

Chapter 14

Revolving Problem Three

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

- 1. Sketch of Revolving Problem Three**
- 2. Starting a 3D Part Drawing**
- 3. Modifying How the UCS Icon is Displayed**
- 4. Creating Center and Perimeter Lines Using the Line Command**
- 5. Creating a Joined Perimeter Using the Edit Polyline Command**
- 6. Revolve a Solid from a Closed Polyline**
- 7. Shade a 3D Solid**
- 8. Rotating a 3D Solid Using the Rotate3D Command**
- 9. Remove the Center Line Using the Erase Command**
- 10. Drawing Two Cylinders**
- 11. Moving a Solid into Position on the Master Solid**
- 12. Array a Solid Using the Array Command**
- 13. Subtracting 3D Solids from the Master Solid**
- 14. Moving the Solid to the Origin Point**
- 15. Saving the Solid Problem**

Starting a 3D Part Drawing

In revolving problem three, you will begin the 3D drawing by selecting the Mechanical template. In Figure 14.2, the window “Select template” is loaded by selecting the New tool on the Standard Toolbar. 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.

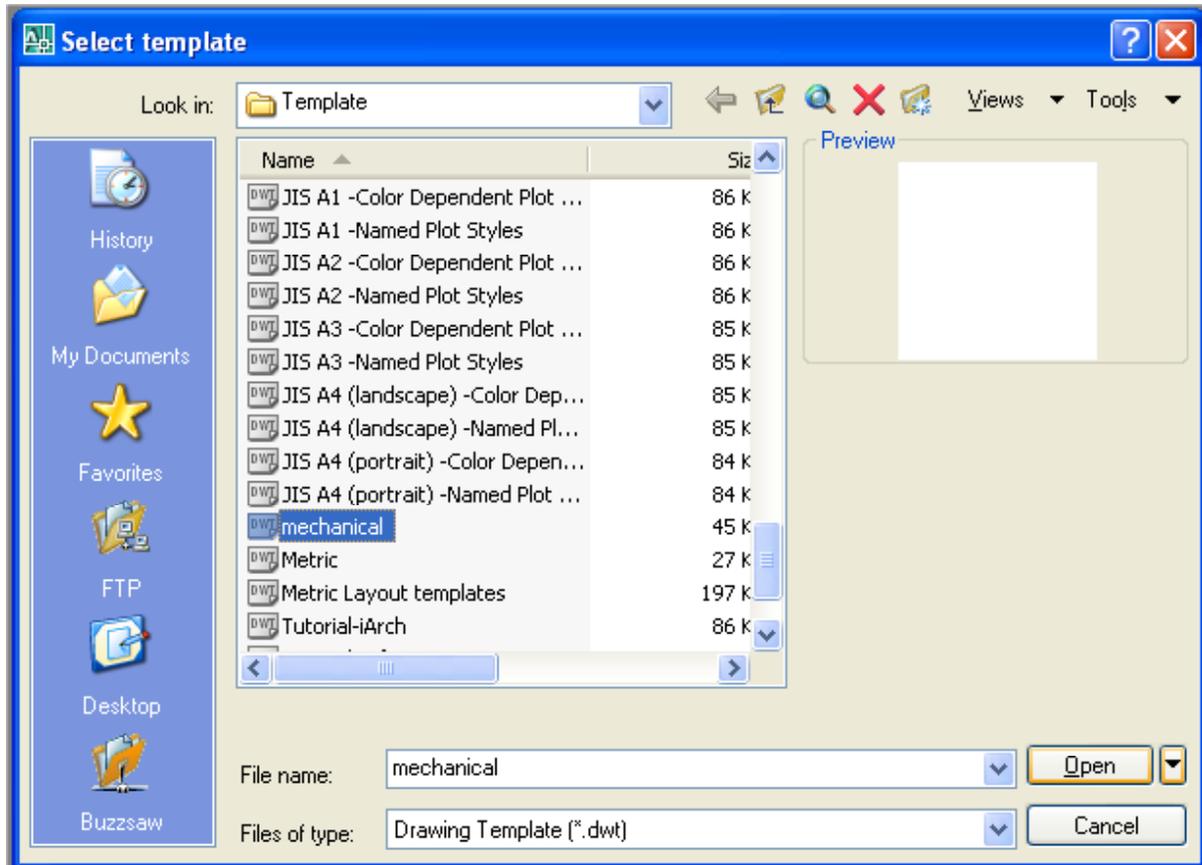


Figure 14.2 – Starting the Drawing Using the Mechanical Template

Modifying How the UCS Icon is Displayed

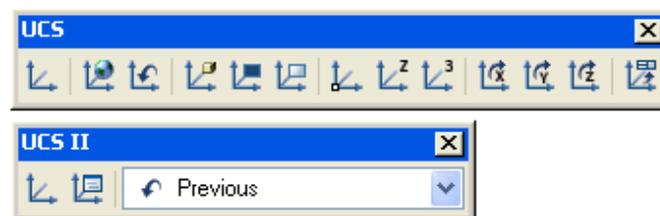


Figure 14.3 - The UCS Toolbar

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 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 Display UCS Dialog tool on the UCS II toolbar (Figure 14.3) and the UCS window will appear the graphical display. Pick the Settings tab to change the UCS Icon settings. Remove the check from “Display at UCS origin point” as shown in Figure 14.4 and press the OK button to have the system variable change. Now you can start to create the solid.

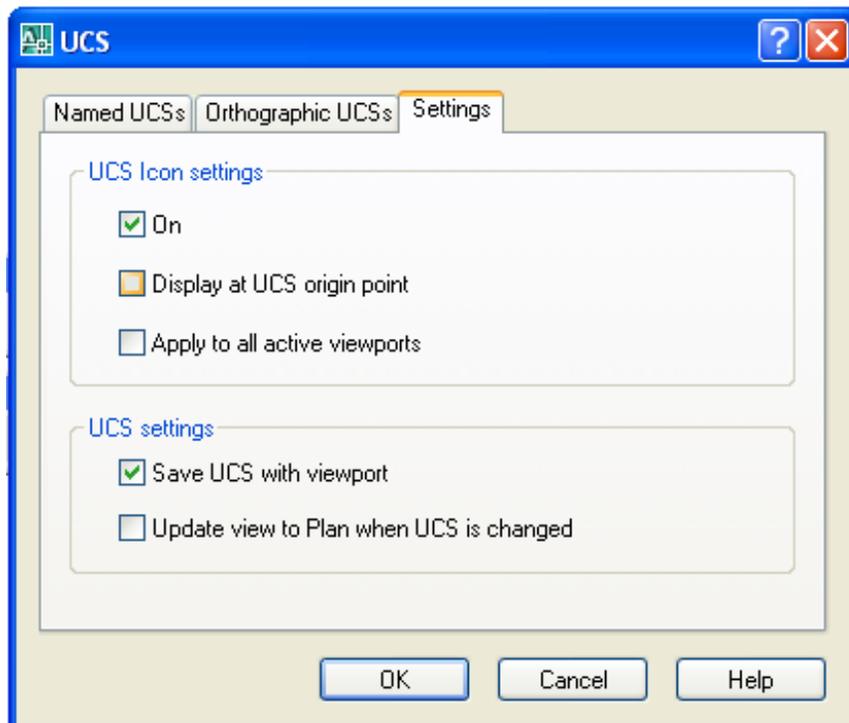
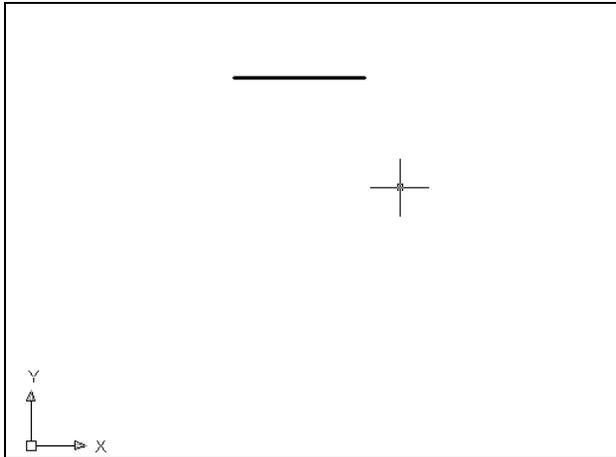


