

Chapter 13

Revolving Problem Two

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

- Sketch of Revolving Problem Two
- Starting a 3D Part Drawing
- Modifying How the UCS Icon is Displayed
- Creating Center and Mirror Lines Using the Line Command
- Using the Offset Command to Create New Lines
- Adding Two Lines Using the Line Command
- Remove the Four Lines Using the Erase Command
- Add Curved Edges Using the Fillet Command
- Rotate Two Lines Using the Rotate Command
- Add Two Additional Curved Edges Using the Fillet Command
- Add Zero Radius Edges Using the Fillet Command
- Creating a Perimeter Using the Polyline Command
- Revolve a Solid from a Closed Polyline
- Shade a 3D Solid
- Rotating a 3D Solid Using the Rotate3D Command
- Remove the Center Line Using the Erase Command
- Drawing a Circle to be Extruded at a Taper
- Extruding a Solid from Closed Polylines and Circles
- Moving a Solid into Position on the Master Solid
- Array a Solid Using the Array Command
- Subtracting 3D Solids from the Master Solid
- Moving the Solid to the Origin Point
- Adding the Other Side Using Mirror3D Command
- Saving the Solid Problem

Sketch of Revolving Problem Two

You will start the second revolving problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across the Z-axis and X-axis. In this exercise, a strategy you will employ is to draw and revolve a closed Polyline that contains half of the most complex shape of the component and then use simple geometric solids to subtract each detail. These exercises in revolving problem two involve you drawing just a portion of the component, revolving the section, and after detailing, mirroring the second half and combining them together. In problem two, the perimeter contains radius edges in the main shape, where you get a chance to use the Fillet tool. After revolving the Polyline perimeter into a solid, you will place a tapered and extruded solid onto that main solid, array the detail and subtract them. You will utilize the Fillet command once more and put a rounded entry on the six mounting holes. Then finally, you will use a new command, Mirror3D to copy the master solid across the plane and union them together to finish the challenge.

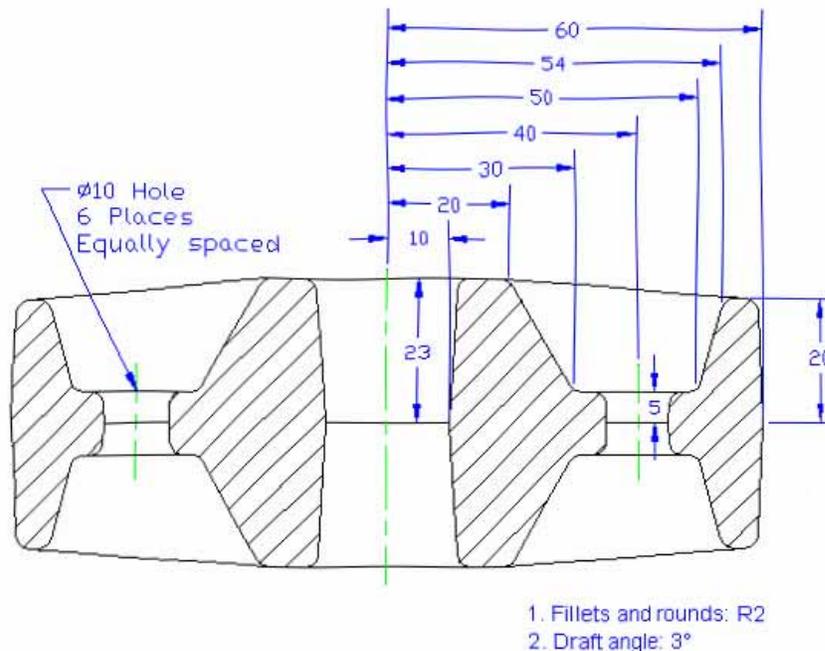


Figure 13.1 – Revolving Problem Two Sketch

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. When you obtain a circular solid, the revolving of a closed Polyline is your most probable action. Place as much detail in the perimeter and interior detail, so you will finish the master solid quickly, then only leaving a few features to finish the exercise. This problem will help you address solid parts where there are different complexities and sizes. All three revolving 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 revolving problem two, you will begin the 3D drawing by selecting the Metric template. In Figure 13.2, the window “Select template” is loaded by selecting the New tool on the Quick Access Toolbar. 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.

