Chapter 9

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**
- **Modifying UCS Settings**
- **Constructing a solid perimeter using the Line command**
- **Adding circles to the detail**
- **Moving a circle inside the perimeter**
- **Using the Mirror 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 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 of the solids**
- **Adding a beveled edge using the Chamfer command**
- **Hiding 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 axes. In this exercise, you will 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. You will place two circles into that shape. After extruding the polyline and circles, you will subtract the cylinders. You will subtract a solid box and a cylinder from the second master solid. You will finish the problem by using the Chamfer tool for the first time in 3D as shown in Figure 9.1.

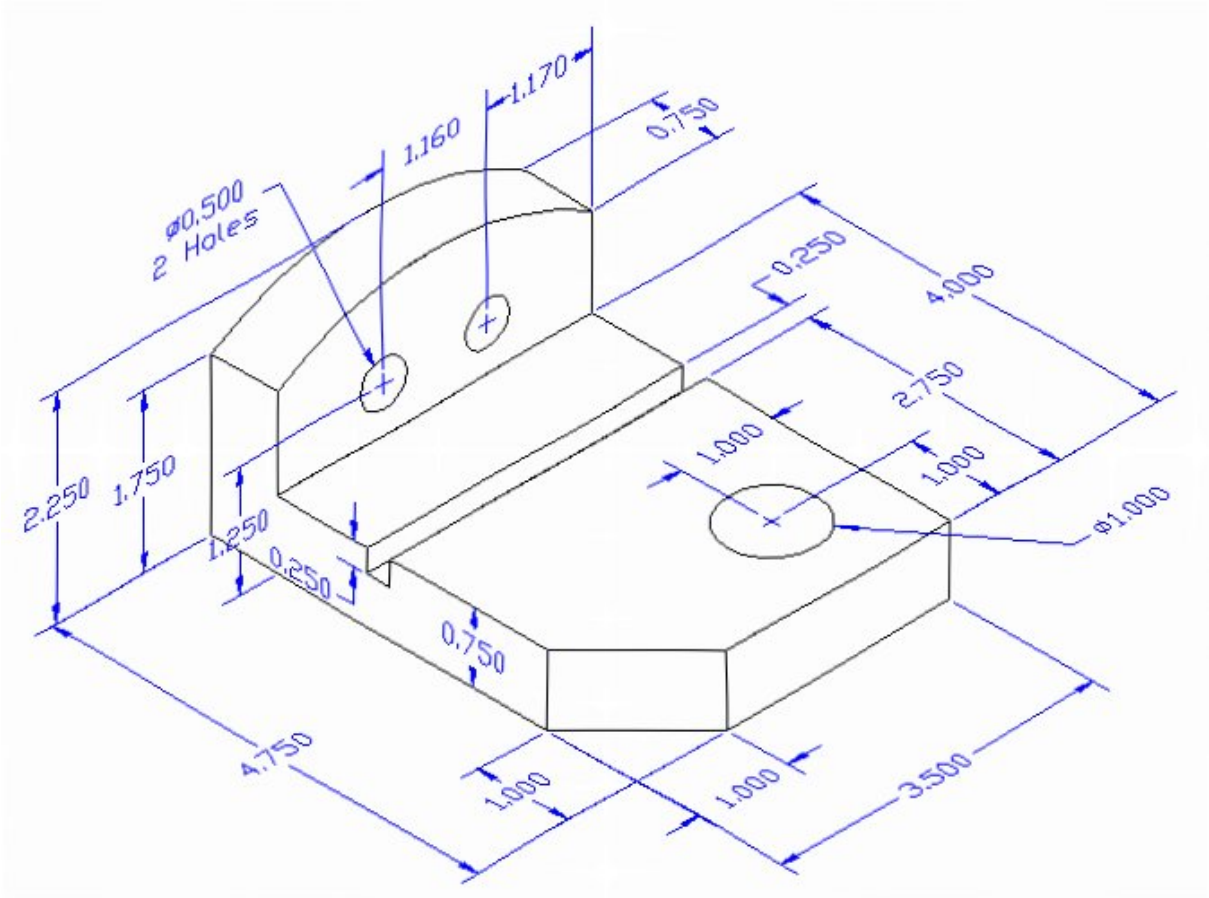


Figure 9.1 – Problem Eight Sketch

Remember, when you receive a sketch from a professional, quickly identify the different shapes and decide begin the drawing. Problem Eight has a number of different details, but you should be able to quickly identify and create these details using strategies you have learned so far in your 3d training. All of the drills in the Fundamentals of 3D Drawing textbook will walk you through such common repetitive maneuvers that you will use often in 3D modeling.

Starting a 3D Part Drawing

You will draw Problem Eight using the Metric template. First select the New tool on the Quick Access Toolbar. A list of templates is loaded in the "Select template" window. Scroll down the list and select Metric.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.

