

Chapter 14

Revolving Problem Three

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

- **Sketch of Revolving Problem Three**
- **Starting a 3D part drawing**
- **Modifying UCS Settings**
- **Creating center and perimeter lines using the Line command**
- **Creating a closed perimeter using the Edit Polyline command**
- **Revolving a solid from a closed polyline**
- **Shading a 3D solid**
- **Rotating a 3D solid using the 3D Rotate command**
- **Drawing two cylinders**
- **Moving a solid into position on the master solid**
- **Creating an array of a solid using the Array command**
- **Subtracting 3D solids from the master solid**
- **Moving the solid to the origin point**
- **Saving the Solid Problem**

Sketch of Revolving Problem Three

You will start the third revolving problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across two axes. In this exercise, you draw and revolve a closed polyline that contains the most complex shape of the component and then use simple geometric solids to subtract each additional detail. The exercise in revolving problem three involves drawing just a portion of the component and revolving the section into a solid. After revolving the polyline perimeter into a solid, you will place two cylindrical solids onto the master solid, create an array from the detail, and subtract the array from the master solid.

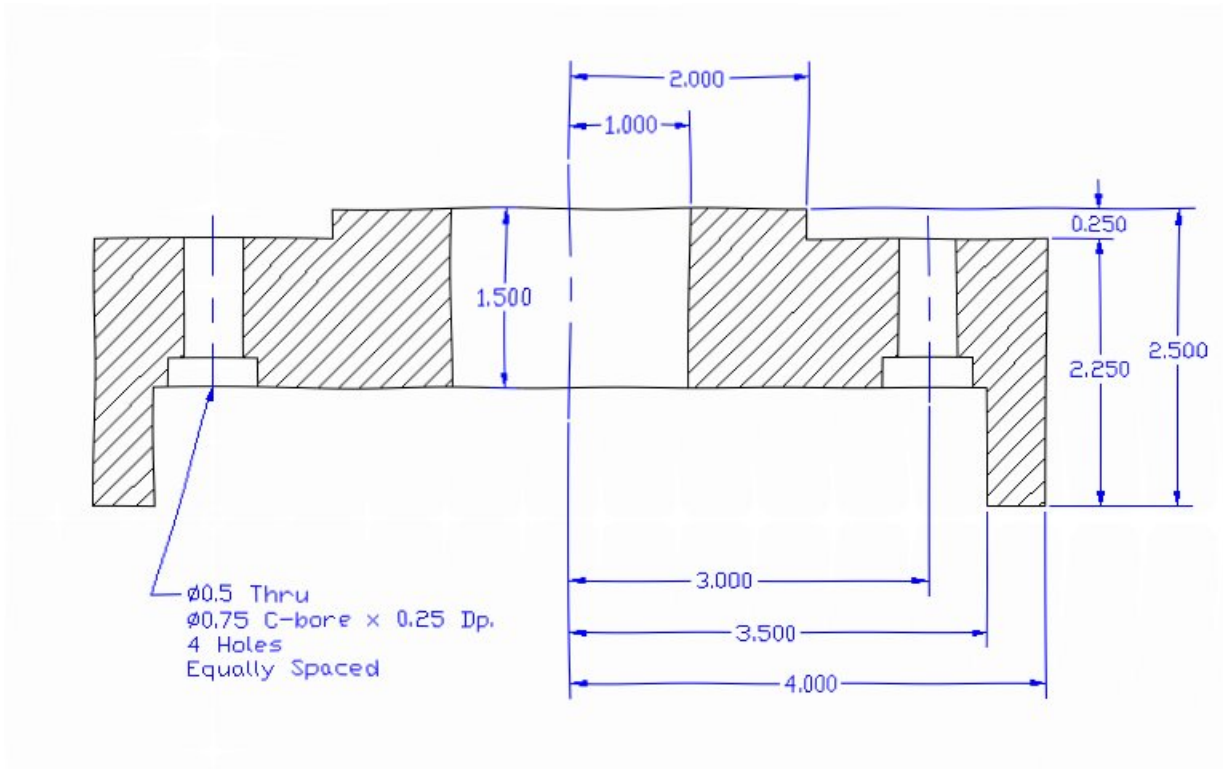


Figure 14.1 – Revolving Problem Three Sketch

Remember, when you receive a sketch from a professional, quickly identify the different shapes and decide how to begin the drawing. You will find that using this strategy of revolving a closed perimeter is very effective for creating circular parts. Place as much detail in the perimeter and interior detail as possible, so that you will have less detail to add to the 3D solid. 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 begin Revolving Problem Three using the Mechanical 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 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.

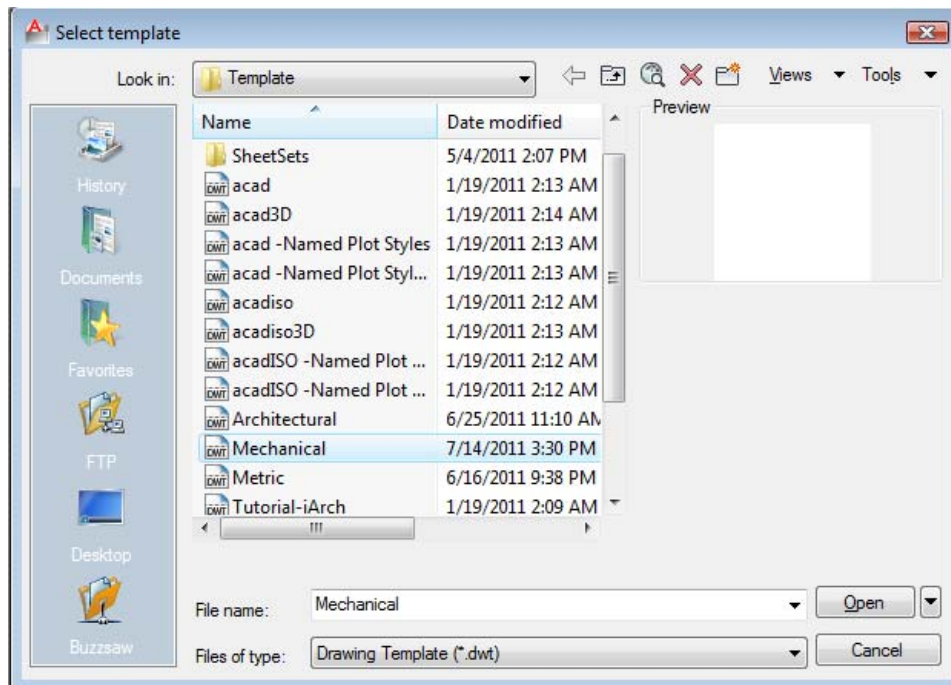


Figure 14.2 – Starting the drawing using the Mechanical template

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 14.3) and select the “Show UCS Icon” option (Figure 14.4). You also have the choice of keeping the UCS displayed at the origin or turning it off completely.

