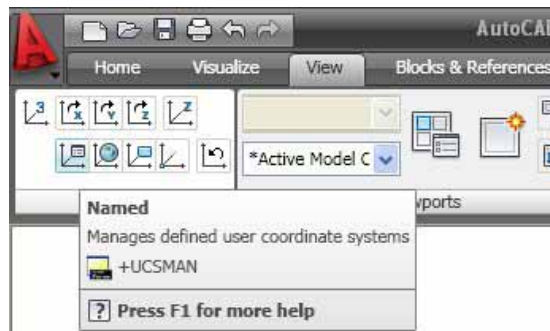


Figure 9.3 - The UCS Settings

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 Named tool on the Coordinates panel of the View tab and the UCS window will appear on 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.

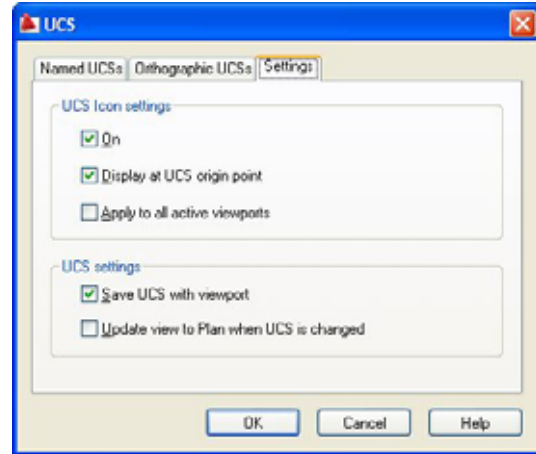
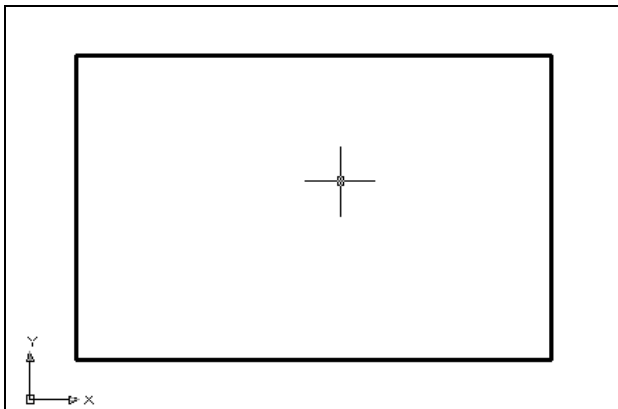


Figure 9.4 - The UCS Window, Settings Tab

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 panel. 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. 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

Select the Center, Radius command on the Draw panel. Place the mouse aperture at the upper right 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 **0.5** for the diameter and the circle will appear on the corner as shown in Figure 9.7

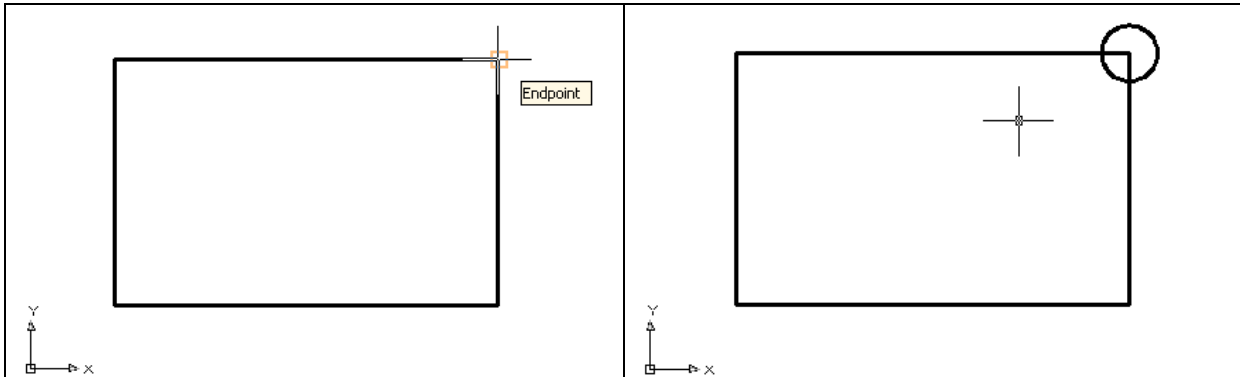
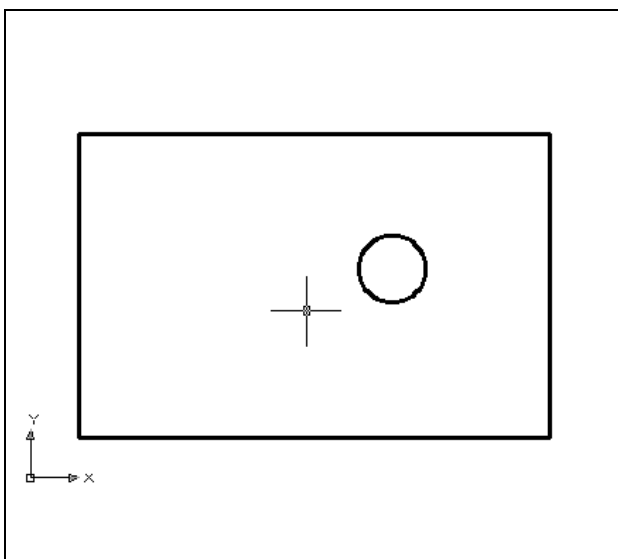