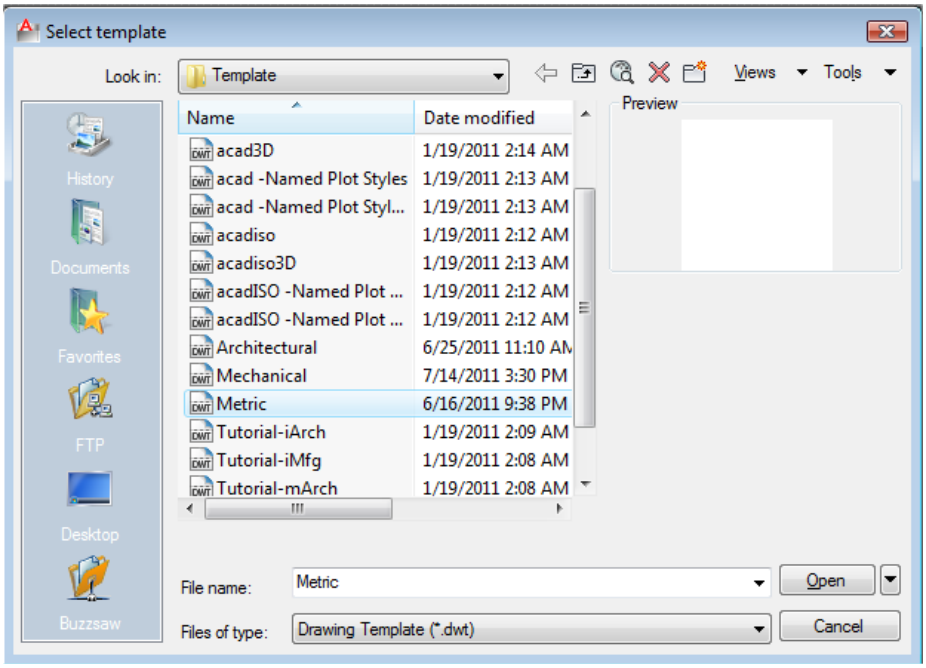


Figure 9.2 – Starting the drawing using the Metric template

Modifying UCS Settings

You may notice the UCS icon moving on the graphical display as you zoom or pan your view. If so, the UCS is currently representing the origin (0,0,0) of the drawing area. This can sometimes be distracting, especially if the origin happens to lie in the vicinity of your drawing. You can change the UCS settings so that the icon is always displayed in the bottom left-hand corner of the drawing area. Select the Show UCS Icon tool from the Coordinates panel under the Home tab (Figure 9.3) and select the "Show UCS Icon" option (Figure 9.4). You also have the choice of keeping the UCS displayed at the origin or turning it off completely.

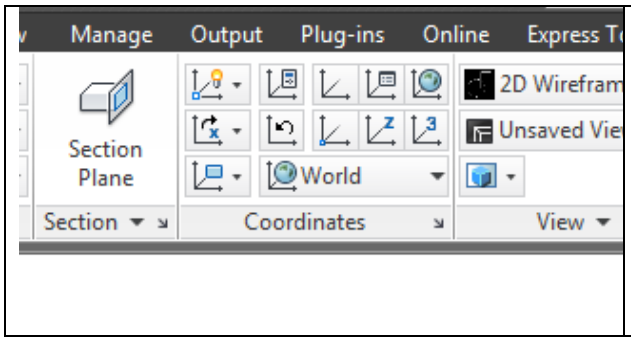


Figure 9.3 - The Coordinates panel

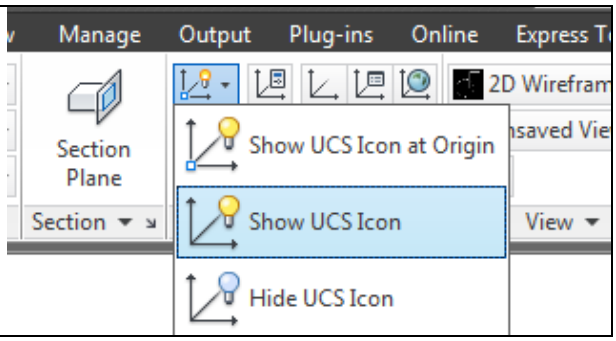
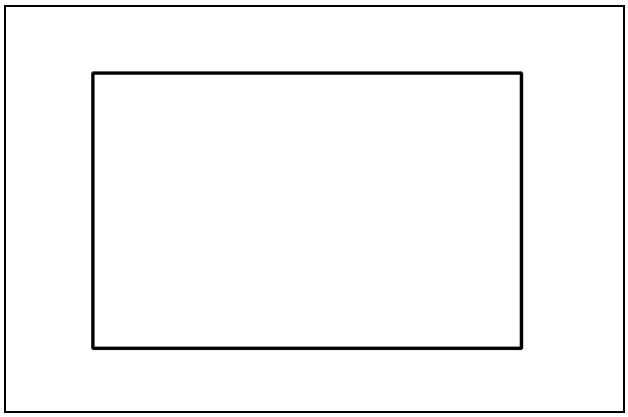


Figure 9.4 - Changing the UCS display settings

Constructing a Solid Perimeter Using the Line Command

First, you need to construct a 3.5 x 2.25 rectangle, so select the Line tool from the Draw panel and start the line in the lower right corner of the graphical display.



With the Ortho mode "on", move your cursor to the right and type "**3.5**" and **ENTER**. To draw the vertical line, move the cursor upward and type "**2.25**" and **ENTER**. Drag the cursor to the left and type "**3.5**" and **ENTER** to draw the third line. To close the rectangle, just type "**C**" and **ENTER**. Notice that you did not draw a Polyline; this is because you will need to add some additional details before you can edit the completed perimeter lines into a polyline.

Figure 9.5 – Drawing a perimeter

Adding Circles to the Detail

Select the Circle command from the Draw panel. Select the endpoint Osnap in the upper right-hand corner of the rectangle as the center point of the circle. Type "**D**" and **ENTER** when the command line prompts you to "specify radius of circle" Then type "**0.5"** and **ENTER** for the diameter and the circle will appear on the corner as shown in Figure 9.7