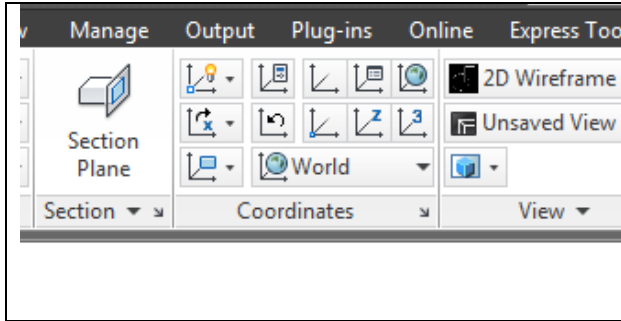


Figure 14.3 - The Coordinates panel

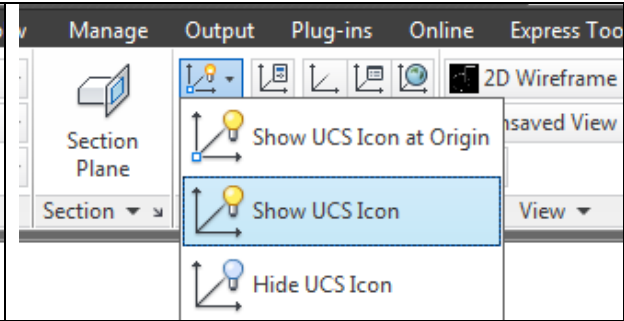
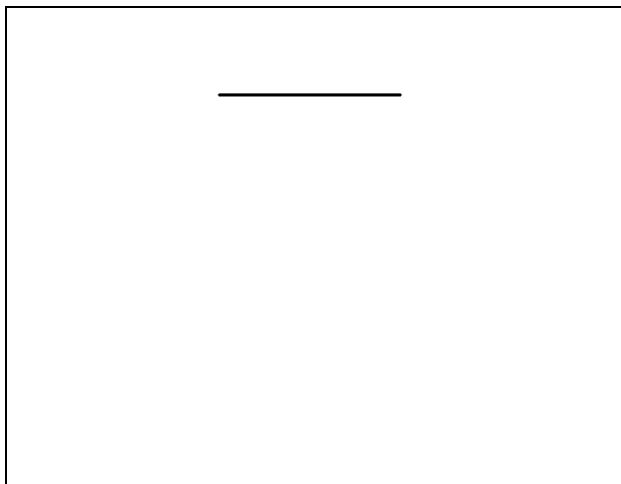


Figure 14.4 - Changing the UCS display settings

Creating Center and Perimeter Lines Using the Line Command

Draw a 1.5 center line from right to left using the Line command. Pick the Line tool on the Draw panel to start drawing the outline that you will later revolve.



Select any point in the middle of the graphical display. With the Ortho mode “on”, move the line on the graphic screen to the left and type **1.5** and **ENTER** (Figure 14.5). Hit **ENTER** twice to exit and repeat the Line command. You will begin the next line exactly 1 unit below the endpoint of the first line. To find this point, use a relative point: type “**@0,-1**” and the starting point of the line will appear on the display 1 unit below the endpoint of the first line. Now draw the outline referencing the command line data in Figure 14.6.

Figure 14.5 – Drawing a center line

```
Command: LINE Specify first point: @0,-1.0
Specify next point or [Undo]: 1.5
Specify next point or [Undo]: 1.0
Specify next point or [Close/Undo]: 0.25
Specify next point or [Close/Undo]: 2.0
Specify next point or [Close/Undo]: 2.25
Specify next point or [Close/Undo]: 0.5
Specify next point or [Close/Undo]: 1.0
Specify next point or [Close/Undo]: c
```

Figure 14.6 – The Line command prompts for the outline

With the Ortho mode “on”, draw a **1.5** unit line to the right, a **1.0** line downward, a **0.25** line to the left, a **2.0** line down, a **2.25** line to the left, a **0.5** line upward, a **1.0** line to the right and then type “**C**” to close the line. AutoCAD computes the last line length and angle when you use the close option. Every step in the process of drawing the detailed perimeter is shown in Figures 14.6 through 14.12. A closed shape will appear in the graphical display as shown in Figure 14.13.