Figure 14.4 - The UCS Window, Settings Tab

Creating a 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 toolbar to start drawing the detail perimeter of the problem.





With your mouse, select any point at the upper right side of the graphical display. Even while in the Line command, continue with the Ortho mode On to keep the lines perfectly horizontal when drawing. The line will appear on your graphical display as shown in Figure 14.5. Press **Enter** to end the line command. Now to get to the second line, you will use a relative coordinate by placing the “@” symbol in front of the X and Y coordinate to move the starting point of the next line 1.0 inch downward.

Figure 14.5 – Draw a Center Line

Enter to repeat the Line command. You will still need to start the new line 1.0 below the last point drawn. The method to accomplish that task is to place an “@” symbol in front of the coordinate, so the starting point will move so far in the X direction and a distance in the Y direction. Pick the Line tool on the Draw toolbar. At the prompt for the starting point, type **@0,-1.0** and **Enter**. The line will begin 1.0 below the centerline. The command line information is shown in Figure 12.6.

```
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 – Draw a Series of Lines

With the Ortho mode “on”, draw a **1.50** 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 right, a **0.5** line upward, a **1.0** line to the right and then type “**C**” to close the line. The AutoCAD program 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 entity of the final shape will appear in the graphical display as shown in Figure 14.13. Select the Zoom Extents tool on the Standard toolbar to have the box fill the monitor, so you can proceed to the next step in the drawing process.

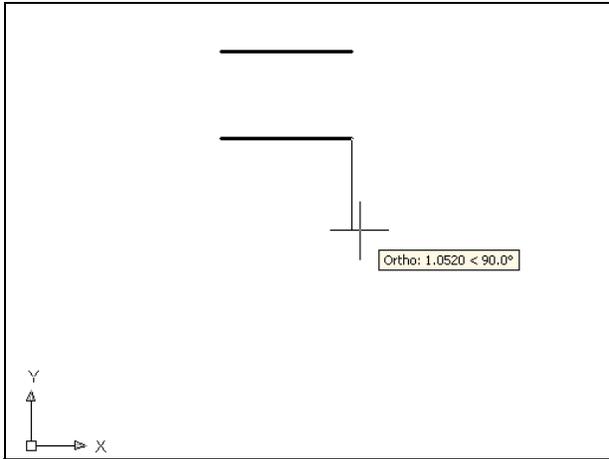


Figure 14.7 – Draw a Line 1.5 to the Right