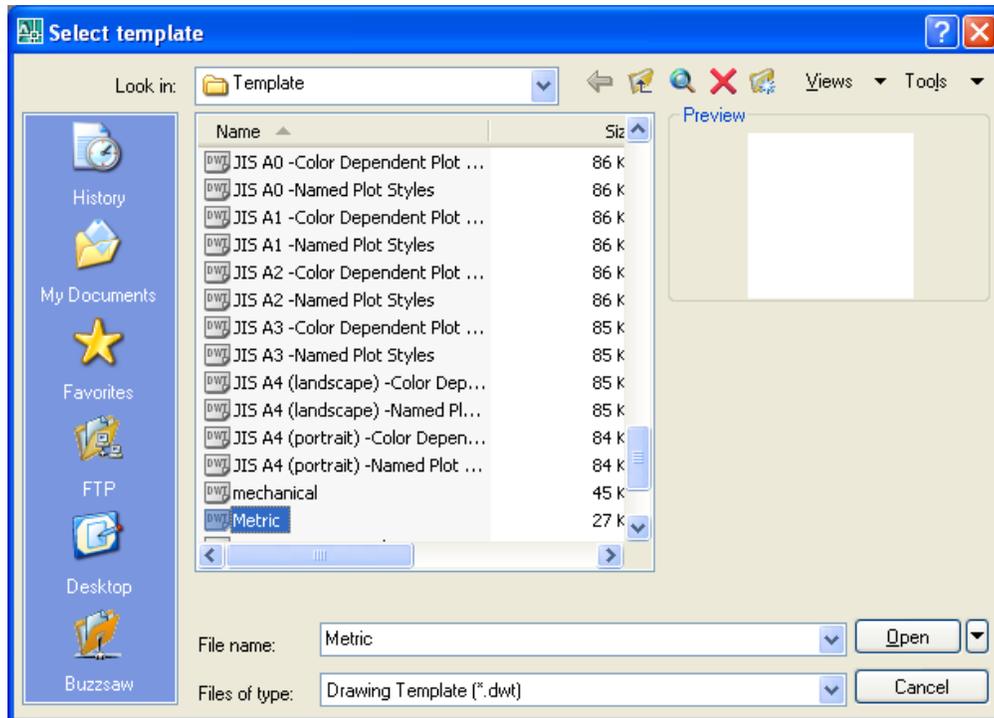


Figure 13.2 – Starting the Drawing Using the Metric Template

Modifying How the UCS Icon is Displayed

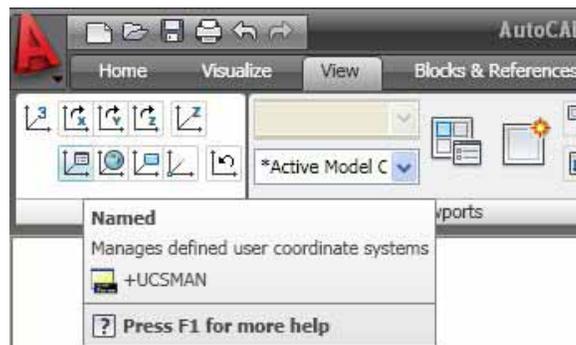


Figure 7.3 - The UCS Settings

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 under the View tab and the UCS window will appear in 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 7.4 and press the OK button to have the system variable change. Now you can start to create the solid.

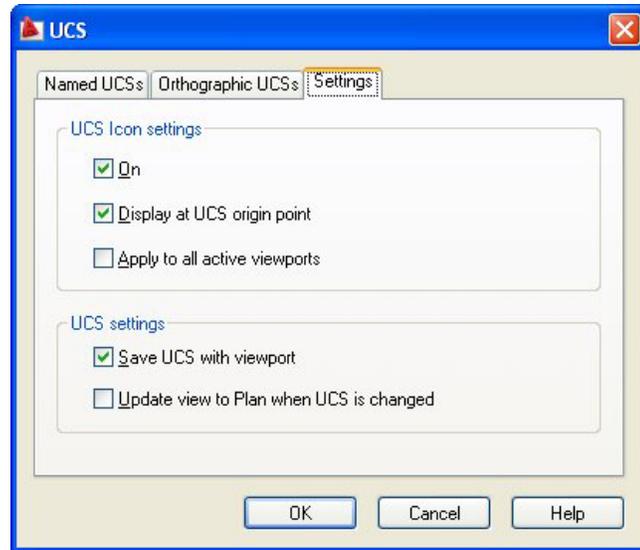


Figure 7.4 - The UCS Window, Settings Tab

Creating a Center and Mirror Lines Using the Line Command

You will draw a 23 line from right to left and then 60 down to create a center and mirror line.