Figure 9.6 – Finding the Endpoint

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 panel to accomplish the task. To move the circle, you select the Move tool on the Modify panel.



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 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. Hit **Enter** to proceed to the “second point of displacement” and input **@-1.17,-1.0** which will move the entity a negative distance of 1.17 on the X-axis, a negative 1.0 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.

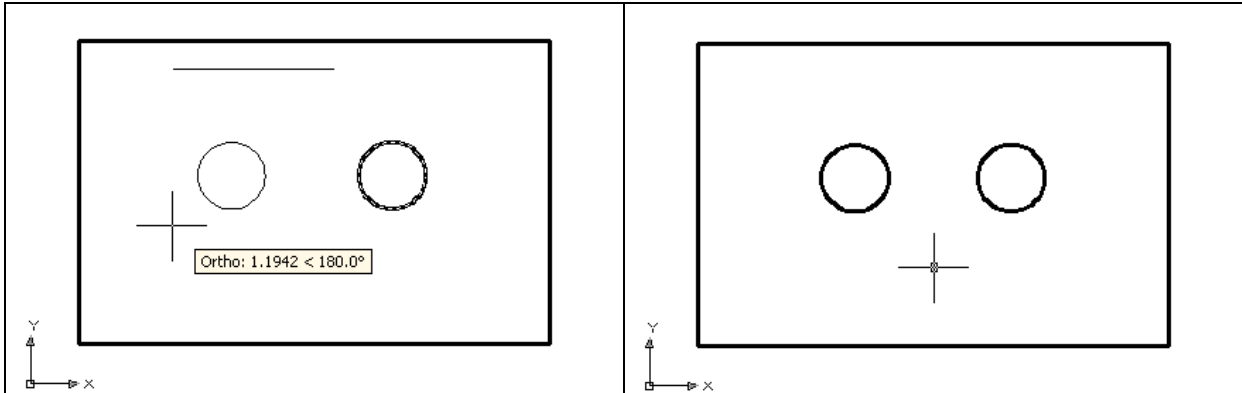


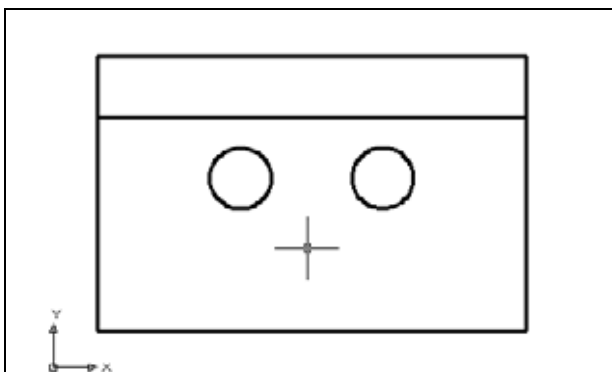
Figure 9.9 – Copying the First Circle

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 panel. 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, but 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 panel. After picking the Offset tool, the command line will prompt you for the offset distance. Type **1.75** and **Enter**.



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 hit **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 many construction problems. Therefore, many 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.