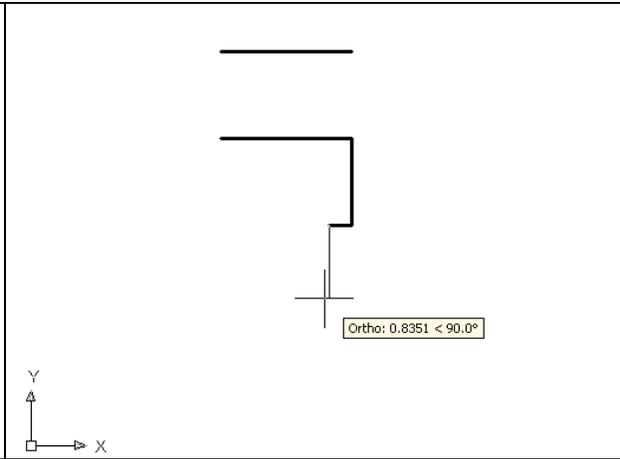


Figure 14.8 – Lines 1.0 Down and 0.25 Left

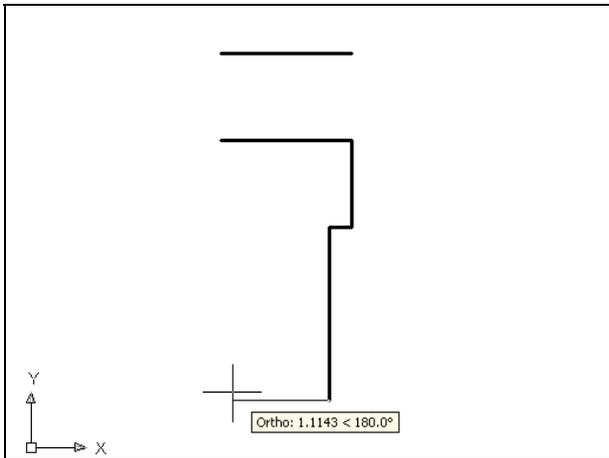


Figure 14.9 – Draw a Line 2.0 Downward

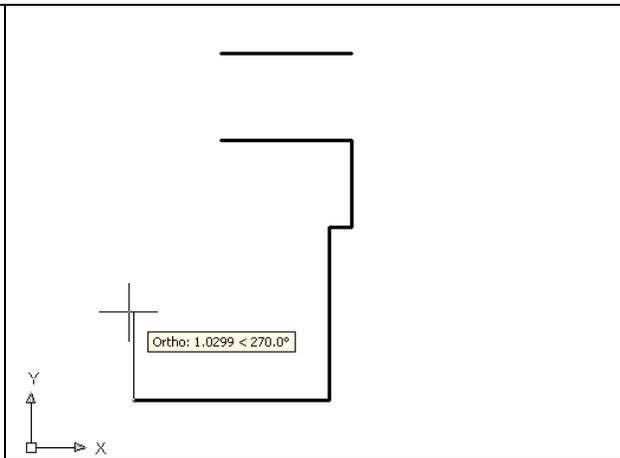


Figure 14.10 – Draw a Line 2.25 Left

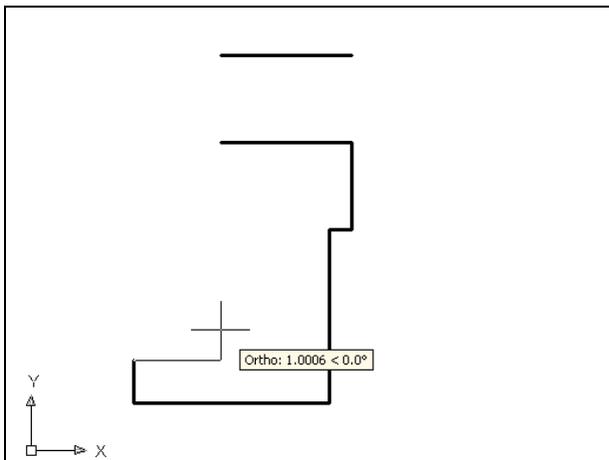


Figure 14.11 – Draw a Line 0.50 Upward

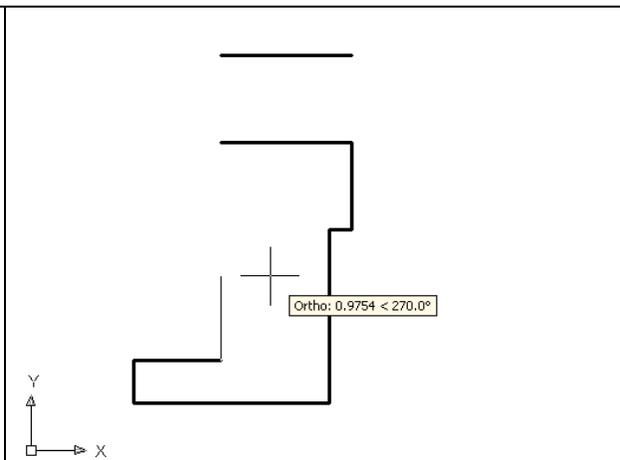


Figure 14.12 – Draw a Line 1.0 Right

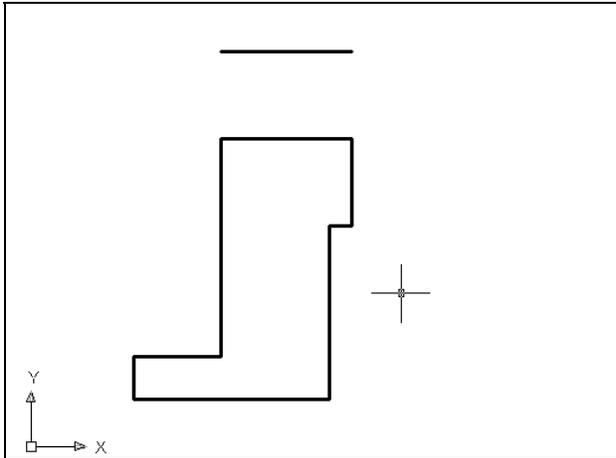


Figure 14.13 – Close the Series of Line

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 is 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 will be your favorites.

Creating a Joined Perimeter Using the Edit Polyline Tool

To merge the lines and arc segments together you need to use the Edit Polyline command. Select the Edit Polyline tool on the Modify II toolbar and the command line on the AutoCAD application will prompt you to “select objects”.

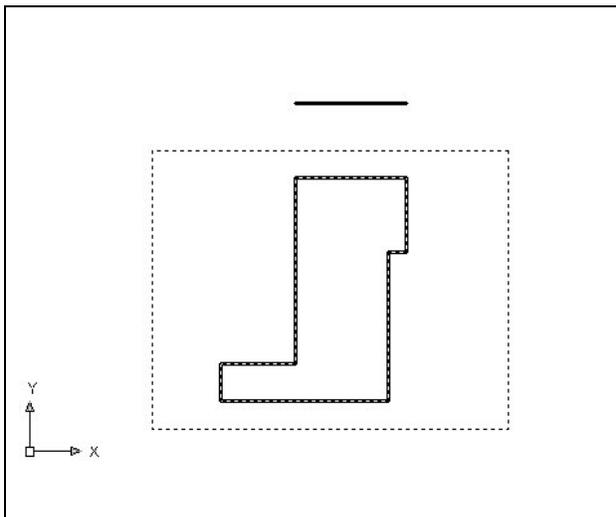


Figure 14.14 – Making a Complex Polyline

You will be prompted to “Select polyline or [Multiple]: Select polyline or [Multiple].” Type “M” and **Enter** for multiple. Pick to the upper right of the graphical display as shown in Figure 14.14 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. Hit **Enter** and **Esc** to exit the command.

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 on the Solids toolbar and the command line on the AutoCAD application will prompt you to “select objects”. Pick the perimeter of the Polyline and the

command line will return with “1 found” as shown in Figure 14.15. Next, you choice two points to describe the axis of rotation. Pick the Endpoint of the centerline as shown in Figure 14.16 for the first point. Pick the Endpoint of the centerline as shown in Figure 14.17 for the second point.

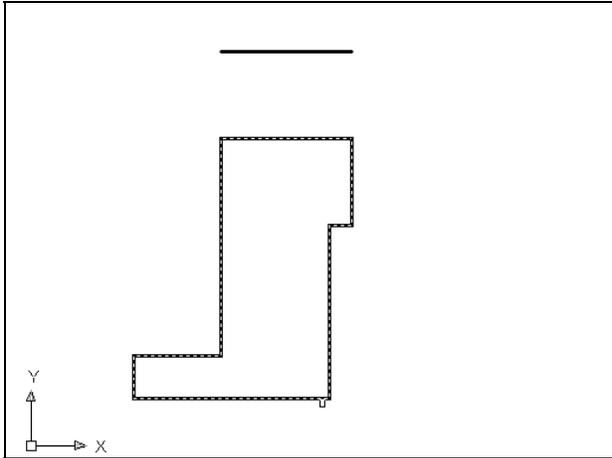


Figure 14.15 – Select the Polyline