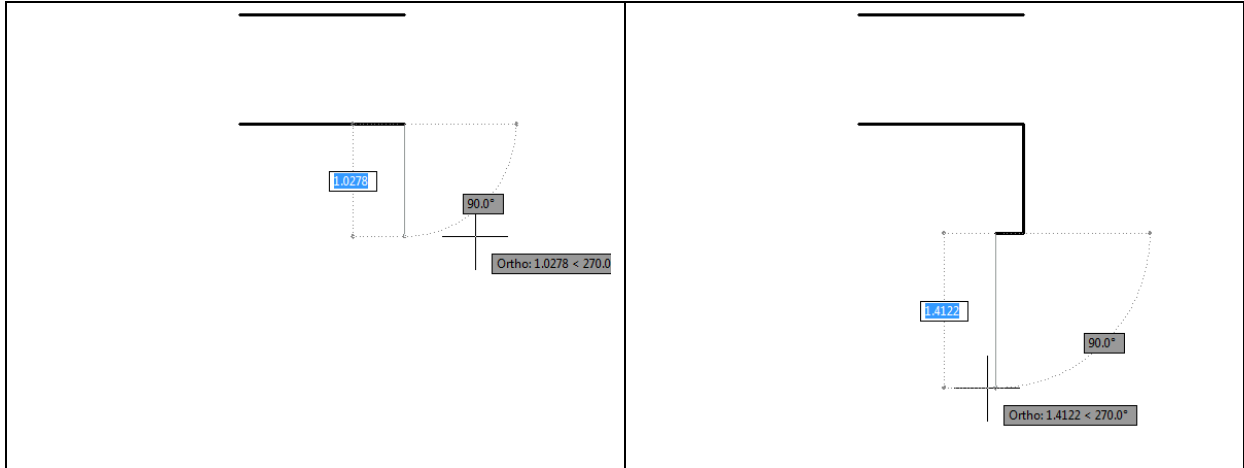


Figure 14.7 – Draw a line 1.5 to the right

Figure 14.8 – Lines 1.0 down and 0.25 left

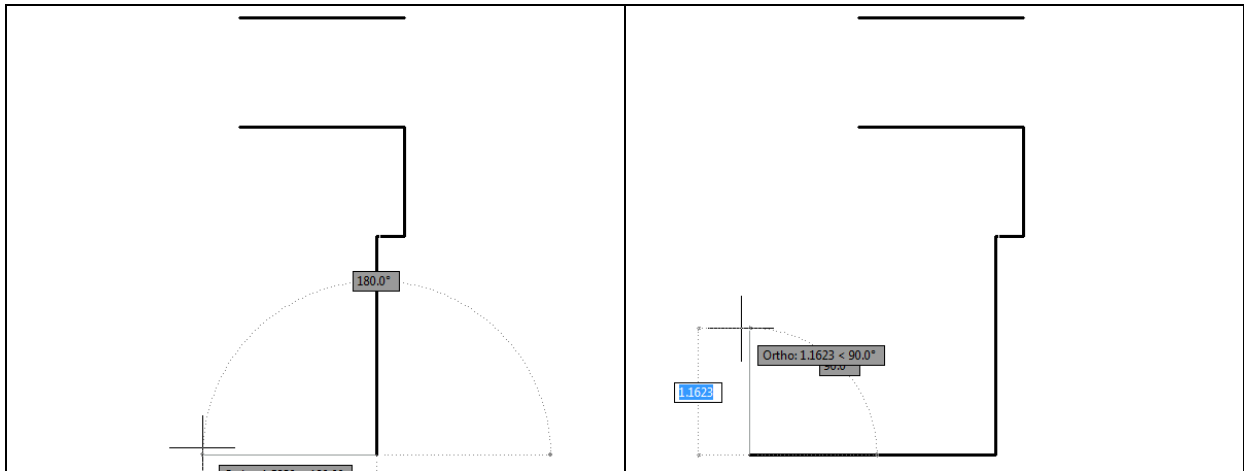


Figure 14.9 – Draw a line 2.0 downward

Figure 14.10 – Draw a line 2.25 left

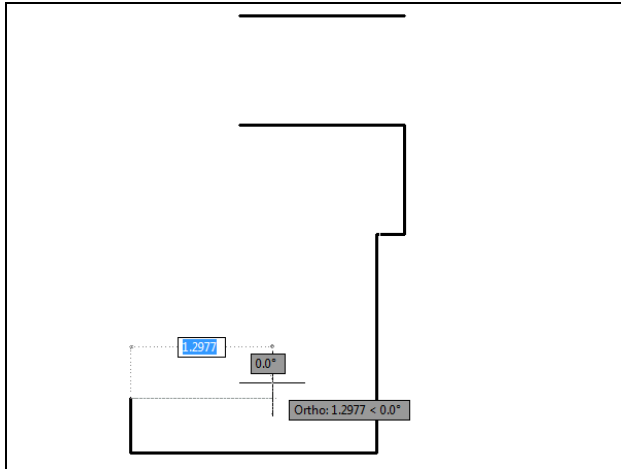


Figure 14.11 – Draw a line 0.50 upward

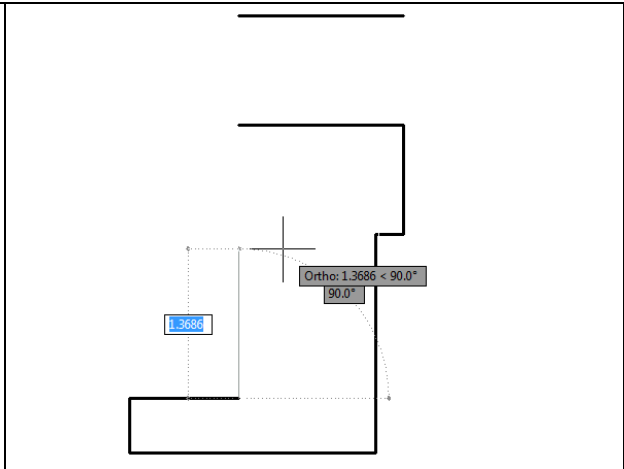


Figure 14.12 – Draw a line 1.0 right

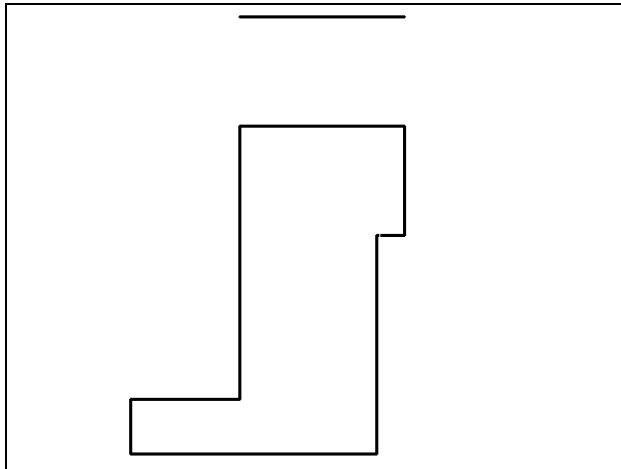


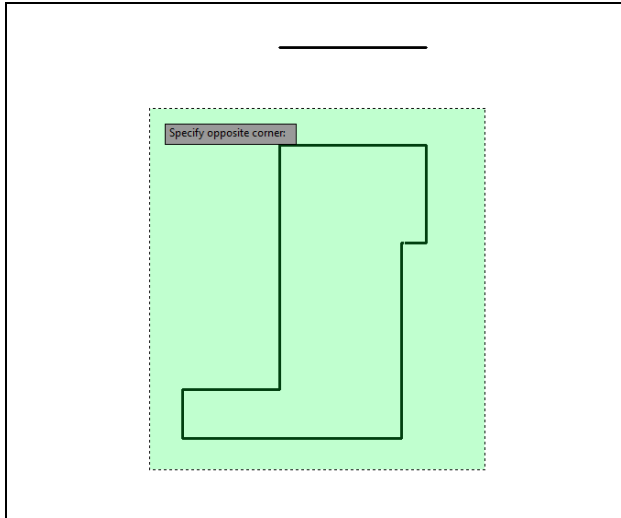
Figure 14.13 – Close the series of lines

You could use the Polyline tool to draw this simple perimeter and then go straight to revolving the perimeter around the center line, but often you may find the task easier to do if you are using the Line tool like you did in this exercise. The Polyline function can be very rigid; many computer aided designers will use the Line function so that they can use the Offset, Trim, or Fillet tools to modify the perimeter. As you complete the 3D problems in this textbook, you will find what tools you prefer.