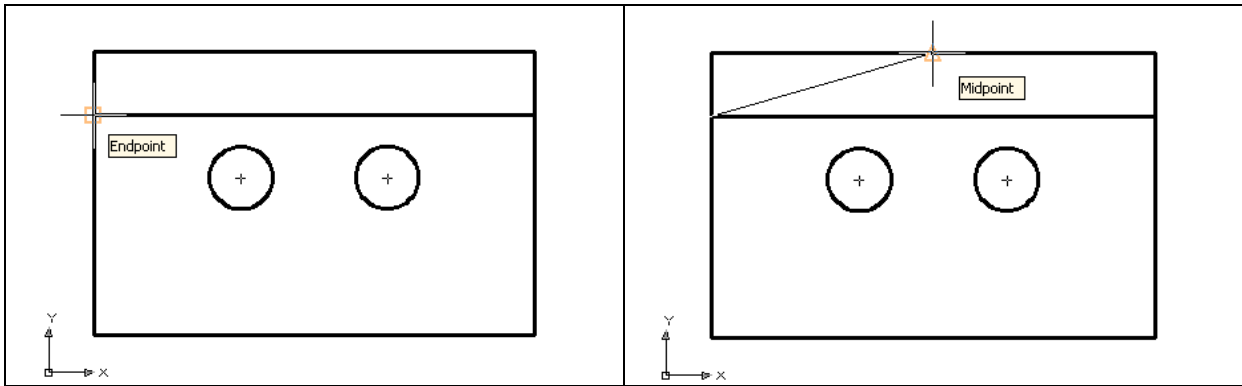


Figure 9.12 – The First Point of the Arc

Figure 9.13 – The Second Point of the Arc

Pick the 3-Point (Arc) tool on the Draw panel. 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. Use the Erase and Trim tools on the Modify panel to remove the offset, top, left, and right lines so the solid looks like the one in Figure 9.16.

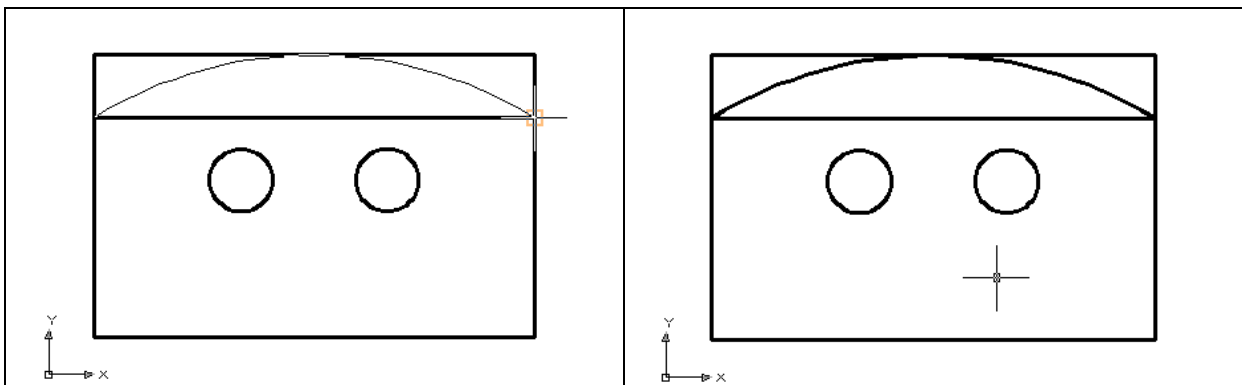

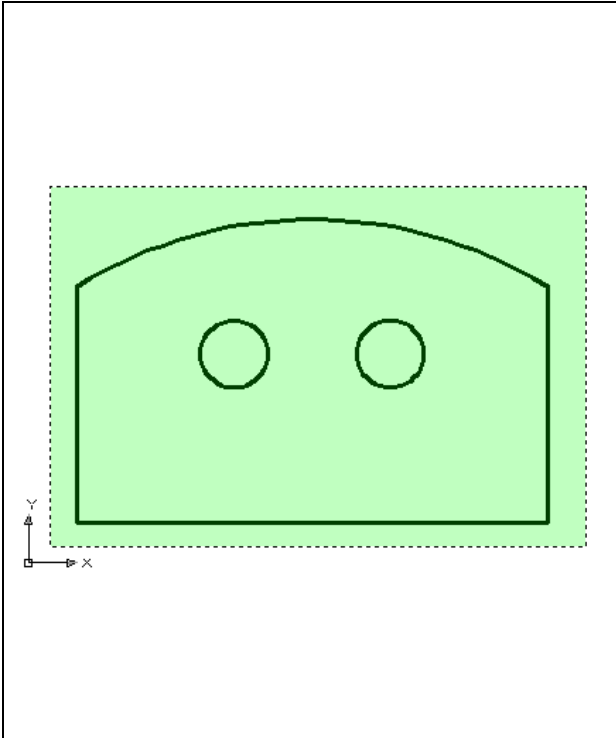


Figure 9.14 – The Third Point of the Arc

Figure 9.15 – The Finished Arc

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 beneath the drop-down arrow on the Modify panel and the command line on the AutoCAD application will prompt you to “**select Polyline or [Multiple]**”. Type “M” for multiple. Then 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>”. Hit Enter to convert a line to a Polyline and at the next prompt, type “J” and Enter for join. When asked about “fuzz distance or [Jointype],” either right click and select cancel or press **Enter** until the command is complete. Reselect the lines and all adjoining lines will become a closed Polyline.

Figure 9.16 – Polyline Edit the Perimeter

On the View panel, 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 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 Modeling panel. At the command line, a prompt asks you to specify the corner of the solid. For this box, specify 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 **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 you will type **0.75** and Enter for the Z coordinate. The second box will appear as shown in Figure 9.17.