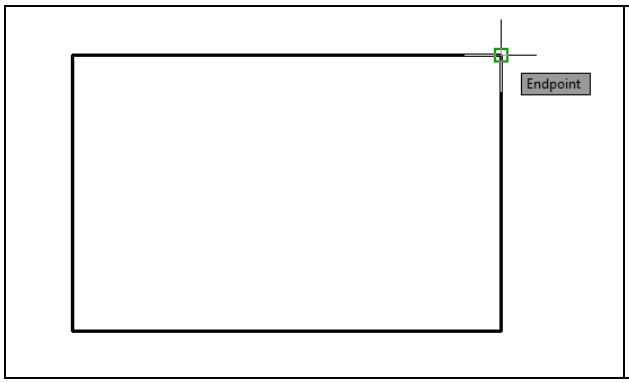


Figure 9.6 – The endpoint Osnap

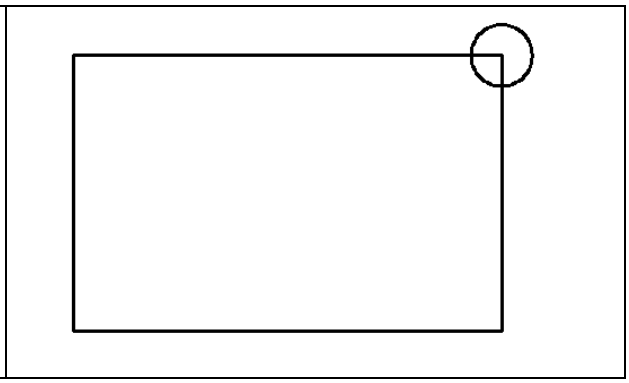
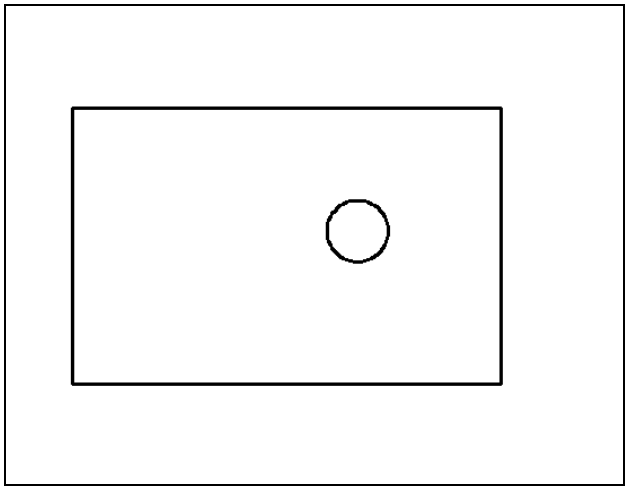


Figure 9.7 – The finished circle

Moving a Circle Inside the Perimeter

Next, you need to position the circle at the precise location inside the rectangle. Use the Move tool on the Modify panel to accomplish the task.



Select the circle when prompted to "Select objects". Pick the circle and the command line will respond with "**1 found**", so hit **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, and then when prompted to select a point of displacement, input **-01.17,-1.0** which will move the circle a negative distance of 1.17 on the X-axis and negative 1.0 unit on the Y-axis. The circle will be as shown in Figure 9.8.

Figure 9.8 – Positioning the circle

Using the Mirror Command to Create Another Circle

You need to copy the circle over to the left. You could use the copy command, but you can also use the Mirror command. In this case Osnap points will make using the Mirror command easier and faster.

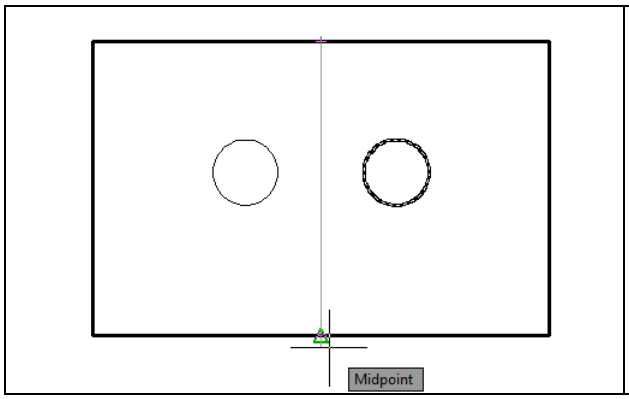


Figure 9.9 – Placing the mirror line

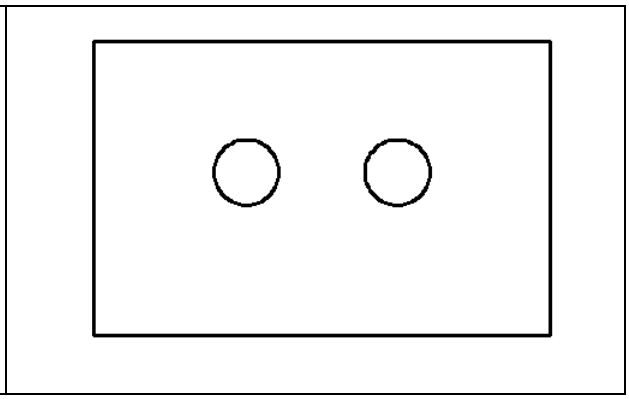


Figure 9.10 – The copied circle

When prompted to select objects, select the circle and hit ENTER. Now you need to create a mirror line that runs down the exact middle of the rectangle. Use the midpoint Osnaps of the top and bottom sides of the rectangle as the first and second points of the mirror line (Figure 9.9). Hit ENTER to accept the default value of "N" for No when prompted to delete the source objects. You should have two circles as shown in Figure 9.10.

Using the Offset Command to Create a New Line

You will use the Offset command to create a new line 1.75 units above the bottom line of the rectangle without selecting points to define endpoints.

Select the Offset command from the Modify panel. Type **1.75** and **ENTER** when prompted to enter the offset distance**.** Next, select the bottom horizontal line as the line you wish to offset. After selecting the line, pick the direction of movement by selecting anywhere above the bottom horizontal line. A new line will appear 1.75 above the bottom line as shown in Figure 9.11. Hit **ENTER** to leave the Offset command after placing the new line.

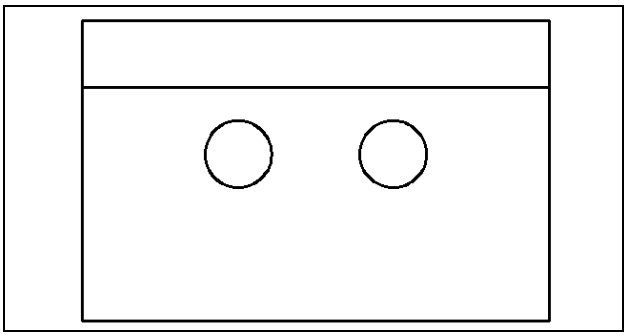


Figure 9.11 – The offset line

Using the Arc Command to Add a Circular Segment

This is the first use of the Arc command in either your 2D or 3D AutoCAD training. Before the development of the Fillet command, CAD drafters would use the Arc command to make any curves in their drawings. However, in a case such as this the Fillet command would not be a very efficient way to create the arc segment.

There are many different ways AutoCAD allows you to place an arc. For now you will use one of the least complex techniques called the 3-Point arc. Select the 3-Point Arc command from the Draw panel.