Creating a Closed Perimeter Using the Edit Polyline Tool

Use the Edit Polyline command to merge the lines together into a closed polyline. Select the Edit Polyline tool by expanding the 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 (Figure 14.14). The command line should read “**8 found**”. Hit **ENTER** to join all of the entities into a closed shape, and then hit **ESCAPE** to exit the Edit Polyline tool.

Figure 14.14 – Making a closed polyline

Revolving a Solid from a Closed Polyline

You will revolve the closed Polyline into a very complex 3D solid. When turning the perimeter into the solid, you will use the centerline for the axis of rotation.



Select the Revolve tool by clicking on the drop-down arrow below the Extrude icon on the Modeling panel and selecting the Revolve tool. The command line will prompt you to “select objects”. Pick the perimeter and the command line will return with “1 found” as shown in Figure 14.15. Next, you will be prompted to “Specify axis start point or define axis by [Object/X/Y/Z] <Object>:” Type “**O**” and **ENTER** to select the axis by picking an object. Pick the axis line as shown in Figure 14.16. Hit **ENTER** to select 360 as the default angle of revolution, and the new solid will appear as shown in Figure 14.17.

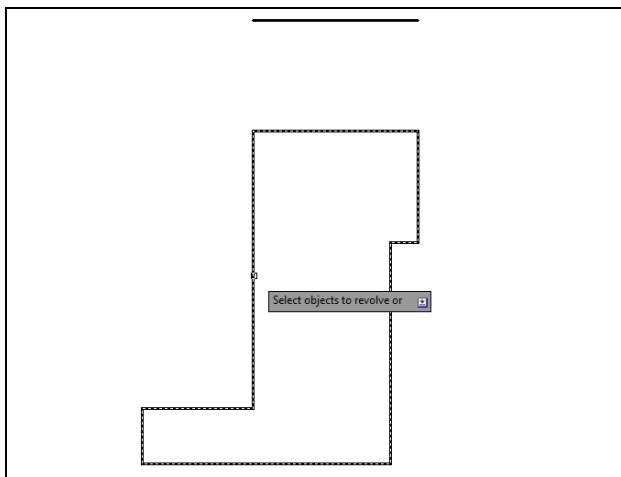


Figure 14.15 – Select the polyline

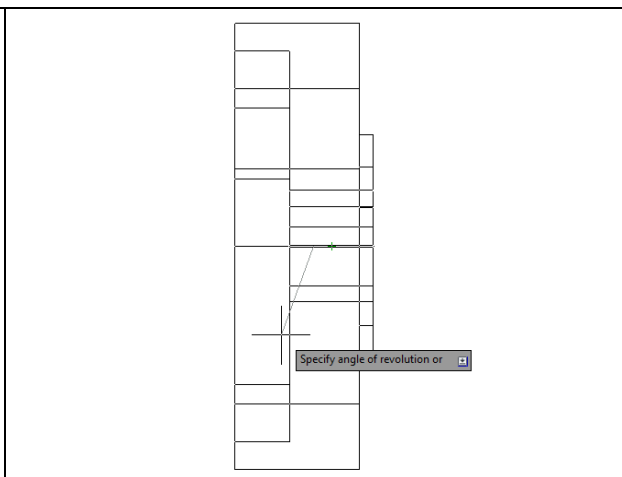


Figure 14.16 – Entering the angle of revolution

On the View panel, select the Southwest Isometric View tool to allow you to see the all three axes of the part as shown in Figure 14.17. Again the tessellation lines on the screen can be confusing. In the next step of this problem you will use another visual style to better conceptualize the part.

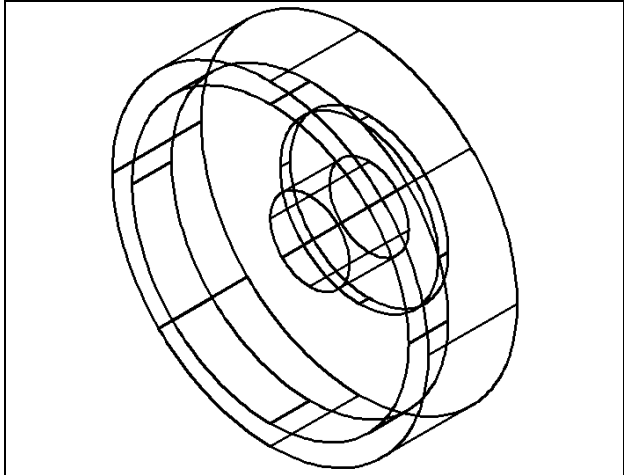
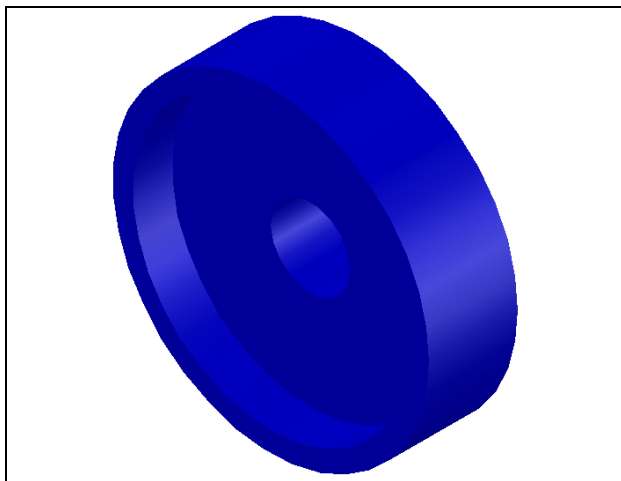
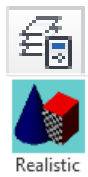


Figure 14.17 – The revolved solid

Shade the 3D Solid

Expand the Layer panel and select the Layer Properties Manager tool. Change the layer “0” color to blue so you can easily see the shaded part. Now select the Realistic Visual Style tool on the View panel. The part is now a solid blue color as shown in Figure 14.18. The tessellation lines associated with the Hide command are no longer interfering with your ability to read the drawing.