Hit 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 you will type **3.5** and Enter for the X coordinate. The second prompt is “Specify Width” where you will type **-0.25** for the y coordinate. The third prompt is “Specify Height” where you will type **-0.25** and **Enter** for the Z coordinate. The second box will appear as shown in Figure 9.18.

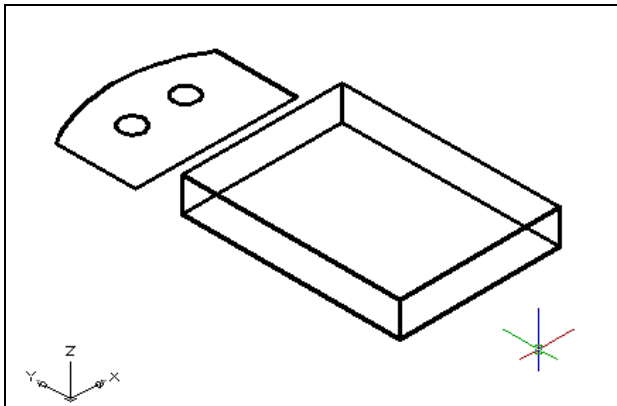


Figure 9.17 – Draw a Solid Box

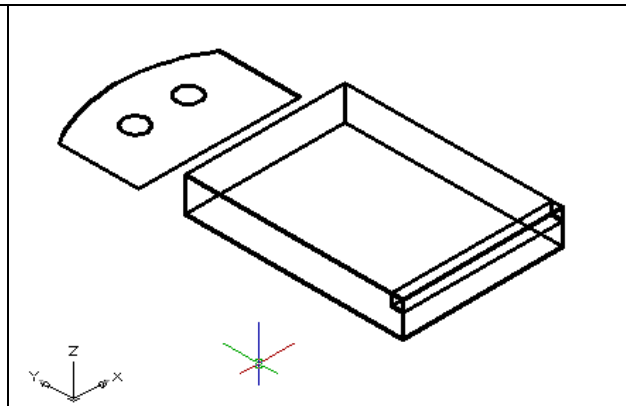


Figure 9.18 – Draw a Second Solid Box

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 Modeling panel. You now have two shapes that are single entities.



Select the Extrude tool on the Modeling panel 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. Select the solid you wish to extrude, along with both circles. 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 hit **Enter** to accept the zero degree default. Three 3D solids will appear in the graphical display as shown in Figure 9.20.