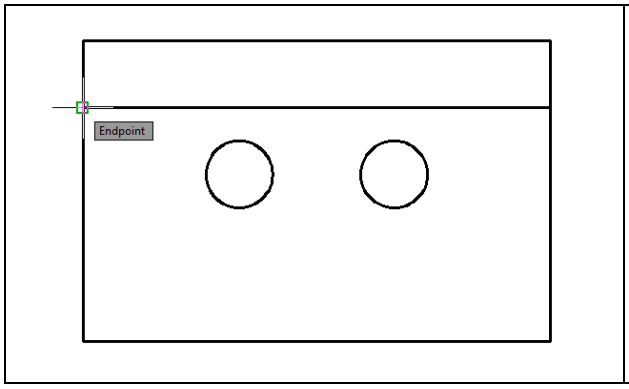


Figure 9.12 – The first point of the arc

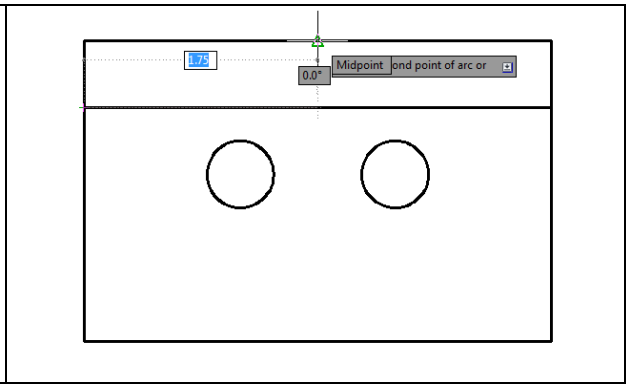


Figure 9.13 – The second point of the arc

As shown in Figure 9.12, pick the endpoint Osnap on the left of the offset line as the first point of the arc. Next, place the second point of the arc by selecting the midpoint Osnap of the top side of the rectangle (Figure 9.13). Now you should see a preview of the arc. Select the endpoint Osnap on the right side of the offset line as shown in Figure 9.14 as the third point of the arc. An arc that intersects all three points will appear inside the rectangle. 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.

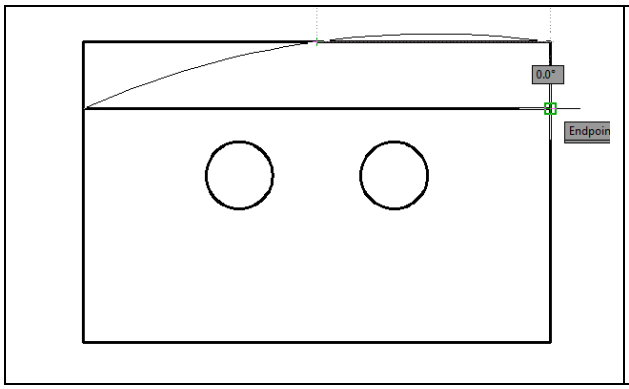


Figure 9.14 – The third point of the arc

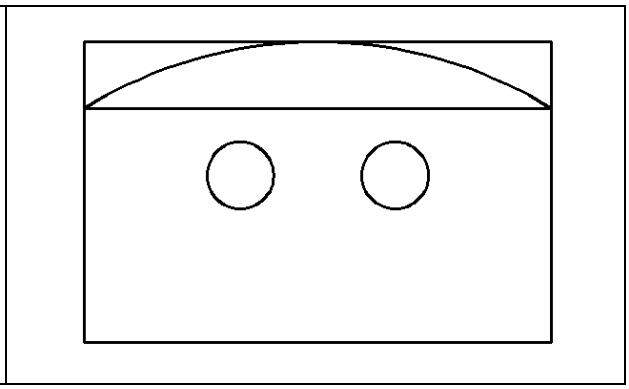
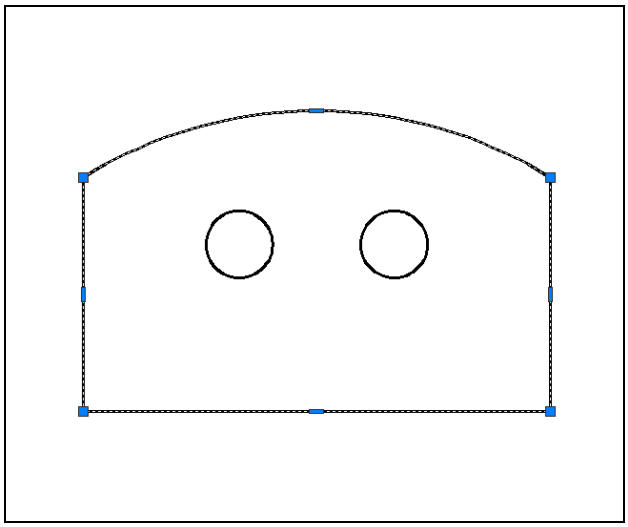


Figure 9.15 – The finished arc

Creating Polylines Using the Edit Polyline Tool

Use the Edit Polyline tool to merge the three line segments and the arc together into a closed polyline. Select the Edit Polyline tool from the expanded Modify panel and the command line will prompt you to "**Select polyline or [Multiple]:**"



Select one of the lines of the perimeter and the command line will reply with "**Object selected is not a polyline Do you want to turn it into one? <Y>**". Type **Y** and **ENTER**, and then the command line will prompt you with a number of editing options from which to choose. Type "**J**" and **ENTER** to choose the Join option. The command line will prompt you to "Select objects" so select all of the lines in the perimeter. The command line should read "**4 total**". Hit **ENTER** to join all of the entities into a closed shape, and then hit ESCAPE to exit the Edit Polyline tool.

Figure 9.16 – The closed polyline selected

Drawing Solid Boxes

Now you will draw a series of solid boxes; one will become a master solid, while another will be subtracted to create a detail. First select the Southwest Isometric view from the View panel so that you will be able to see all three dimensions of the boxes.

Select the Box command from the Modeling panel and select a point to the right of the 2D perimeter as the first corner of the box. Type **L** after specifying the corner on the graphical display to utilize the Length option of the Box command. Type **3.5** and ENTER as the length of the box. Type **4.75** and ENTER for the width, and **0.75** and ENTER for the height. The box will appear as shown in Figure 9.17.

Hit Enter to repeat the Box command and select the bottom left corner of the top face of the first box as the first corner of the new box. Type **L** after specifying the starting point to use the Length option of the Box command. Select the other corner of the top face of the first box 3.5 units away to specify the length of the box. Then enter **-0.25** for the both the width and the height of the box so that it appears as shown in Figure 9.18. Remember that if you are using Dynamic Input, the position of your cursor will affect the way AutoCAD interoperates the values that you enter at the keyboard.