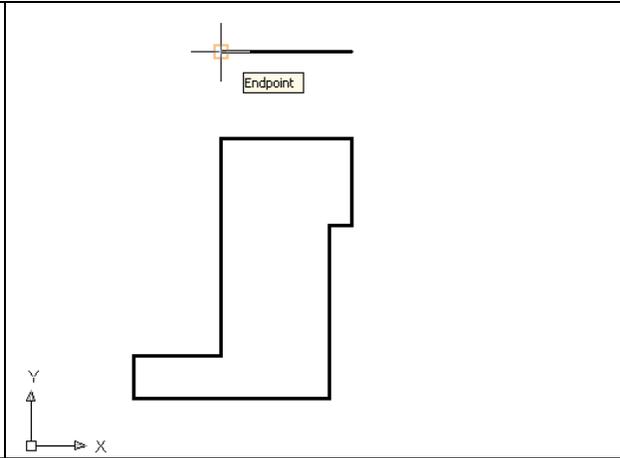


Figure 14.16 – The First Point of the Axis

After defining the axis of rotation, Hit enter to select 360 as the degree of rotation. A new master solid will appear on the graphical display. On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane and the vertical Z-axis will rise upward as shown in Figure 14.18. Again the tessellation lines on the screen can be confusing, so you may wish to proceed to the next step of this exercise where you will color the entire surface of the master solid.

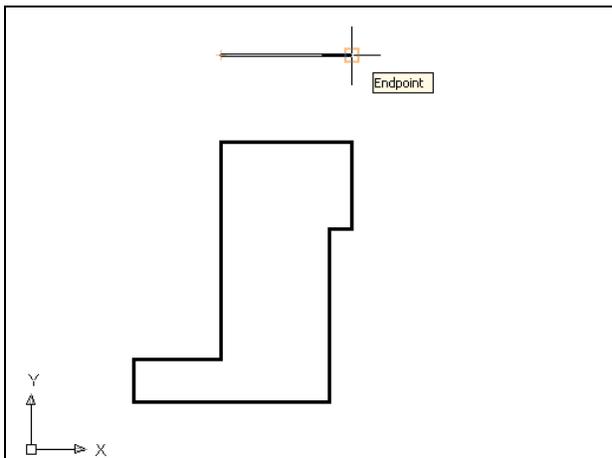


Figure 14.17 – Second Point of the Axis

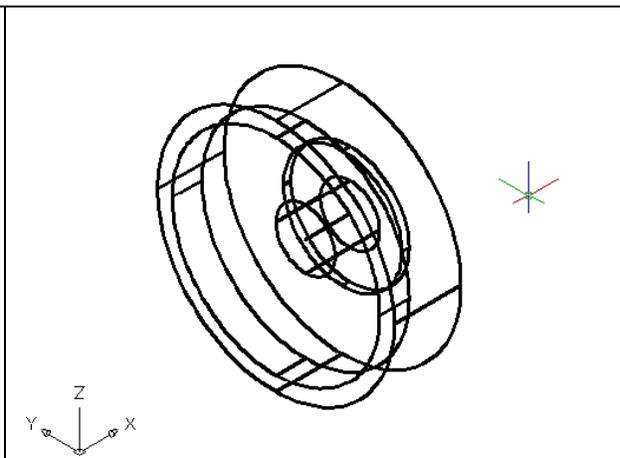


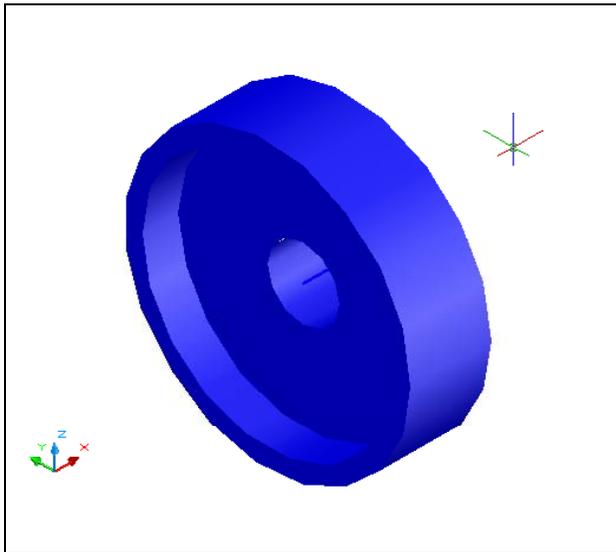
Figure 14.18 – The Revolved Solid

Shade the 3D Solid

To view the 3D solid with holes in a better means, select one of the tools on the Visual Styles toolbar. The solid will acquire a painted look matching the color of the layer,



which the part resides. Make the layer “0” color blue, so you can easily see the shaded part.



Select the Realistic Visual Style tool on the Visual Styles toolbar. The part is now a solid blue color as shown in Figure 13.26. 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 since they can visualize the product clearly. If you ever desire to return to standard viewing, choose the 2D Wireframe tool on the Visual Styles toolbar to return the drawing to the previous settings. These images were taken using a modified visual style, created by selecting the Manage Visual Styles tool on the Visual Styles toolbar.

Figure 14.19 – Shade the Revolved Solid

Rotating a 3D Solid Using the Rotate3D Command