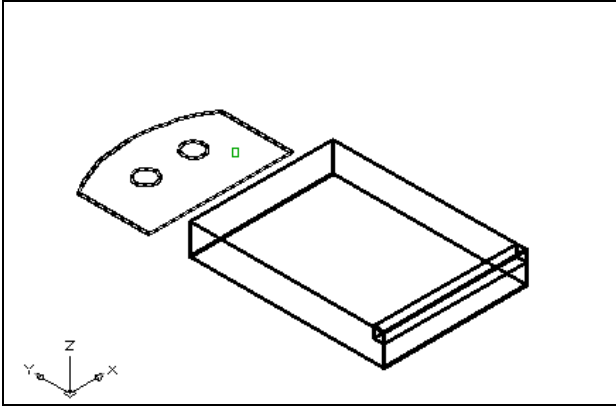


Figure 9.19 – Select Polyline and Circles

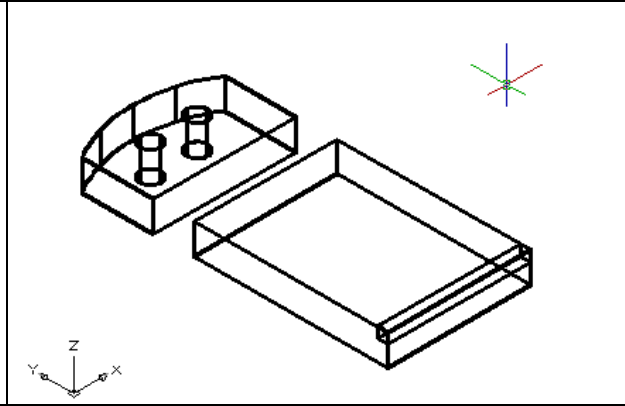


Figure 9.20 – Extruding Two Entities

Drawing a Solid Cylinder

You need to add a 1.0 diameter hole in the master solid, so you need to use the

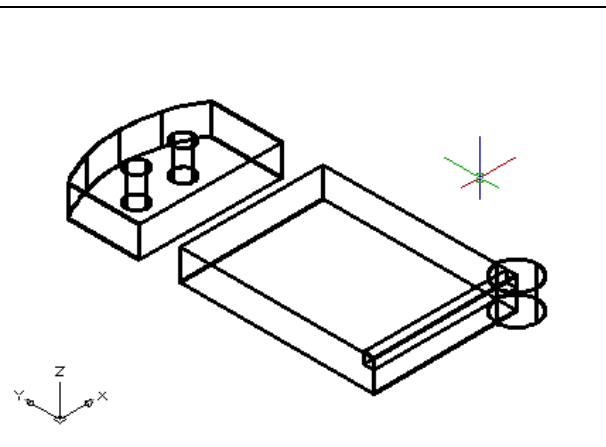


Figure 9.21 – Adding a Solid Cylinder

Pick the Cylinder tool on the Modeling panel under the drop-down arrow beneath the Box command. Specify the center point for the base of the cylinder as the lower right Endpoint on the solid box. Type “**D**” for diameter and **Enter** to input the sketch’s measurement of **1.0**. Hit **Enter** to go to the next prompt, which is to “specify the height of the cylinder”. Key in **0.75** for the elevation of the cylinder and **Enter**. The cylinder will appear in the graphical display as shown in Figure 9.21.

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 when the Z coordinate is not 0. You can use Move to relocate any entity to another point (X,Y,Z) in Model Space.



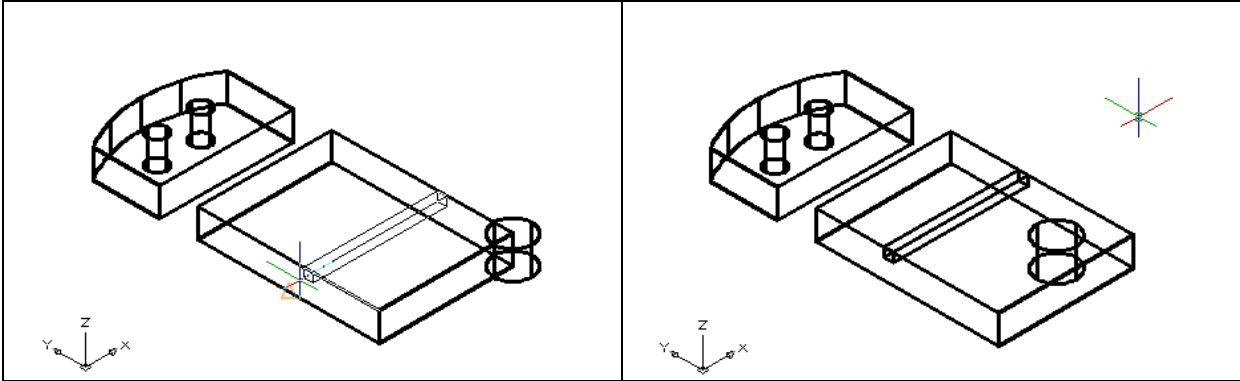


Figure 9.22 – Moving the Box

Figure 9.23 – Moving the Cylinder

To move the box, select the Move tool on the Modify panel. 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” anywhere on the graphical display. For the “second point of displacement”, 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”, 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 positive 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 Solid Editing panel.