Your customers will most likely prefer this mode to the Hide command because they can visualize the product clearly. If you ever desire to return to standard viewing, choose the 2D Wireframe tool on the View panel to return the drawing to the previous settings.

Figure 14.18 – Realistic visual style

Rotating a 3D Solid Using the 3D Rotate Command

Select the 3D Rotate command on the Modify panel. At the prompt “select objects”, pick the solid on the graphical display and hit **ENTER**.

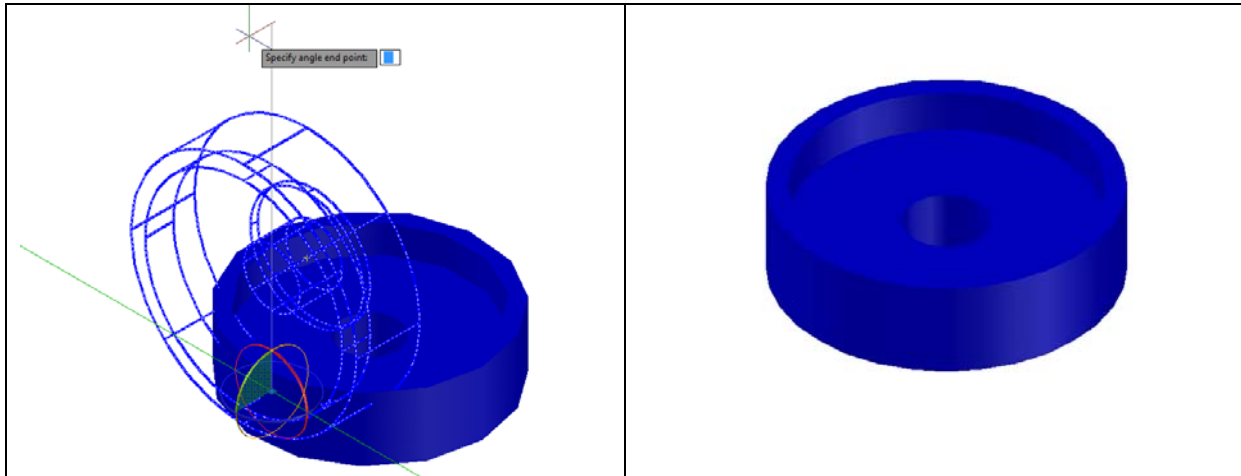


Figure 14.19 – The angle end point

Figure 14.20 – The rotated solid

Now select the bottom quadrant Osnap as the base point of rotation. Specify the Y axis as the axis of rotation by selecting the circle that orbits the Y axis (Figure 14.19). With the Ortho mode “on”, specify an angle start point and then use the cursor to move the solid 90 degrees clockwise (Figure 14.19). Finish the rotation by clicking with the left mouse button and the solid will appear rotated on its base as shown in Figure 14.20.

Creating a New Layer in an AutoCAD Drawing

In the past you created an extra layer so that you could control groups of entities by freezing and thawing the new layer. In this case, the new layer will serve the purpose of allowing you to visualize more clearly the solids that with which you will be working by differentiating the colors of their respective entities. To create the “Cylinder” layer using the Layers Properties Manager, select the Layers command button under the drop-down arrow on the Layers panel on the Home tab. Add the “Cylinder” layer using the New button in the Layer Properties Manager. Make the layer color “red”. Set this layer as the active layer in the Layer Control list box.



Drawing Two Cylinders

Now you will draw two cylinders: one to create a hole completely through the body of the master solid, and the other to serve as a counterbore for each hole.



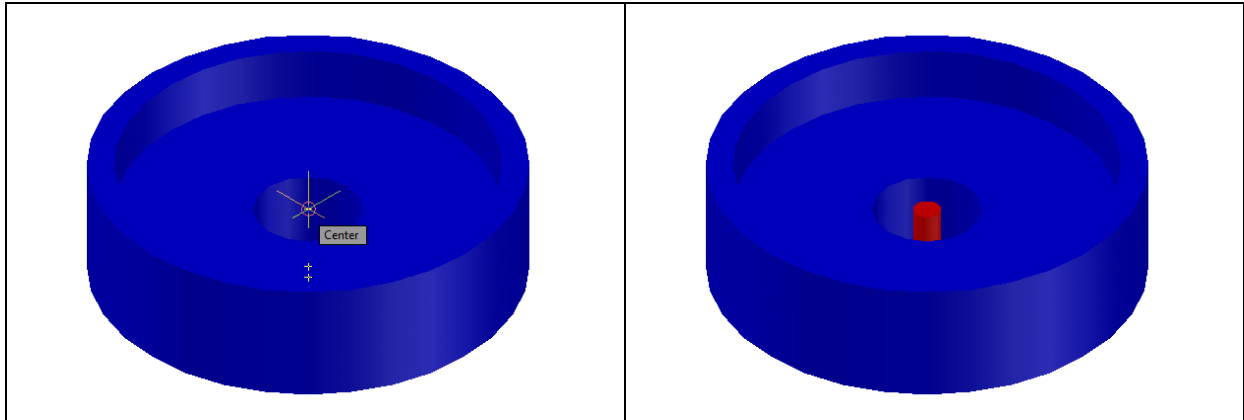


Figure 14.21 – Center of the first cylinder **Figure 14.22 – The first cylinder**

Select the Cylinder tool from the Modeling panel. Specify the center point for the top of the cylinder as the center Osnap of the top of the center hole of the master solid as shown in Figure 14.21. Type “D” for diameter and hit **ENTER** to input the sketch’s measurement of **0.5**. Hit **ENTER** and then type **1.5** as the height of the cylinder, making sure that you are pulling the cylinder down into the area of the master solid (with Dynamic Input “on”). The cylinder will appear in the graphical display as shown in Figure 14.22.

Select the Cylinder tool again, and select the same center point as before. This time, enter the diameter as **0.75** and the height as **-0.25** (Figure 14.23). The second cylinder will appear as shown in Figure 14.24.