Select Modify on the Menu Bar, then 3D Operations and pick Rotate 3D from the list of commands. At the command, “select objects”, pick the solid on the graphical display and hit **Enter** to proceed to the second part of the function. You are going to rotate the solid part on the y-axis, so pick the bottom quadrant of the part and select the y-axis diameter from the globe. With the Ortho mode still “on” pick a second point drawing a line on the y-axis as shown in Figure 14.20. Type **90** to rotate the solid as shown in Figure 14.21.

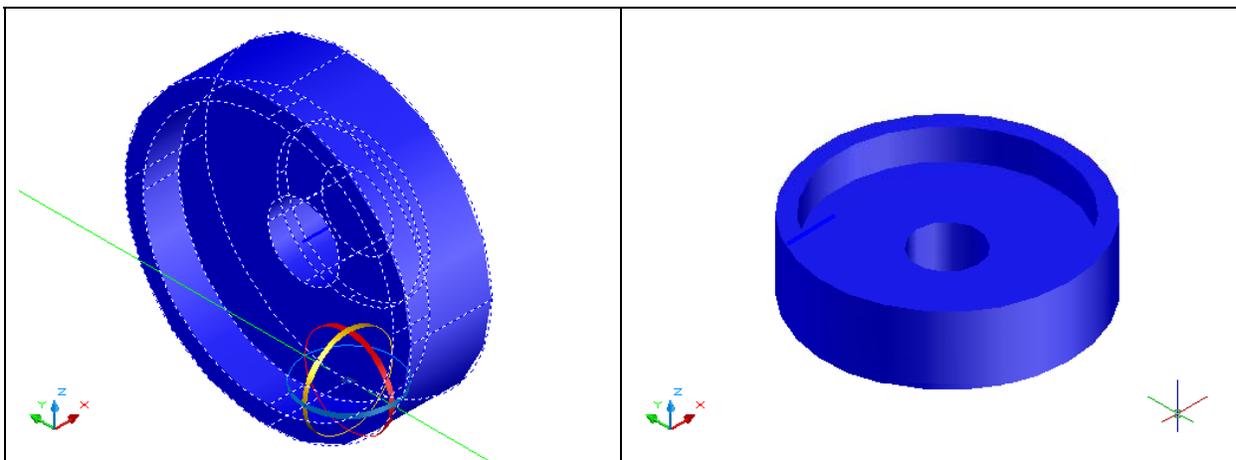
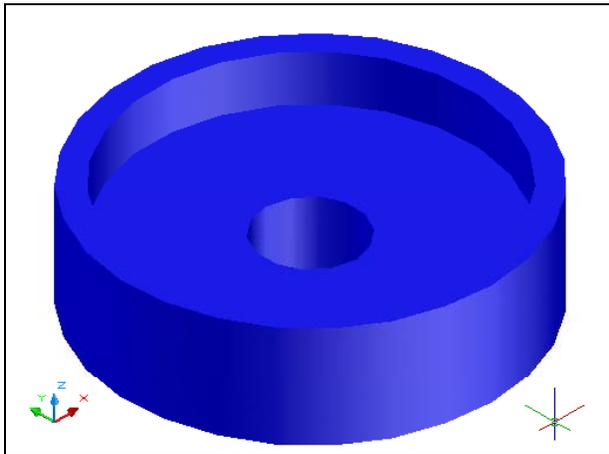


Figure 14.20 – The Rotation Axis

Figure 14.21 – The Rotated the Solid

Removing the Center Line Using the Erase Command

Again, do not excessively rely on the erase function since removing entities once drawn is losing efficiency. In many revolved solids, the closed Polyline can have points that you will use as a turning point, but when you want a hole in the middle of the master solid, use a centerline to reference as the axis.



Use the Erase tool on the Modify toolbar to remove the centerline that you no longer need. Select the Erase tool on the Modify toolbar, and select the entity. Hit **Enter** to remove the centerline on the front side as shown in Figure 14.22. You may have to move the master solid if the line entity is inside or behind the larger solid. You will eventually move the master solid to the origin at the end of the exercise.

Figure 14.22 – Erase the Center Line

Creating a New Layer in an AutoCAD Drawing

Many times you may wish to add a special layer so that you can control the formation of the solid part by freezing or thawing the desired entity which is on a layer. In this exercise you need to create a new layer which you will call “Cylinder” so you can have that extra measure of control. To create the “Cylinder” layer using the Layers Properties Manager, select the Layers command button to the left of the Layers Selection list box. Add the “Cylinder” layer using the New button in the Layer Properties Manager. Make the layer color “red”.



Drawing Two Cylinders