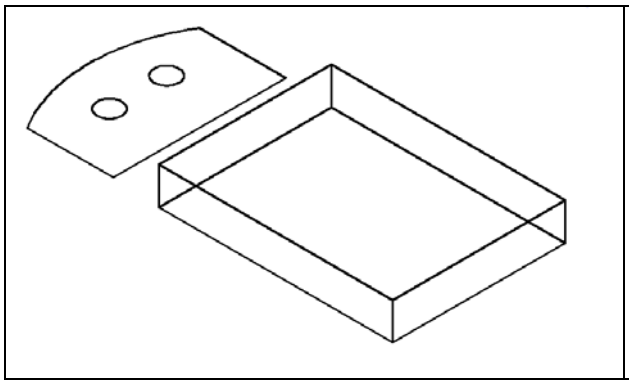


Figure 9.17 – The first box

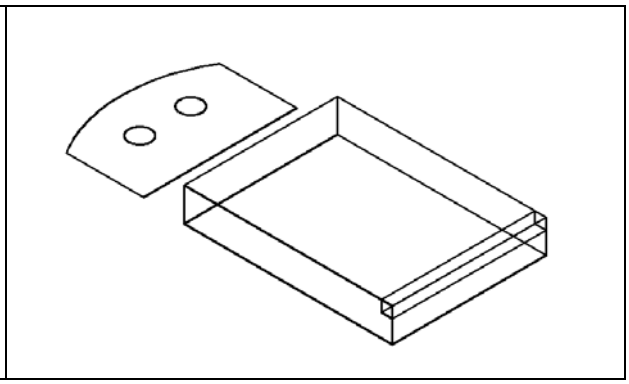


Figure 9.18 – The second box

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 can extrude multiple entities at a time into solids with the Extrude command.

Select the Extrude tool on the Modeling panel and the command line will prompt you to "**select objects**". Select the closed polyline and both circles and the command line will return with "**3 found**". Hit **ENTER** to specify the height or path of the solid. Type "**0.75**" and **ENTER** for the height of the extrusion (Figure 9.19). The three solids will expand into 0.75 units into the Z axis as shown in Figure 9.20.

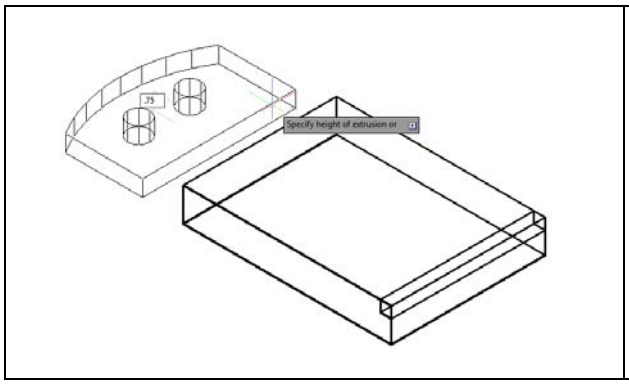


Figure 9.19 – Entering extrusion height

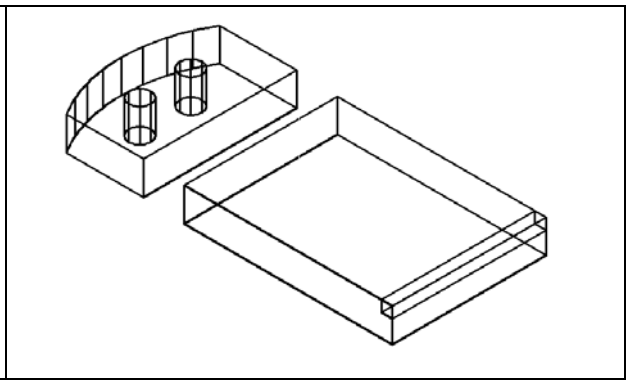
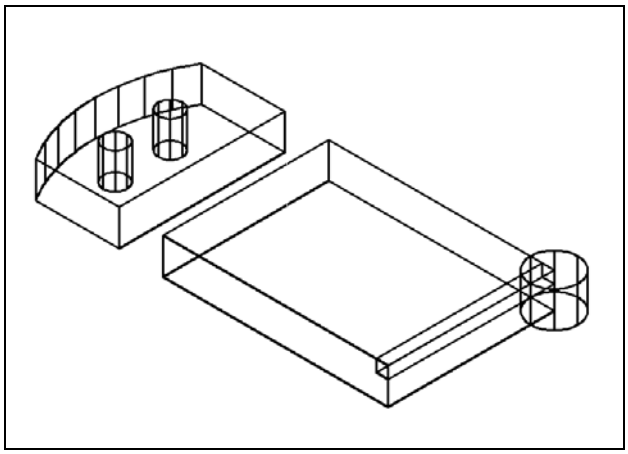


Figure 9.20 – The extruded entities

Drawing a Solid Cylinder

You need to add a 1.0 diameter hole in the master solid. Use the Cylinder command to create the feature.



Select 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 corner endpoint Osnap on the first solid box. Type "**D"** for diameter and hit **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". Type **0.75** for the height of the cylinder and hit **ENTER**. The cylinder will appear as shown in Figure 9.21.

Figure 9.21 – Adding a solid cylinder

Moving 3D Solids into Position on the Master Solid

You need to move two of the solids into position on the master solid. Select the Move command from the modify panel.

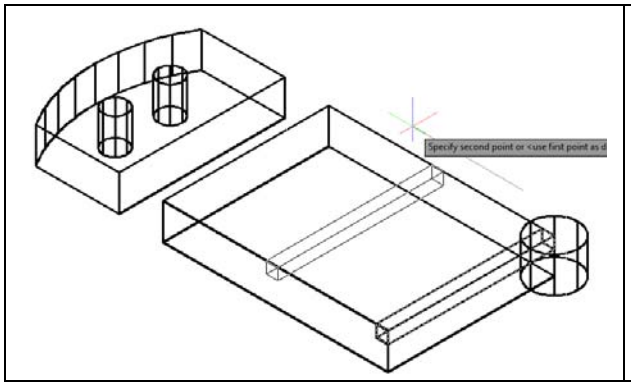


Figure 9.22 – Moving the box

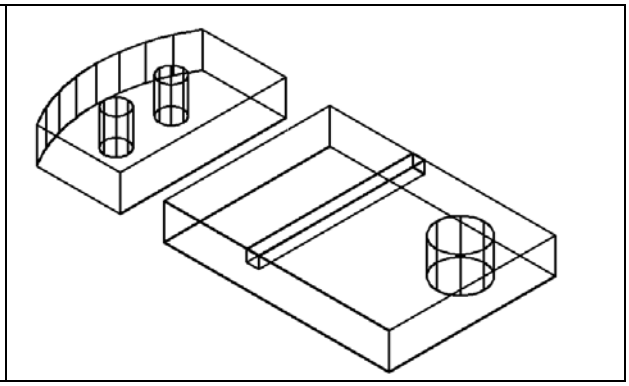


Figure 9.23 – Moving the cylinder

Select the second box as the first object you wish to move. The command line will respond with "1 found" so hit ENTER. 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, hit ENTER to repeat the Move command. Pick the cylinder and the command line will respond with "1 found", so hit 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"** or just "**-1,1"** if using Dynamic Input to move the cylinder as shown in Figure 9.23.