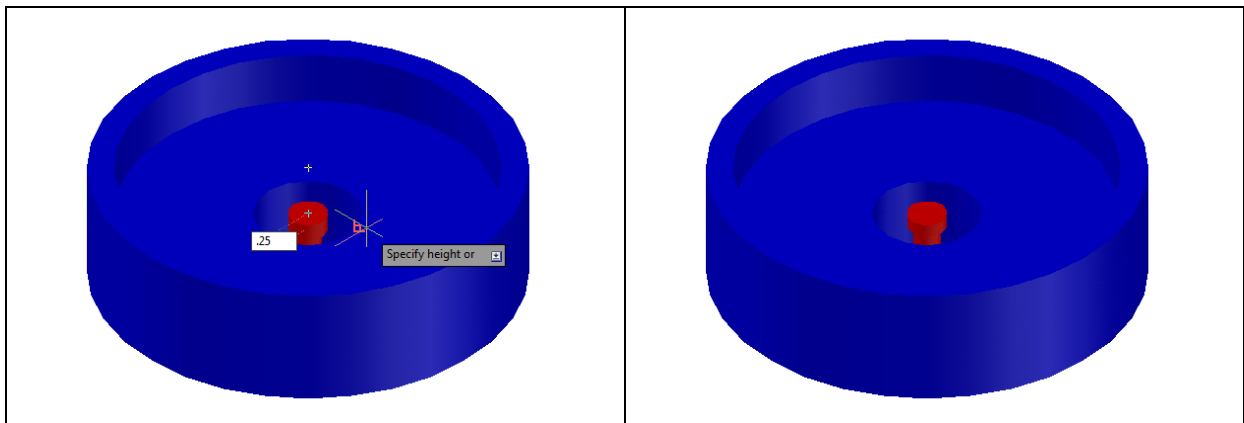


Figure 14.23 – Center of the second cylinder **Figure 14.24 – The second cylinder**

Union Two Solids

Use the Union command to combine both cylinders into one solid that you can then create into a polar array. Select the Union tool from the Solid Editing panel.



AutoCAD will prompt you with “Select objects”. Select both shapes and the system will respond with “2 found” (Figure 14.25). Hit **ENTER** and the solids will combine into one entity as shown in Figure 14.26.

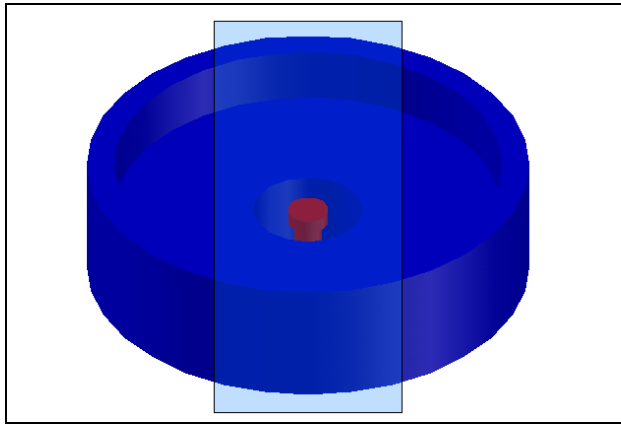


Figure 14.25 – Selecting both cylinders

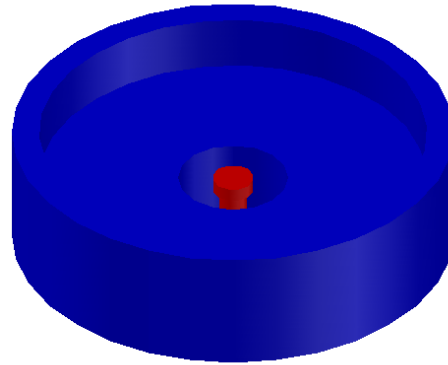


Figure 14.26 – The cylinders combined

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.

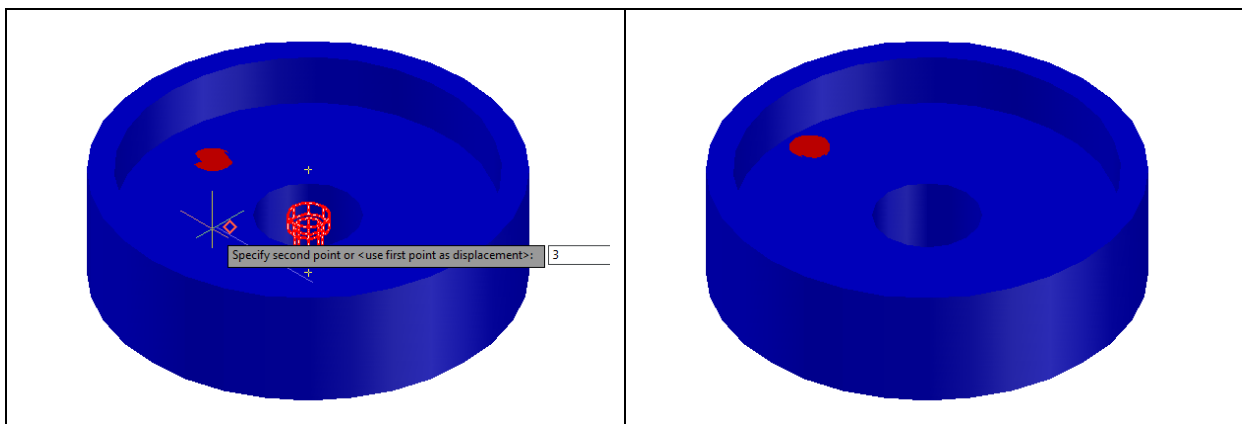


Figure 14.27 – Moving the solid

Figure 14.28 –The Solid in position

To move the cylinder, select the Move tool on the Modify panel. Once you choose the Move command, the prompt will be “Select Objects”. Pick the combined cylinders and the command line will respond with “1 found”, so just hit **ENTER** to go the second part of the command. Next you need to “specify base point or displacement”, which will be anywhere on the graphical display as shown in Figure 14.27. For the “second point of displacement”,

move the cursor to the left with the Ortho “On”, and type **3** and **ENTER**. The cylinder will be ready for its polar array as shown in Figure 14.28.

Using the Array Command to Create Circular Patterns

You now need to copy and rotate the solid cylinder, spacing each one 90 degrees apart. Use the Polar Array function for this operation.



Select the drop-down arrow next to the Array tool on the Modify panel and select the Polar Array tool. First you will be prompted to select objects, so select the combined cylinders and hit **ENTER** (Figure 14.29). Next you need to specify the center of your polar array, so select any center Osnap point that corresponds with the middle of the master solid (Figure 14.30). Enter **4** for the number of items, and when prompted to specify the angle to fill type **360** and **ENTER** (Figure 14.31). Three more cylinders should appear so that each is 90 degrees from the other. Use the Orbit tool on the Navigation bar to bring the entire array into view as shown in Figure 14.32.