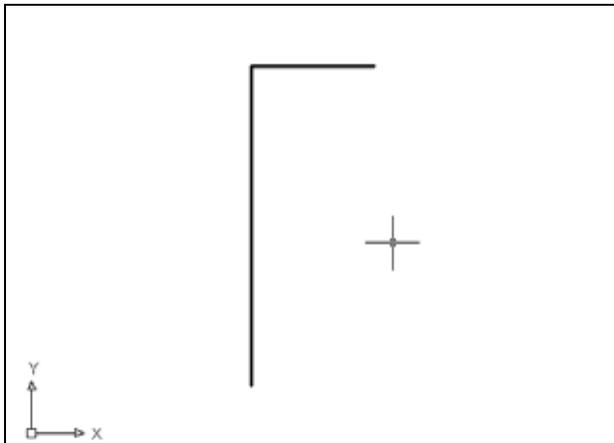


Figure 13.5 – Draw Two Lines

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. You then move the line on the graphic screen to the left and type **23**, then change direction with the mouse downward and type **60** and **Enter** at the keyboard to complete the function. Two lines will be on your graphical display as shown in Figure 13.5.

Using the Offset Command to Create New Lines

To create additional lines to construct the perimeter, you need to offset lines at different values to create a matrix to form the outline. This is not the first time you have done this drill, because the shaping of the boundary is a basic concept in the World Class CAD's Fundamental 2D Drawing training module.



To create a new line 10 below the horizontal centerline 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 which you will type **10** and **Enter**. Now select the horizontal centerline and pick below that line to create a new line 10 below the first. Again select the new line and you pick below that line to create another. Do this process one more time to generate a third line below the centerline and hit **Enter**.

To create a new line 20 below the last line, hit **Enter** to repeat Offset command. The command line will prompt you for the offset distance which you will type **20** and **Enter**. Now select the last line and pick below that line to create a new line 20 below the one selected and hit **Enter**.

To create another horizontal line 10 below the last line, hit **Enter** to repeat Offset command. The command line will prompt you for the offset distance which you will type **10** and **Enter**. Now select the last line and pick below that line to create a new line 10 below the one selected and hit **Enter**. All the lines will appear as shown in Figure 13.6.

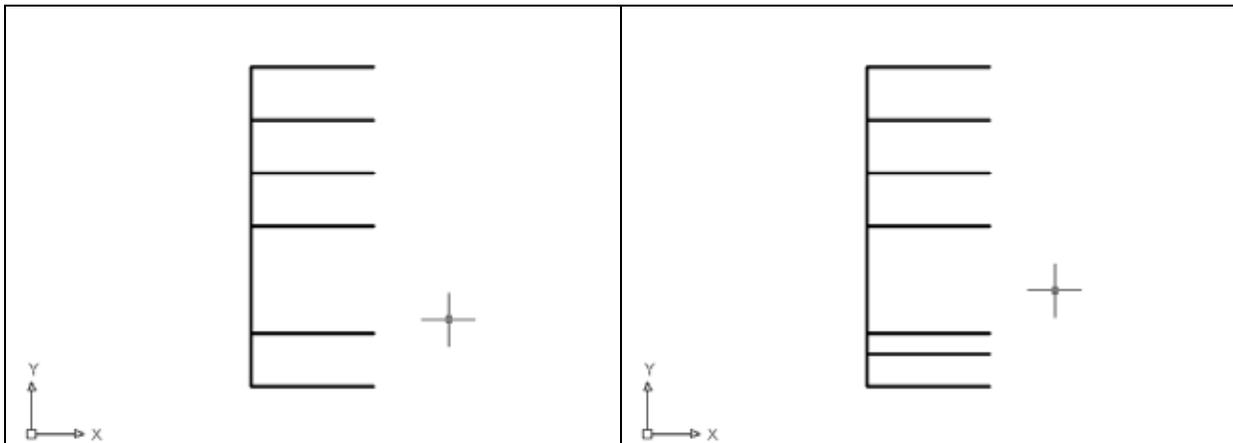


Figure 13.6 – Offset Lines by 10

Figure 13.7 – Offset a Line by 6

To create the last horizontal line 6 above the last line, hit **Enter** to repeat Offset command. The command line will prompt you for the offset distance. Type **6** and hit **Enter**. Now select the last line and pick above that line to create a new line 6 above the one selected and hit **Enter**. The line will appear as shown in Figure 13.7.