Subtracting 3D Solids from the Master Solids

To remove the cylinders and box from the master solids, select the Subtract tool on the Solid Editing panel.

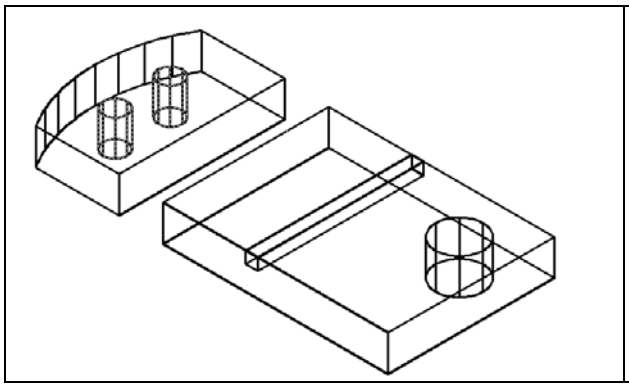


Figure 9.24 – Subtracting cylinders from left master solid

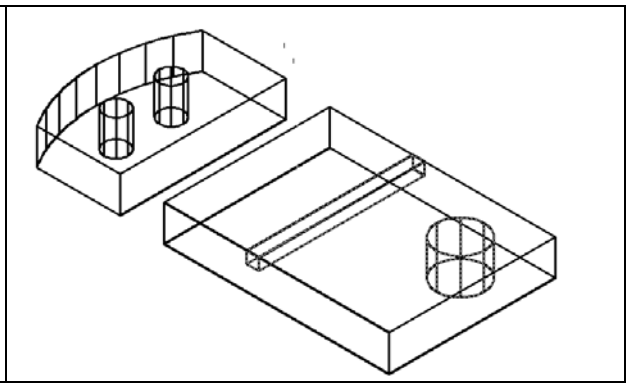


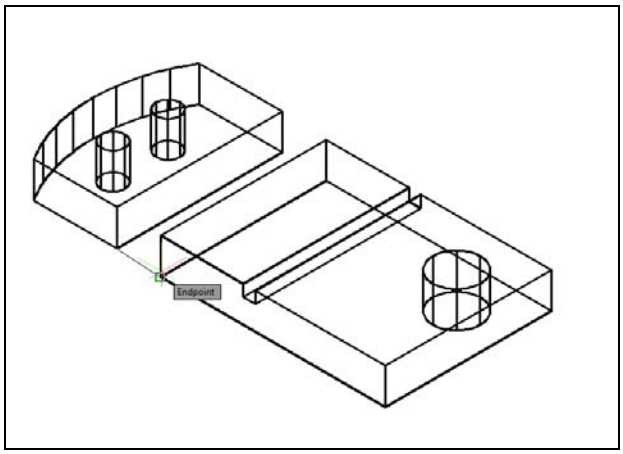
Figure 9.25 – Subtracting cylinder and box from right master solid

The command line will prompt you to "**Select solids and regions to subtract from**". Select the master solid on the left 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 two cylinders inside of the master solid and the command line will respond with "**2 found**" (Figure 9.24). Hit **ENTER** and the shape of the two cylinders will disappear from the master solid as shown in Figure 9.25.

Hit **ENTER** to repeat the Subtract command. The command line will prompt you to "**Select solids and regions to subtract from**". Select the master solid on the right 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 box and the cylinder and the command line will respond with "**2 found**". Hit ENTER and the shape of the two solids will disappear from the master solid as shown in Figure 9.25.

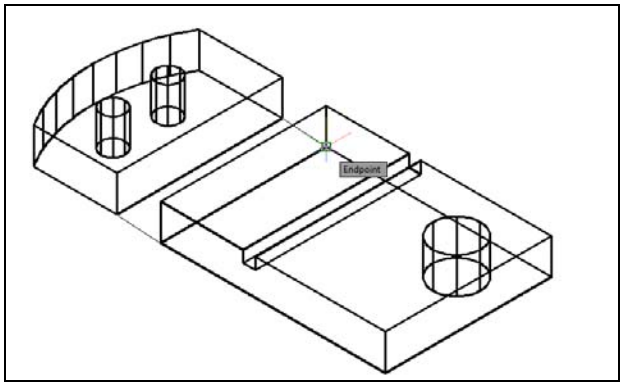
Aligning Two Solids

Select the Align command by expanding the Modify panel. At the command, "**select objects**", pick the prism hit ENTER to proceed to the second part of the function. Notice that this is not the 3D Align command.



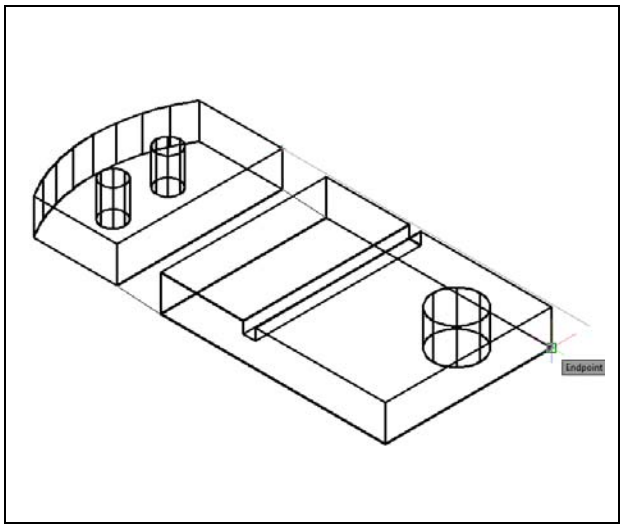
The first source and destination points will end up becoming the same point in the finished alignment. In this case, select the endpoint Osnap of the bottom left corner of the left solid as the first source point and then the endpoint Osnap of the bottom right corner of the back face of the right solid as the first destination point. These two points will join to become the same point when the alignment is finished. A temporary line will appear on the screen showing the relationship of the two points (Figure 9.26).