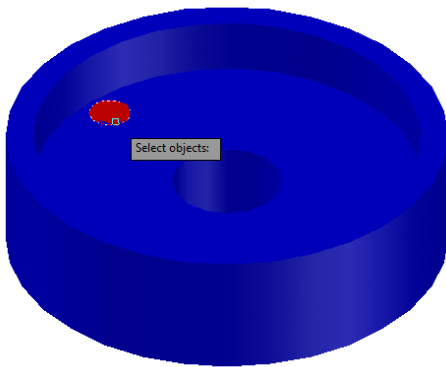


Figure 14.29 – Selecting the cylinder

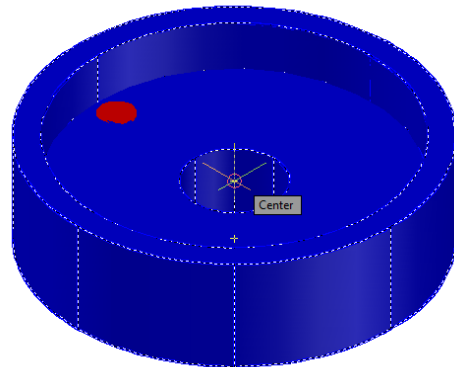


Figure 14.30 – Selecting the Center Osnap

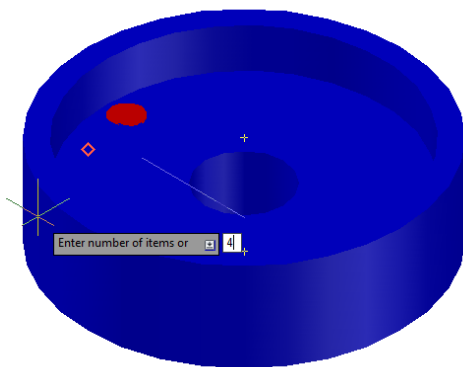


Figure 14.31 – Specifying number of items

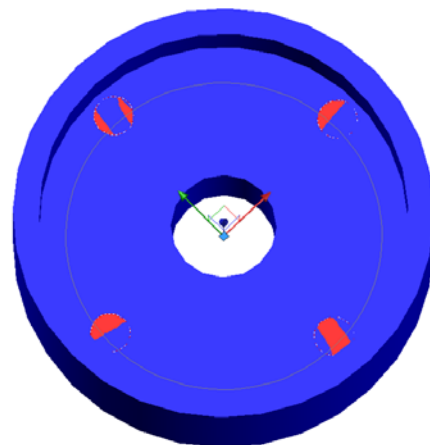


Figure 14.32 – The finished array

Subtracting 3D Solids from the Master Solids

Before subtracting the solids, you must explode the polar array by selecting the Explode tool from the Modify panel and then selecting the array and hitting ENTER. Now you are ready to subtract the three cylinders from the master solid. To remove the cylinders from the master solid, select the Subtract tool on the Solid Editing panel.