You need to add four holes to the master solid, one hole going completely through the master solid, and the second through fourth holes will be a copy of the first after you perform a polar array. These openings will not be straight, but will be at an angle for ease of casting.



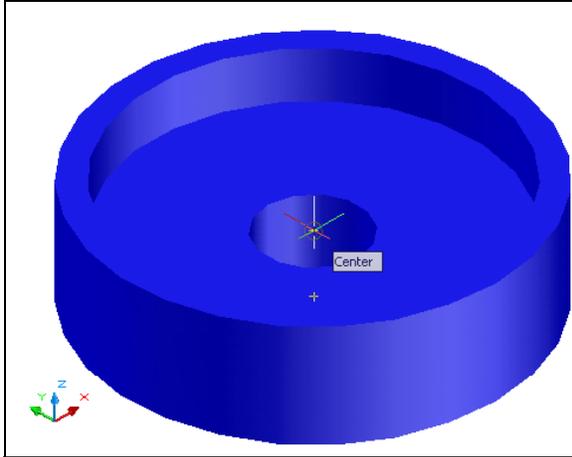


Figure 14.23 – Center of a Cylinder

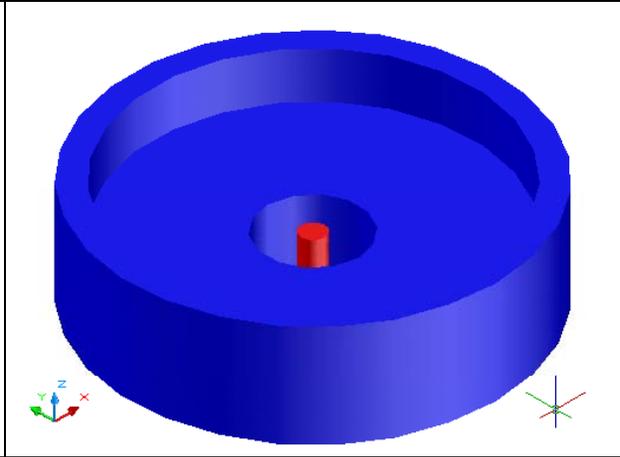


Figure 14.24 – The Cylinder

On the Draw toolbar, pick the Cylinder tool. Specify the center point for the top of the cylinder as the center of the master solid as shown in Figure 14.23. You need to type “D” for diameter and **Enter** to input the sketch’s measurement of **0.5**. Hit **Enter** and then type **-1.5** as the height of the cylinder. The Cylinder appears in the graphical display as shown in Figure 14.24.

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**. The second cylinder will appear just like the one in Figure 14.26.

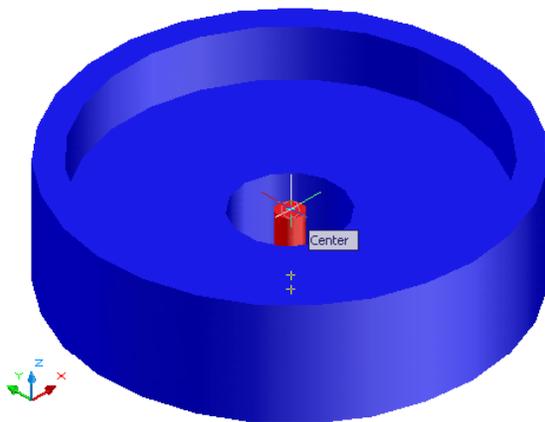


Figure 14.25 – Center of Second Cylinder

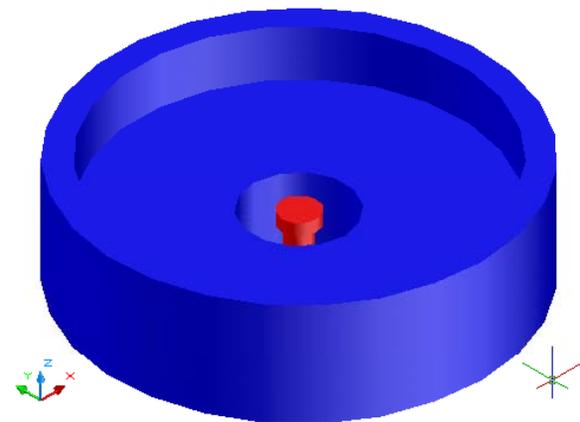


Figure 14.26 – The Second Cylinder

Union Two Solids

To combine both cylinders into a single 3D solid shape in AutoCAD, you need to use the Union command.



To add the two solids together, select the Union tool on the Solids Editing toolbar. AutoCAD

will prompt you with “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 14.27. 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.

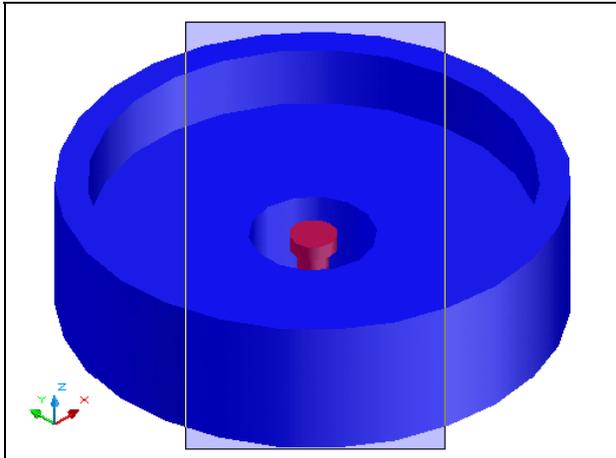


Figure 14.27 – Select Both Cylinders

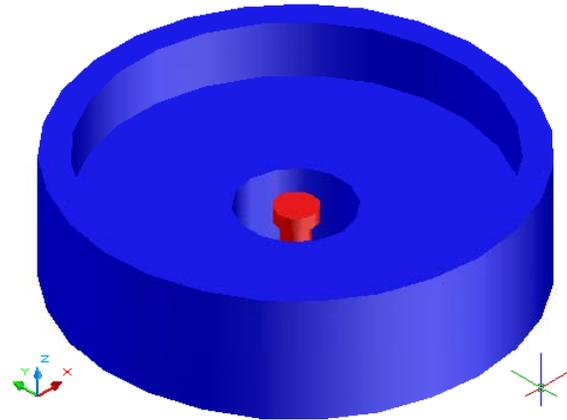


Figure 14.28 – The Cylinders Are Joined

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.

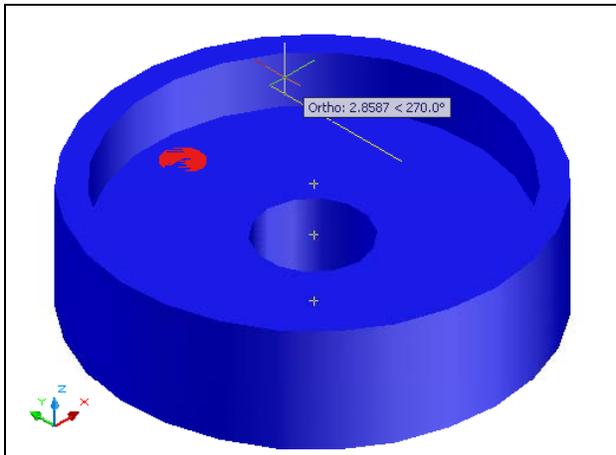


Figure 14.29 – Moving the Solid

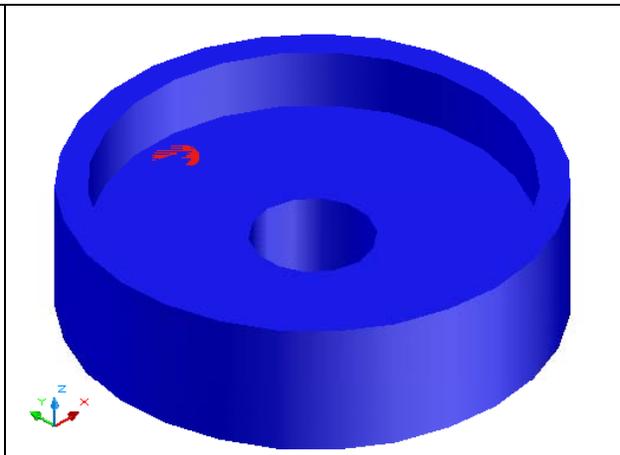


Figure 14.30 –The Solid in Position