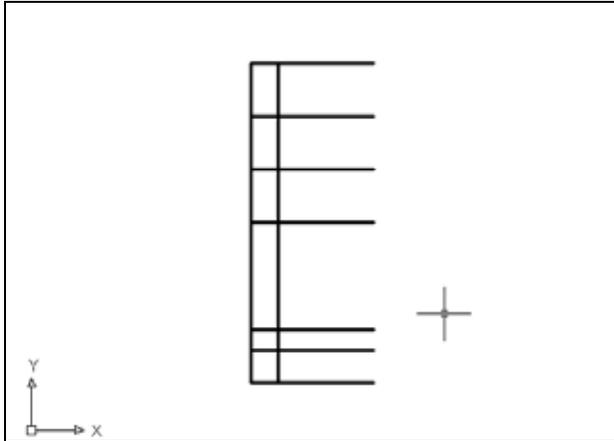


Figure 13.8 – Offset a Line by 5

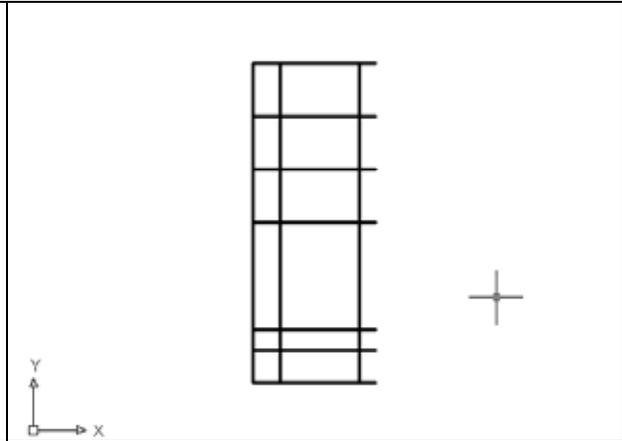


Figure 13.9 – Offset a Line by 20

To create a new vertical line 5 to the right of the vertical mirror line, hit **Enter** to repeat Offset command. The command line will prompt you for the offset distance, so type **5** and hit **Enter**. Now select the vertical mirror line and pick to the right of that line to create a new line 5 to the right of the one selected and hit **Enter**. The line will appear as shown in Figure 13.8.

To create a new vertical line 20 to the right of the vertical mirror line, hit **Enter** to repeat Offset command. The command line will prompt you for the offset distance. Type **20** and hit **Enter**. Now select the vertical mirror line and pick to the right of that line to create a new line and hit **Enter**. The line will appear as shown in Figure 13.9.

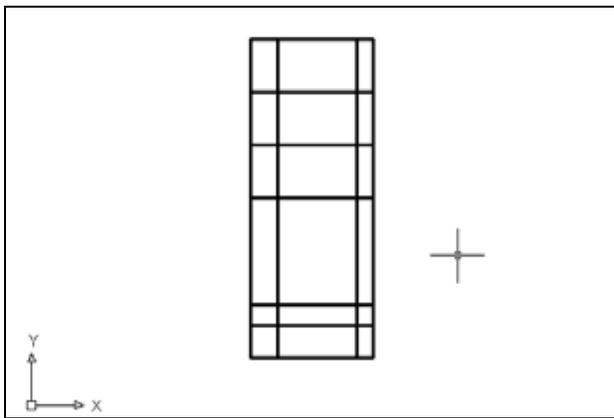


Figure 13.10 – Offset a Line by 23

For the last offset line, create a new vertical line 23 to the right of the vertical mirror line. Hit **Enter** to repeat Offset command. The command line will prompt you for the offset distance, so type **23** and hit **Enter**. Select the vertical mirror line and pick to the right of that line to create a new line and hit **Enter**. The line will appear as shown in Figure 13.10. Next, you will draw two angled lines using points of intersection in the matrix.

Add Two Lines Using the Line Command

You next need to place two lines using the Line tool from the Draw panel. With your mouse, select points using the Intersection Object Snap and pick two places to make the first line as shown in Figure 13.11 and hit **Enter** to end the Line command.