Figure 9.26 – First alignment points



In this case, the second source and destination points will also end up becoming the same point in model space. For the second source point, select the bottom right corner of the left solid. Select the back right corner of the right master solid as the destination point. Again a temporary line will appear connecting both points, showing their relationship in the alignment (Figure 9.27).

Figure 9.27 – Second alignment points



The third set of points will finalize the definition of the planes of the two shapes that you are aligning. Select bottom right corner of the top face of the left solid as the third source point. Select the front bottom right corner of the right master solid as the third destination point (Figure 9.28). Immediately the left master solid will snap into alignment with the right master solid (Figure 9.29). Again, the difficulty of the Align command lies not in its complexity as a tool but in properly identifying the source and destinations points necessary to create the desired alignment.

Figure 9.28 – Third alignment points

Union of 3D Solids

With the Union command, you can join one shape or more shapes into a single solid. You will use the Union tool to bring the two master solids together.

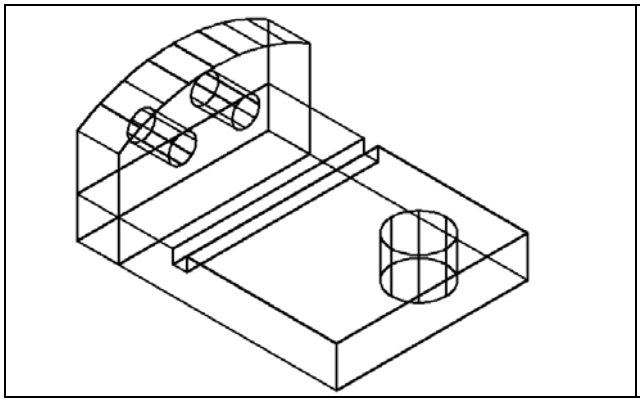


Figure 9.29 – The completed alignment

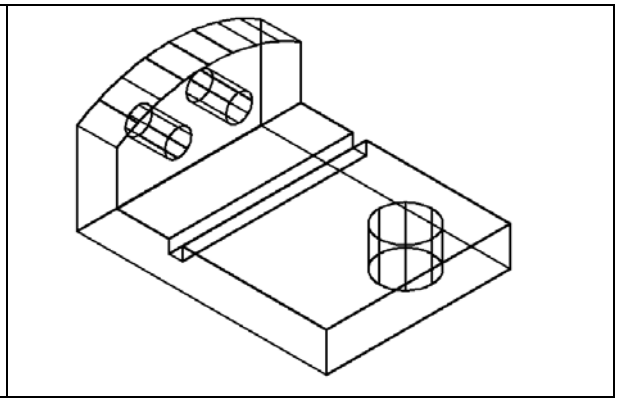
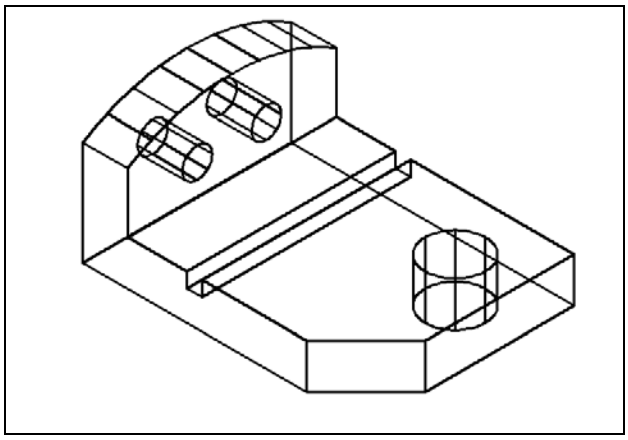


Figure 9.30 – The union of the two solids

To combine the two solids, select the Union tool on the Solid Editing panel. The command line will prompt you to "Select objects". Select both solids and the system will respond with "2 found". Hit ENTER and the solids will become as one entity as shown in Figure 9.30. You can see in the graphical display that the overlapping tessellation lines outlining the two solids To test your solid, select the entity with one pick of the mouse and the entire solid will become dashed.

Add a Beveled Edge Using the Chamfer Command

You can use the Chamfer command to add a beveled edge to a corner. The Chamfer command works much like the Fillet command in 3D: you select the edge rather than the two converging lines to define the corner that you wish to Chamfer. Pick the Chamfer tool on the Modify panel by clicking on the drop-down arrow next to the Fillet tool and selecting the Chamfer command.

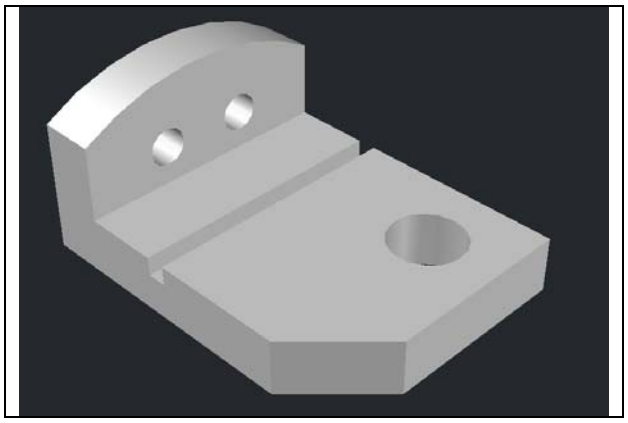


Select the lower vertical corner on the edge of the box. Hit **ENTER** and then type "**1.00**" for the offset distance. Hit **ENTER** again and type and "**1.00**" for the second offset distance. Hit **ENTER** and select the vertical tessellation line when prompted to "Select and Edge or loop". Hit **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.

Figure 9.31 – Placing the beveled edge

Hiding 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 a different visual style to see a more realistic view of the part.



Select the Realistic visual style from the Visual Style panel under the View tab. The part will appear rendered with a metallic sheen as shown in Figure 9.32. If you wish to return the graphical display to the previous visual style, choose the visual style called 2D Wireframe. Neither command alters the mass of the solid or any of the dimensions, but both are available for the computer aided designer as a viewing option.