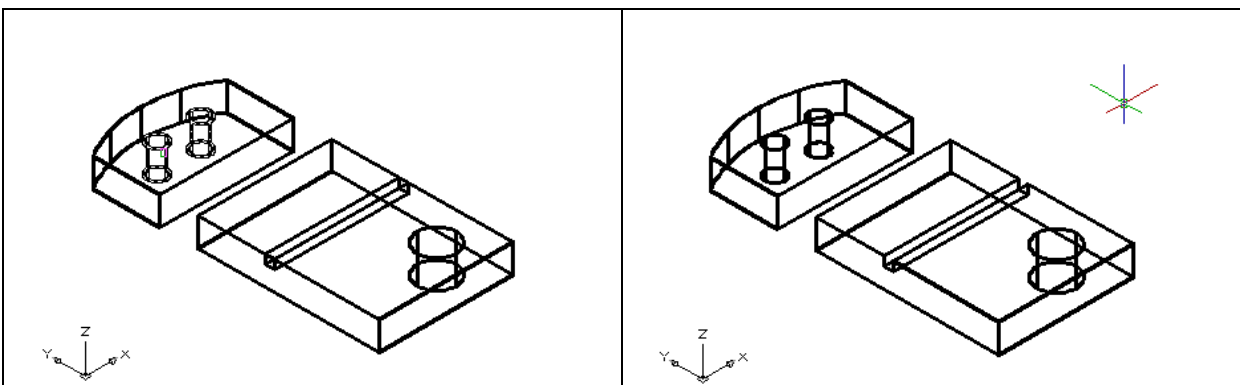


Figure 9.24 – Subtracting From Left Master Solid

Figure 9.25 – Subtracting From Right Master Solid

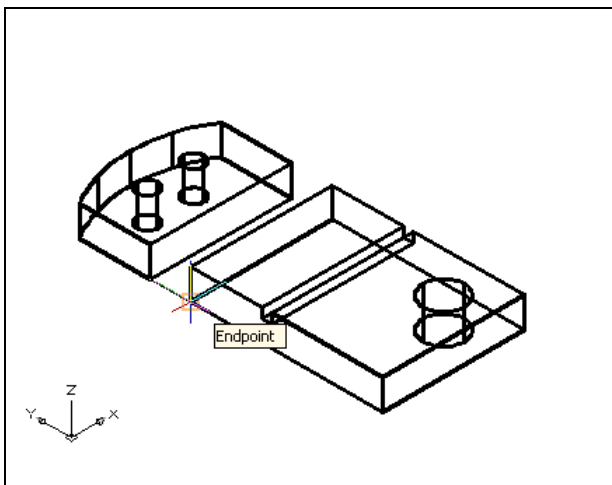
Then the AutoCAD program will prompt you with “Select solids and regions to subtract from”. Select the left master solid and the system will respond with “1 found”. You need to hit **Enter** to proceed to the next step, which is “Select solids and regions to subtract”. Pick the

two cylinders and command line will counter with “2 found”. Hit **Enter** and a region exactly the shape of the two cylinders disappears 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 command line will counter with “2 found”. Hit **Enter** and a region exactly the shape of the two cylinders is missing from the larger one as shown in Figure 9.24.

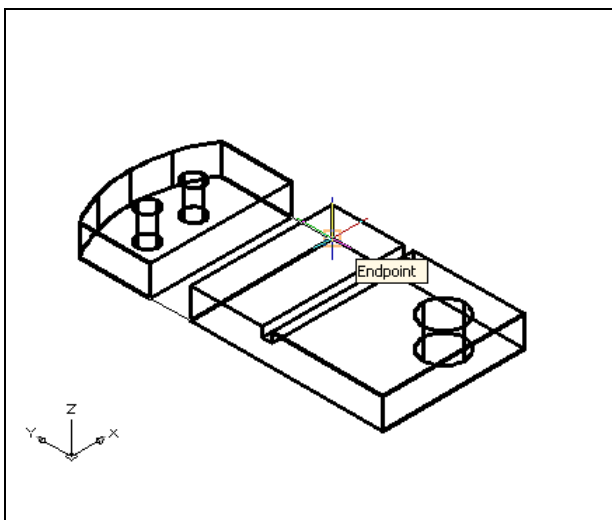
Aligning Two Solids

Select the drop-down arrow on the Modify panel and click on the Align command. At the command, “select objects”, pick the solid on the graphical display and **Enter** to proceed to the second part of the function.



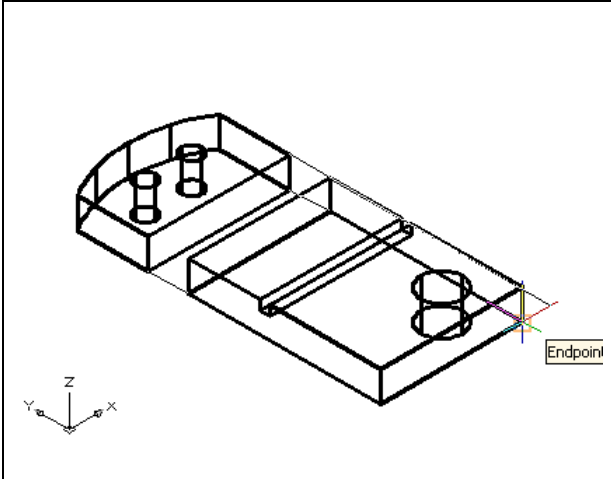
The first alignment point brings the two points selected together. Of the three choices for alignment, you can only guarantee that the first group will unite exactly. Unless the solids are exactly the same size this will always be true. This makes the first selection the most important, since you will 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 9.26 – Align First 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 9.27 – Align Second 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.

Figure 9.28 – Align Third Points

Union 3D Solids

With the Union command, you can join one shape or more shapes to each other. To bring the two master solids together, you will use the Union tool.