To move the cylinder, you select the Move tool on the Modify toolbar. Once you choose the Move command, you will be prompted to “Select Objects”. Pick the cylinder 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”, which will be anywhere on the graphical display as shown in Figure 14.35. For the “second point of displacement”, move the cursor to the right with the Ortho “On”, type **3** and **Enter**. The cylinder will be ready to polar array as shown in Figure 14.36.

Using the Array Command to Create Circular Patterns

You now need to copy and rotate the solid cylinder, spacing each one 60 degrees apart. Use the Array function with the polar option as the tool for this operation. 

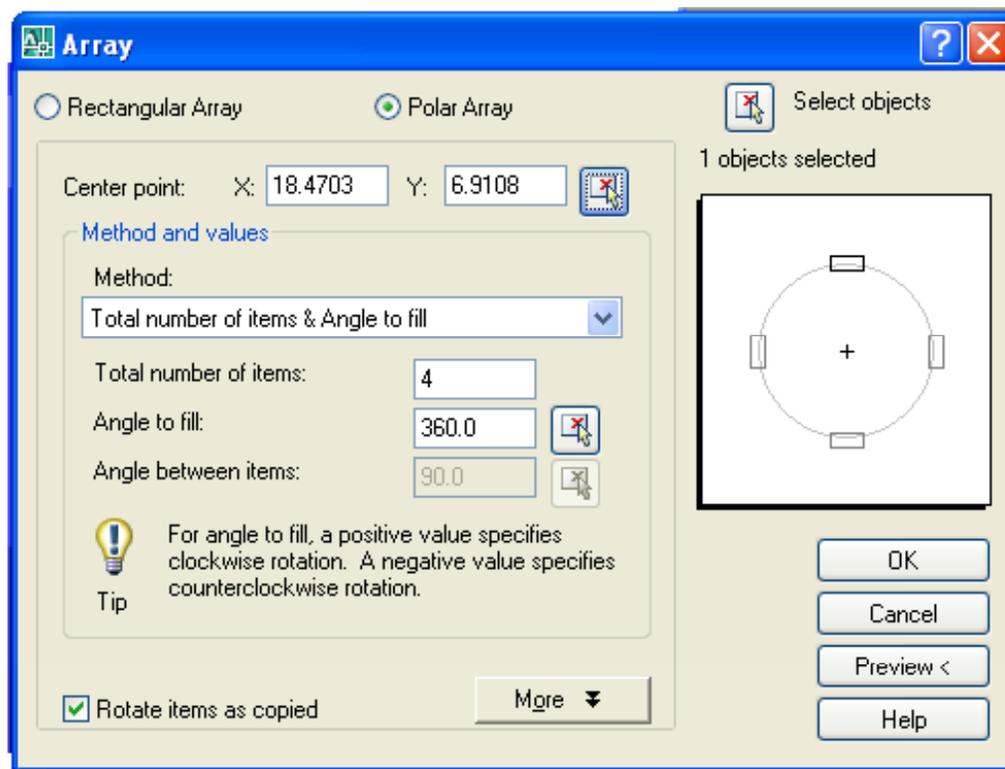


Figure 14.31 – The Array Window

Select the Array tool on the Modify toolbar, which will cause the Array window as shown in Figure 14.37 to appear. Pick the Polar radial button, then the Select Objects button that will allow you to select the tapered cylinder as shown in Figure 14.32 and **Enter** to return to the Array window. Next, select the Pick Center Point Button, which brings you back to the drawing to select the center of the part as shown in Figure 14.33.

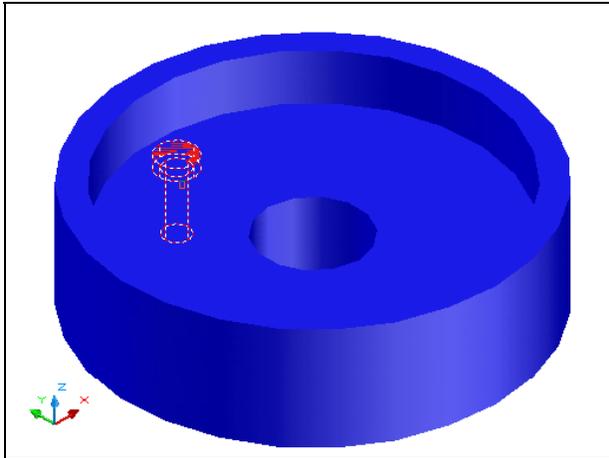


Figure 14.32 –Select Object

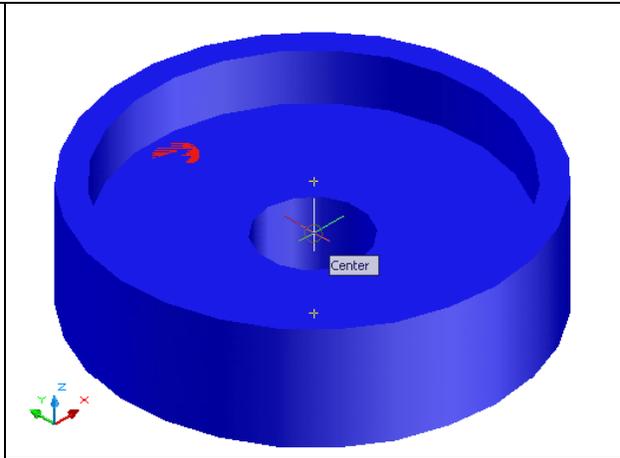


Figure 14.33 – Center of the Polar Array

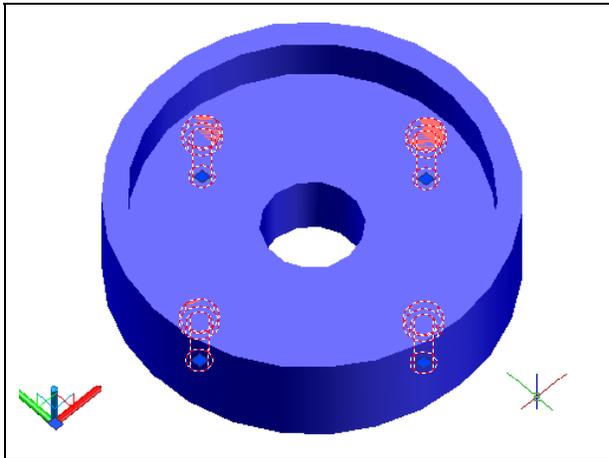


Figure 14.34 – Six Solids to Subtract

Pick the center of the part by selecting the Center of the master solid that is the heart of the part. Change the total number textbox to “4” and select the Preview button. The array will be shown on the display screen, along with an Array Accept, Modify or Cancel window. If the Circular problem looks like Figure 14.34, press the Accept button and the array will be completed. If the Circular problem does not look like Figure 14.40, press the Modify button repeat the array process.