Figure 9.32 – The Realistic visual style

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, select the Move tool from the Modify panel under the Home tab, select the 3D solid, and hit ENTER. At the prompt "specify base point or displacement," select the bottom left endpoint Osnap as shown in Figure 9.33, which will become its insertion point into an assembly drawing. At the "second point of displacement," type 0,0,0 and ENTER, or #0,0,0 and ENTER if using

Dynamic Input. Problem Eight will move to the origin point. Moving a part at the origin is important for when you wish to insert the solid into an External Referenced Assembly drawing.

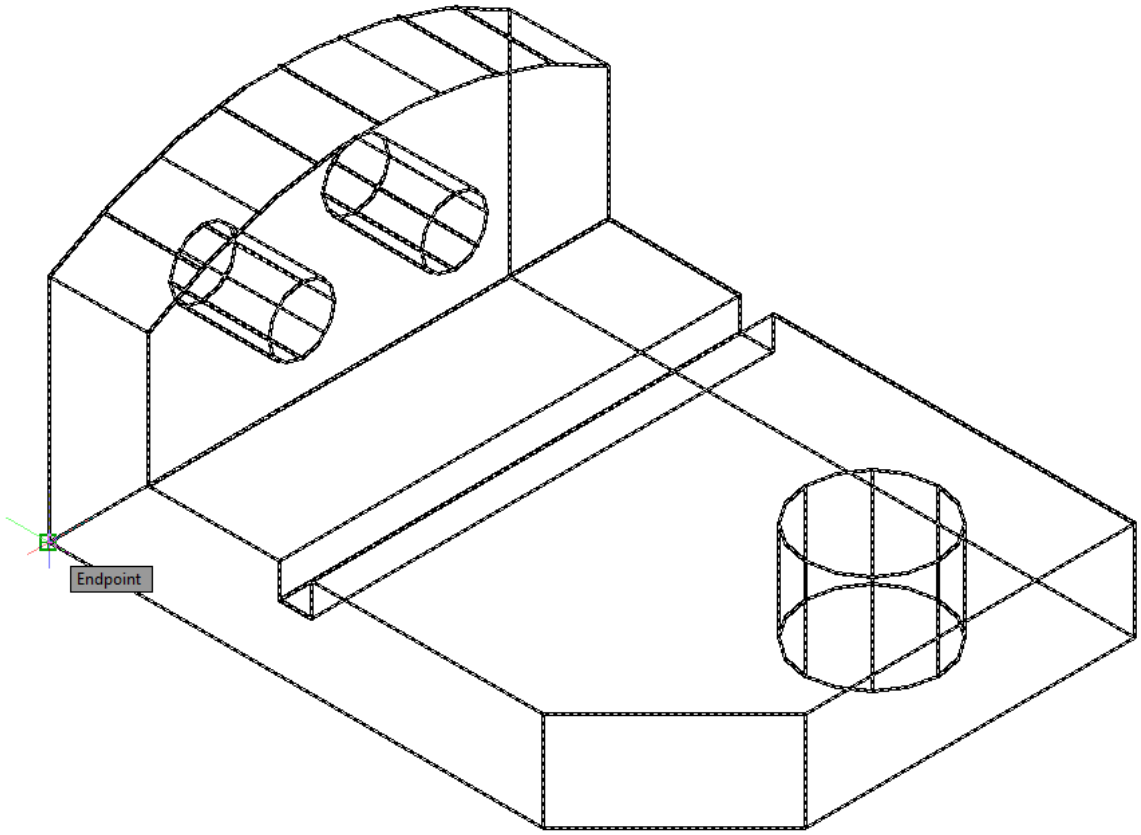


Figure 9.33 – Moving the finished solid to the origin

Saving the Solid Problem

To save Problem 8 in your Fundamentals of 3D Drawing Folder, select the Save As 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. Type “Problem 8” in the File Name textbox and hit the Save button to save the drawing (Figure 9.34).

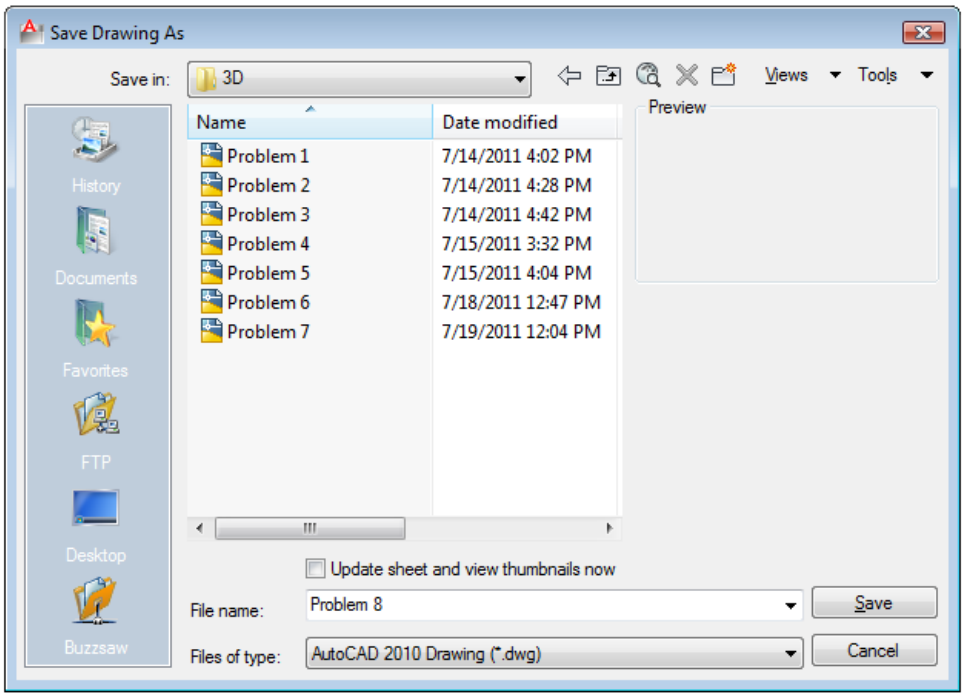


Figure 9.34 – Saving Problem Eight

*** 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 in less than 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.**