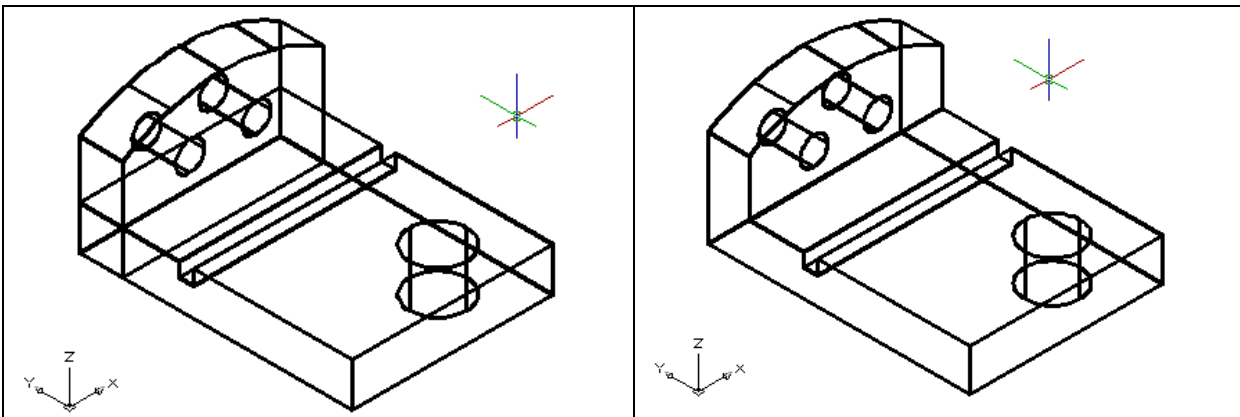


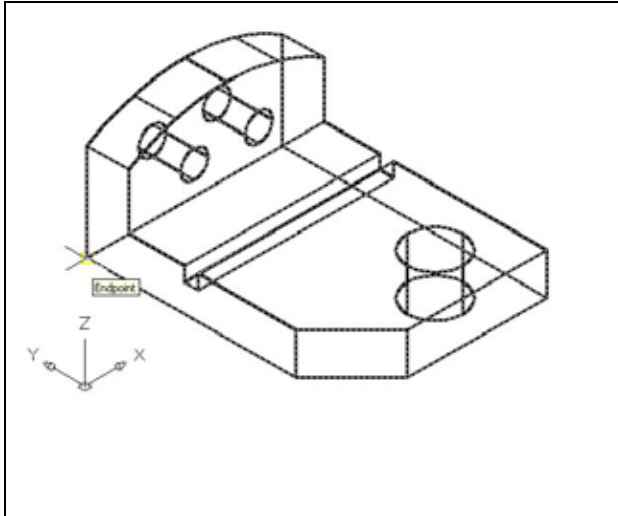
Figure 9.29 – Before Joining Two Solids

Figure 9.30 – After Joining Two Solids

To add the two solids together, select the Union tool on the Solid Editing panel. AutoCAD will prompt you to “Select objects”. Select both shapes and the system will respond with “2 found”. Hit 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.

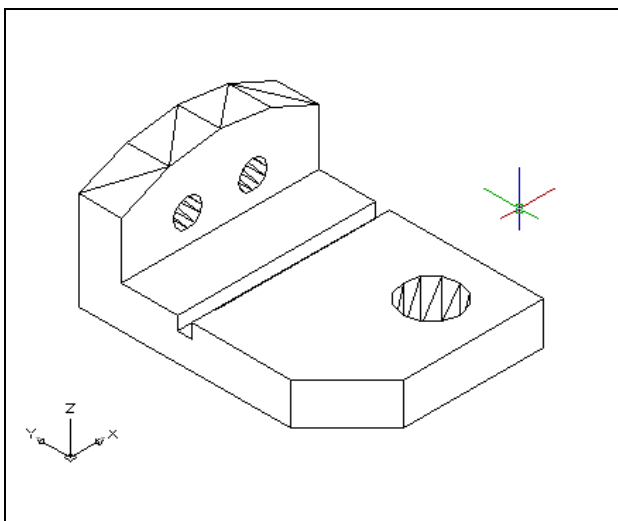


Pick the Chamfer tool on the Modify panel by clicking on the drop-down arrow next to the Fillet tool and picking the Chamfer command. Select the lower vertical corner on the edge of the box. Hit **Enter** and then type “1.00” for the offset distance. Hit **Enter** again and type and “1.00” for the second offset distance. **Enter** and select the vertical tessellation line. Hit **Enter** once more 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 3D Hidden tool on the View panel. 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. If you wish to return the graphical display to the previous setup where all the tessellation lines are seen, then choose the tool under the drop-down box on the View panel called 3D Wireframe. By choosing the 3D wireframe tool, 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 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 panel 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.