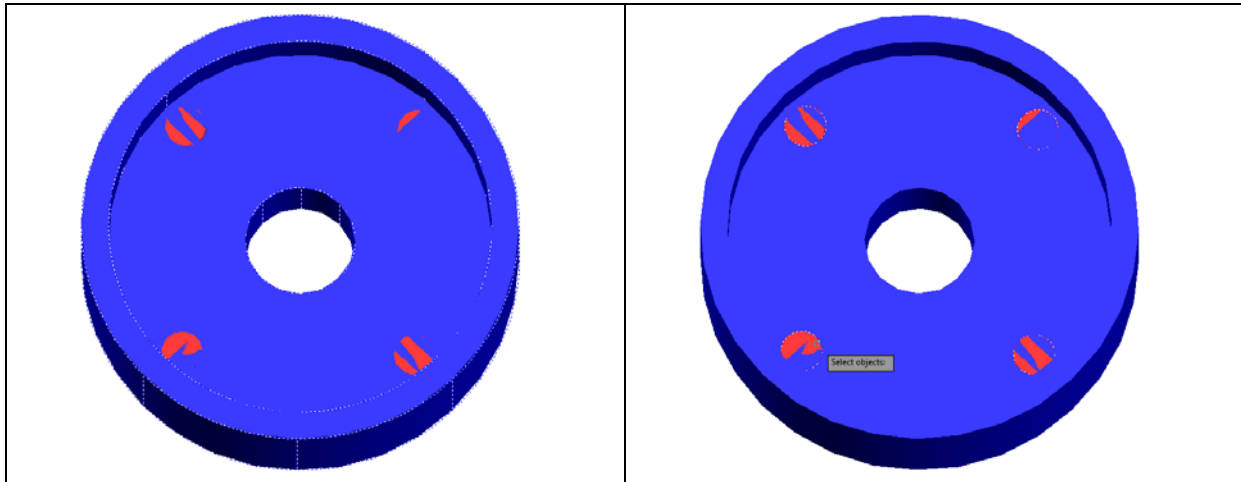


Figure 14.33 – The master solid

Figure 14.34 – Cylinders to be subtracted

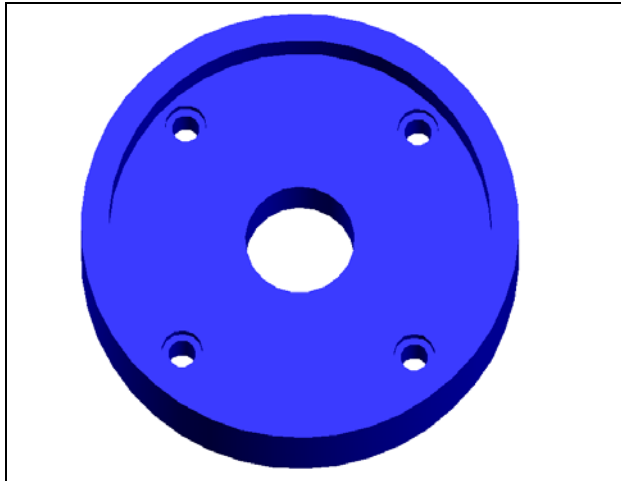


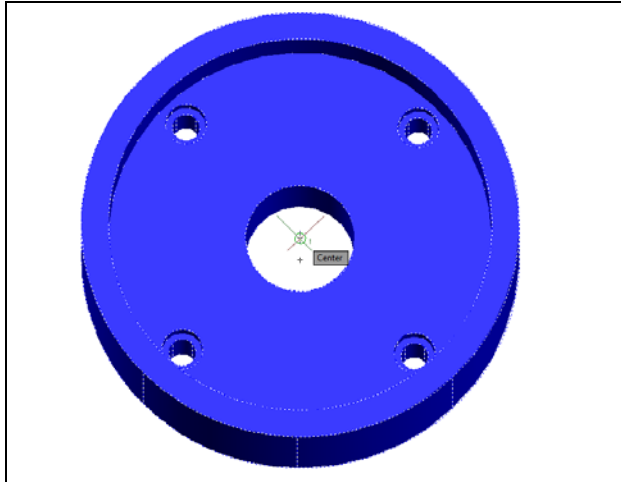
Figure 14.35 – Master solid with holes

The command line will prompt you to “**Select solids and regions to subtract from**”. Select the master solid and the system will respond with “**1 found**” (Figure 14.33). Hit **ENTER** to proceed to the next step, which is to “**Select solids and regions to subtract**”. Pick the six cylinders inside of the master solid and the command line will respond with “**4 found**” (Figure 14.34). Hit **ENTER** and the shape of the four cylinders will disappear from the master solid as shown in Figure 14.35.

Moving the Solid to the Origin Point

To move the master solid 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 center Osnap of the top face of the part as shown in Figure 14.36, 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. Revolving Problem Three 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 14.36 – Moving to the origin

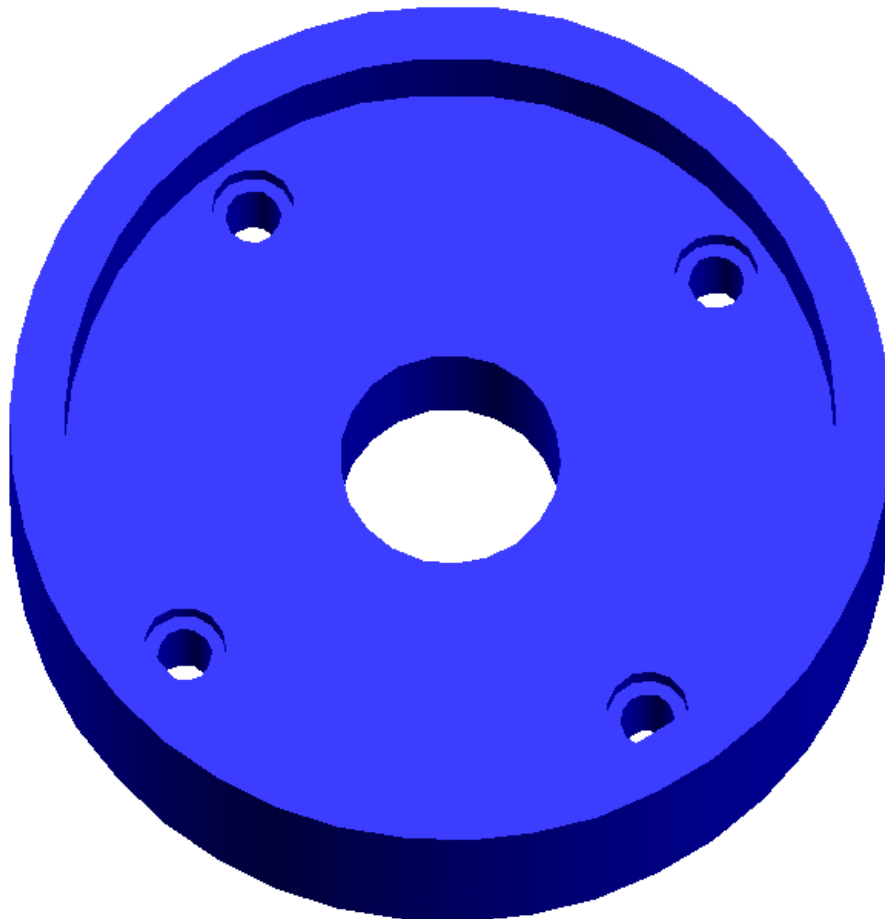


Figure 14.37 – The finished part

Saving the Solid Problem

To save Revolving Problem Three 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 “Revolving Problem 3” in the File Name textbox and hit the Save button to save the drawing (Figure 14.38).

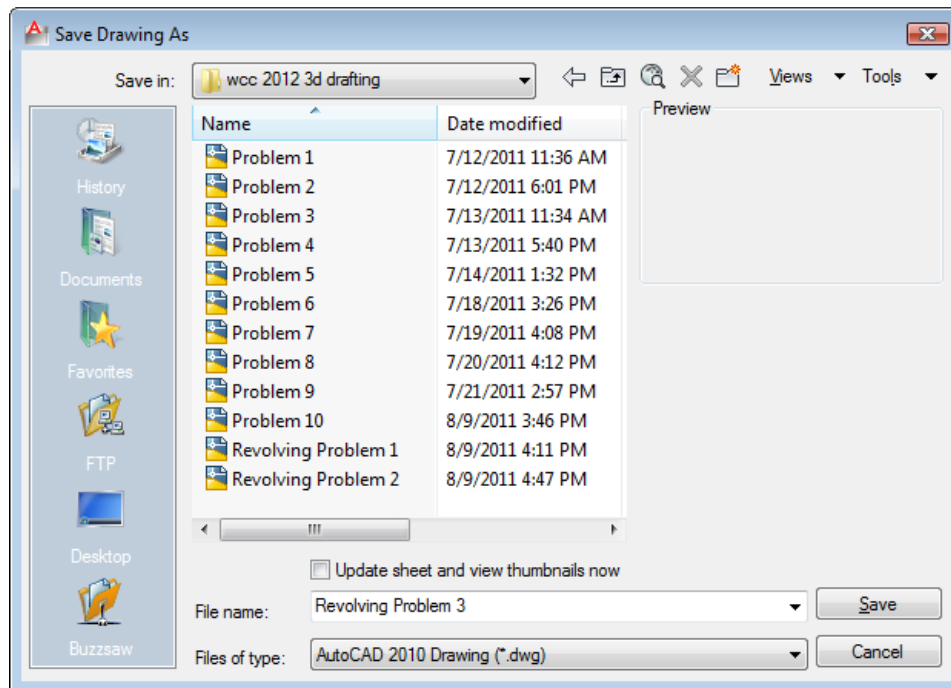


Figure 14.38 – Saving Revolving Problem Two

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

*** World Class CAD Challenge 03-13 * - 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.**