Subtracting 3D Solids from the Master Solids

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



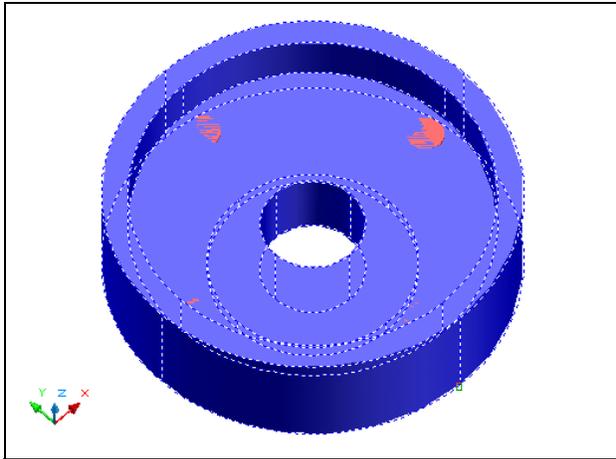


Figure 14.35 – Subtract Solids

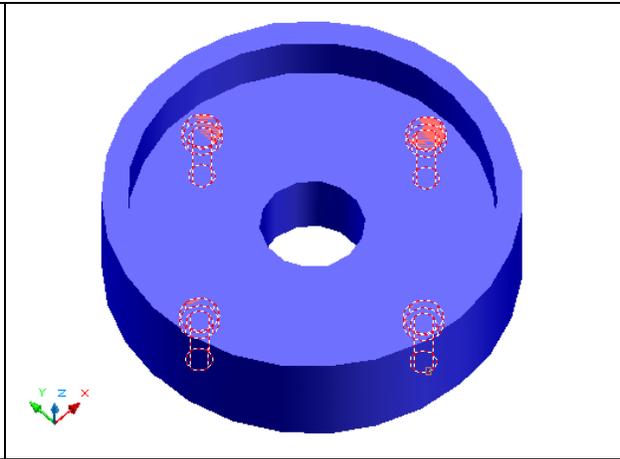


Figure 14.36 – Subtracting Solids

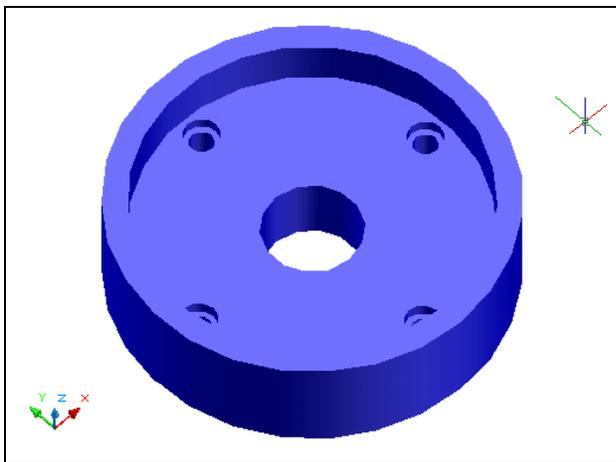
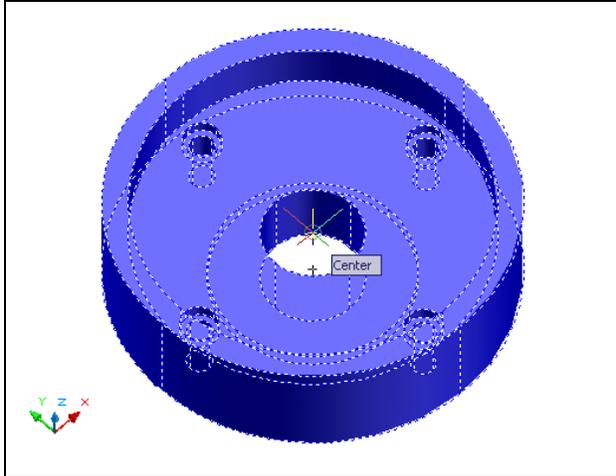


Figure 14.37 – Master Solid with Holes

AutoCAD will prompt you to “Select solids and regions to subtract from”. Select the master solid as shown in Figure 14.35 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 four cylinders as shown in Figure 14.36 and the software application will counter with “4 found”. Hit **Enter** and a region exactly the shape of the four cylinders is missing from the larger one as shown in Figure 14.37.

Moving the Solid to the Origin Point

To move revolving problem three to the drawing origin of 0,0,0 on the x, y and z-axis, pick the Move tool on the Modify toolbar, select the 3D solid, and hit Enter.



At the prompt, “**specify base point or displacement**”, select the endpoint shown in Figure 14.39 at the top center of the revolving problem three, which will become the problem three’s insertion point into an assembly drawing. At the “second point of displacement”, type **0,0,0** and **Enter**. Problem three 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 14.39 – Moving to the Origin

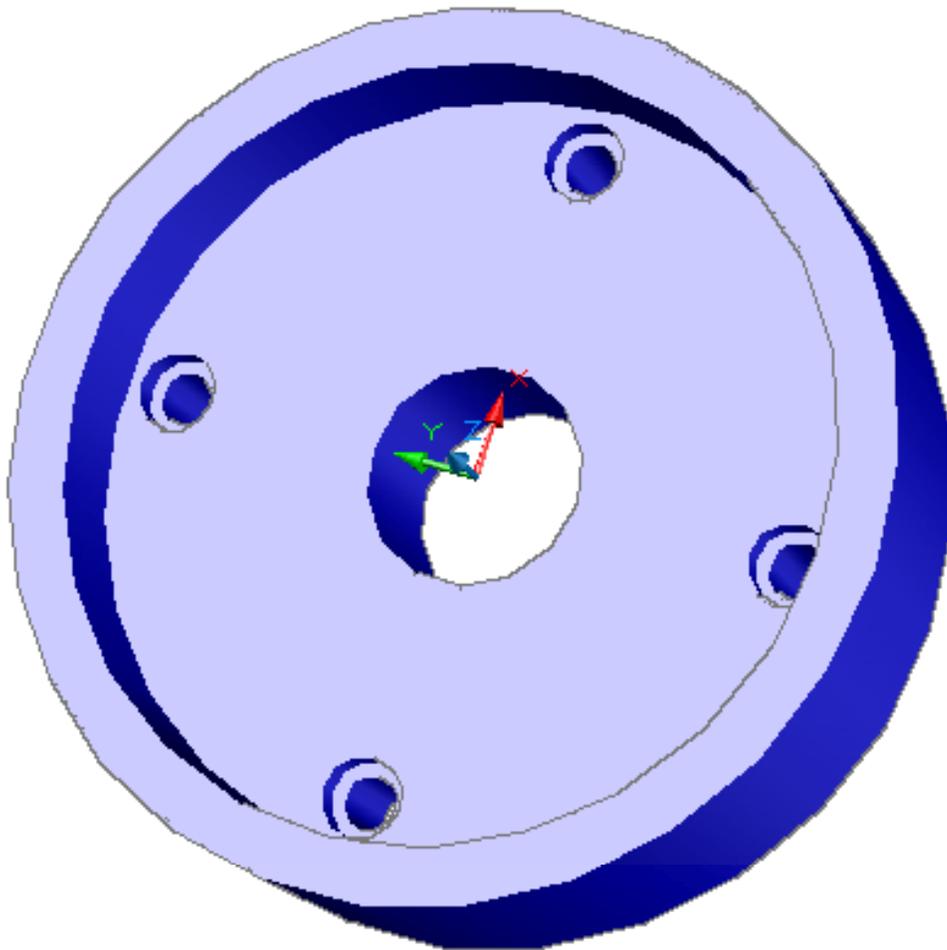


Figure 14.40 – The Finished Part

*** 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 under 5 minutes to maintain your World Class ranking.

Saving the Solid Problem

To save Revolving Problem 3 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 “Revolving Problem 3” and press the Save button to save the drawing. (See Figure 14.41)

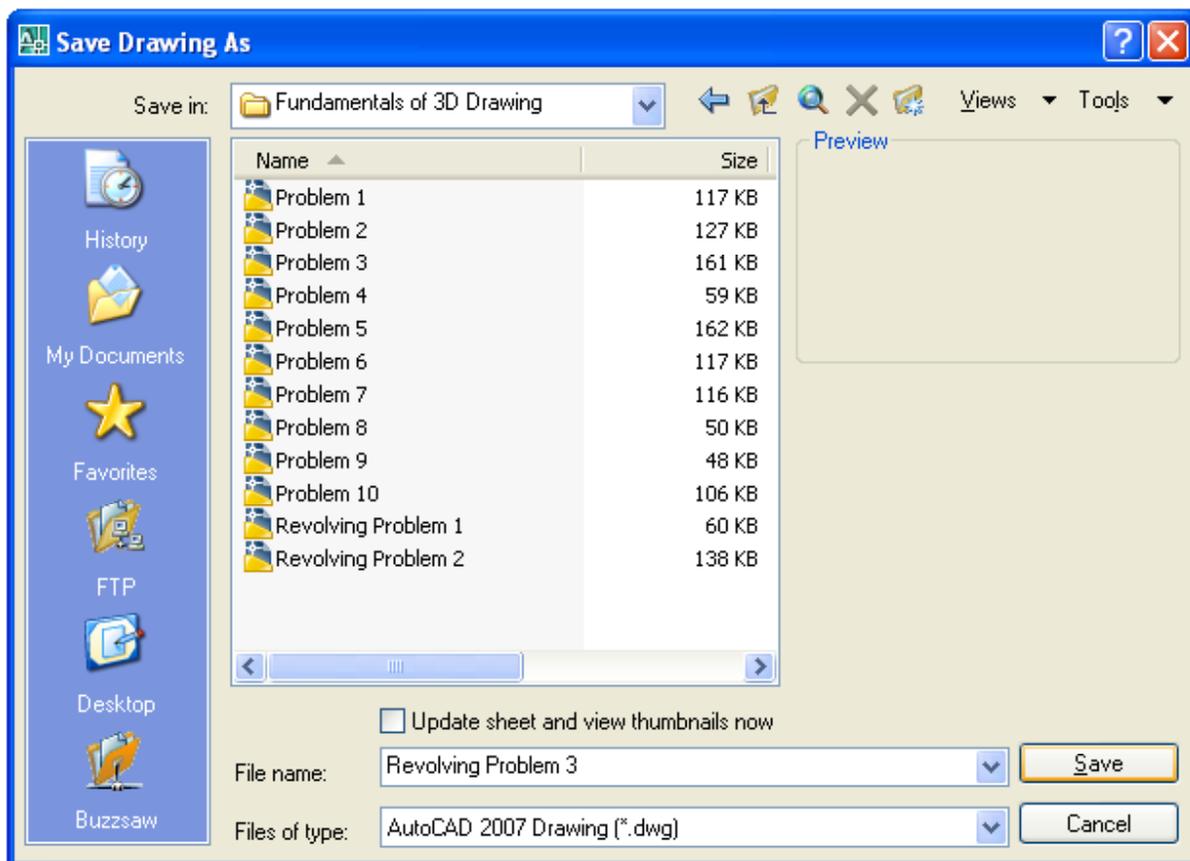


Figure 14.41 – Saving Revolving Problem Two

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