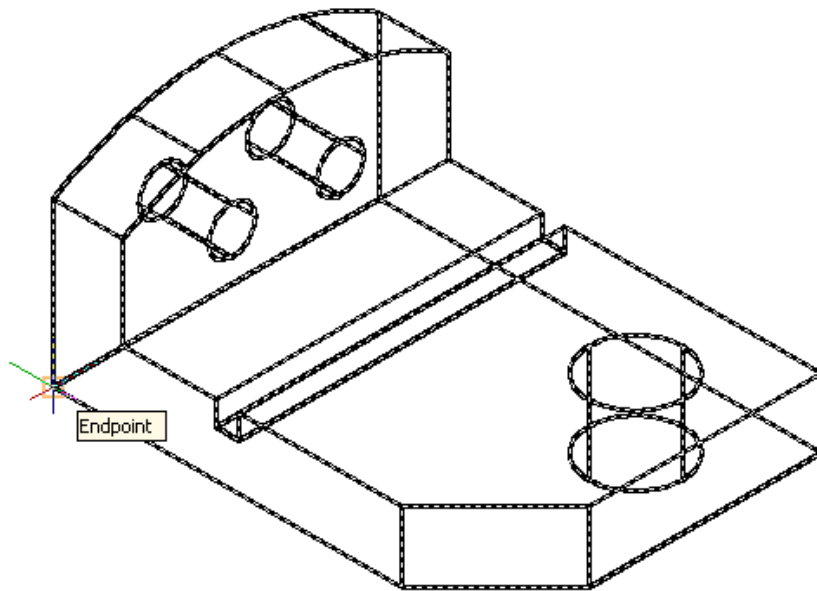


Figure 9.33 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Problem 8 in your Fundamentals of 3D Drawing Folder, select the Save tool on the Quick Access 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)



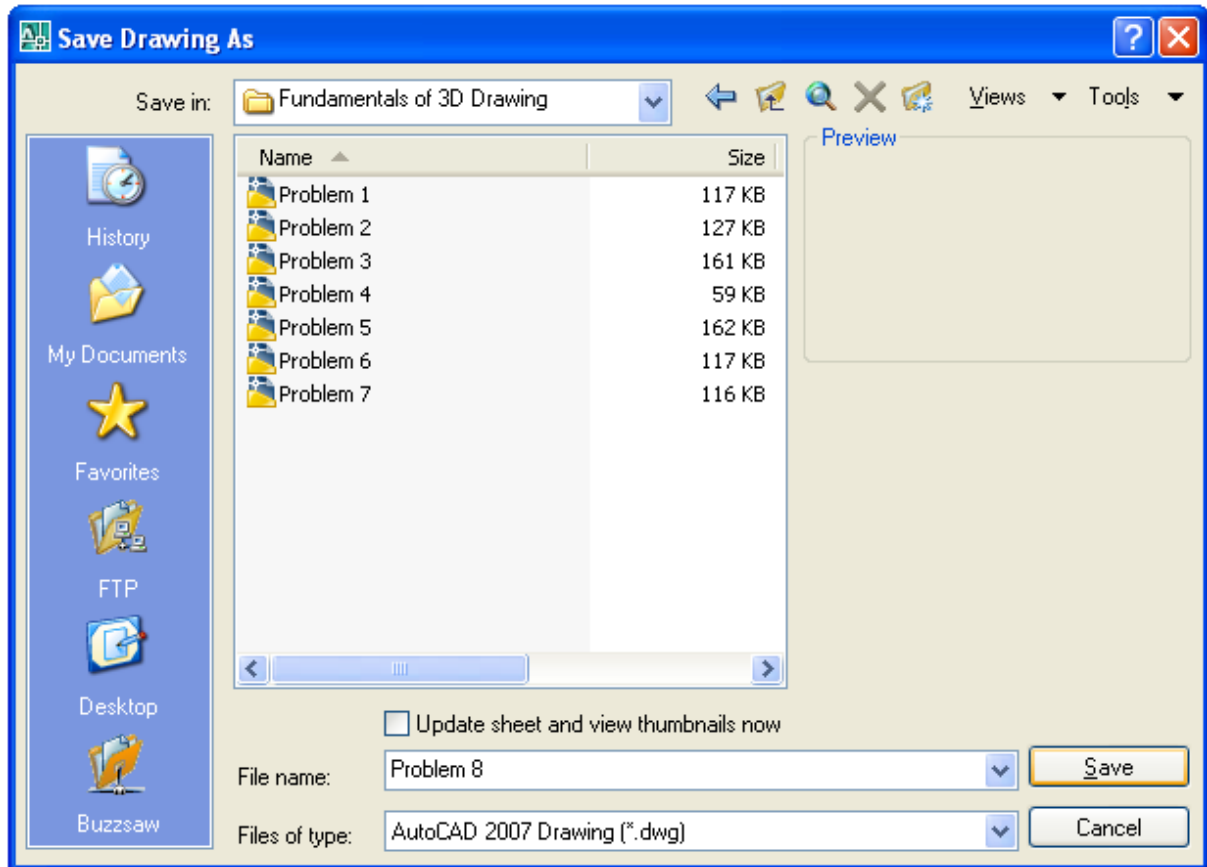


Figure 9.34 – Saving Problem Eight

*** World Class CAD Challenge 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.**

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