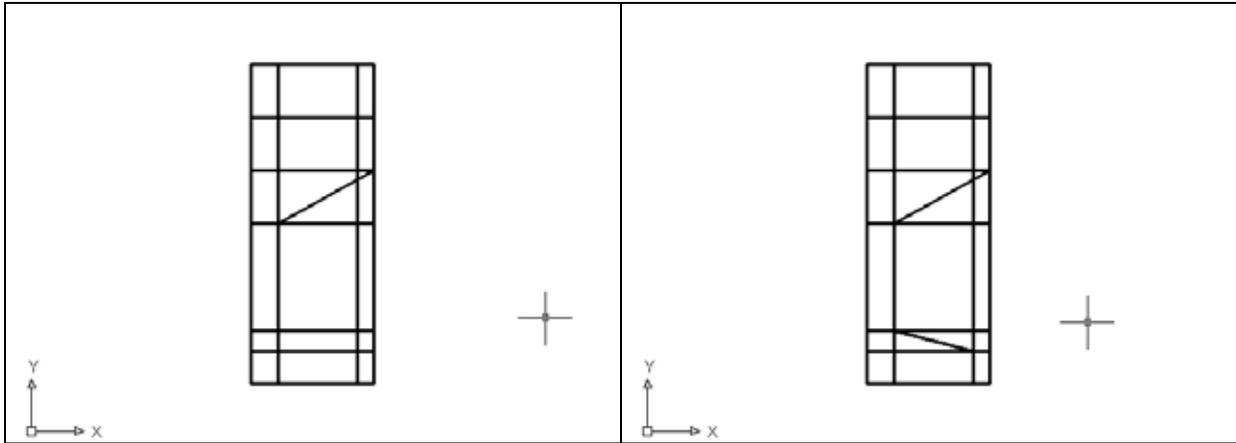


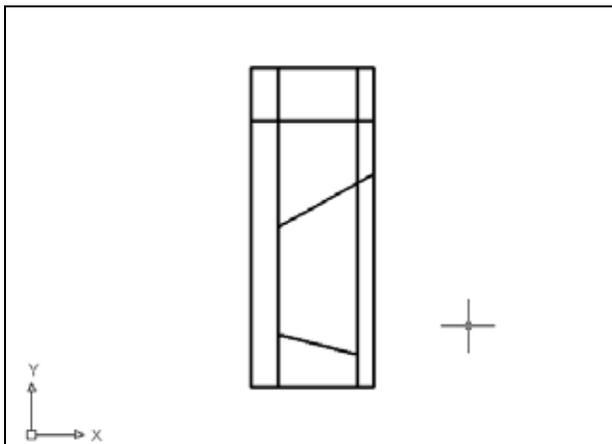
Figure 13.11 – Adding First Angled Line

Figure 13.12 – Add Second Angled Line

Hit **Enter** to repeat the Line command, and with your mouse, select points using the Intersection Object Snap, and pick two places to make the second line as shown in Figure 13.12. Hit **Enter** to end the Line command.

Removing Four Lines Using the Erase Command

Again, do not excessively rely on the erase function since removing entities once drawn is losing efficiency. Now that two lines are present, eliminate four construction lines

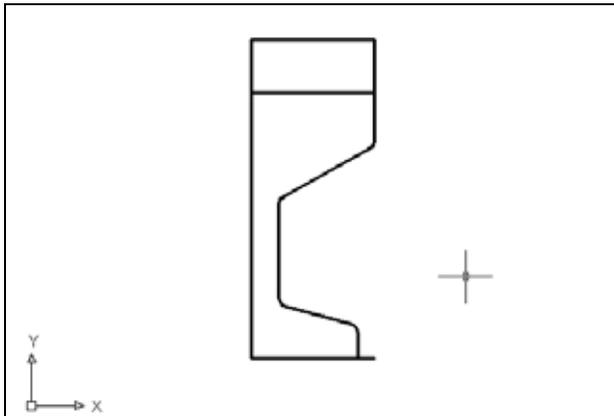


Use the Erase tool on the Modify panel to remove the four horizontal lines that you no longer need. Select the Erase tool on the Modify panel and select the four entities that contain endpoints of the angled lines. Hit **Enter** and this will remove the four lines as shown in Figure 13.13. Now all the remaining lines will change length by using the radius 2 or zero Fillet tool to create intersections in the perimeter to match the sketch at the beginning of the chapter.

Figure 13.13 – Removing Four Lines

Add a Curved Edge Using the Fillet Command

To add a radius edge to a corner, you can use the Fillet command to create the feature rather than drawing a curved segment using the Arc command. In a previous problem, you placed the Fillet on a 3D solid edge, but on this exercise, you will change the lined matrix by placing the arc in the 2D perimeter.



Pick the Fillet tool on the Modify panel and then type **R** and **Enter**. Type **2** and **Enter** to set the system for a 2-radius arc. Select the two lines representing a corner where one of the angled lines meet a vertical line and the 2 arc will appear on the edge of the perimeter as shown in Figure 13.14. There are four arcs in this boundary. Hit **Enter** to repeat the Fillet command each time, and pick twice to define the intersection that will remain.

Figure 13.14 – Adding Four Fillets

Rotating Two Line Using the Rotate Command

The Rotate command allows you to rotate entities around a base or center point. Select the Rotate command on the Modify panel and pick the line just below the horizontal centerline. The selected entity will be dotted as shown in Figure 13.15. To continue with the command, press **Enter** to select the base point of rotation. Select the Endpoint of the line as shown in Figure 13.15. Type **3** and **Enter** for the degrees of clockwise rotation to finish this detail will appear as shown in Figure 13.16.

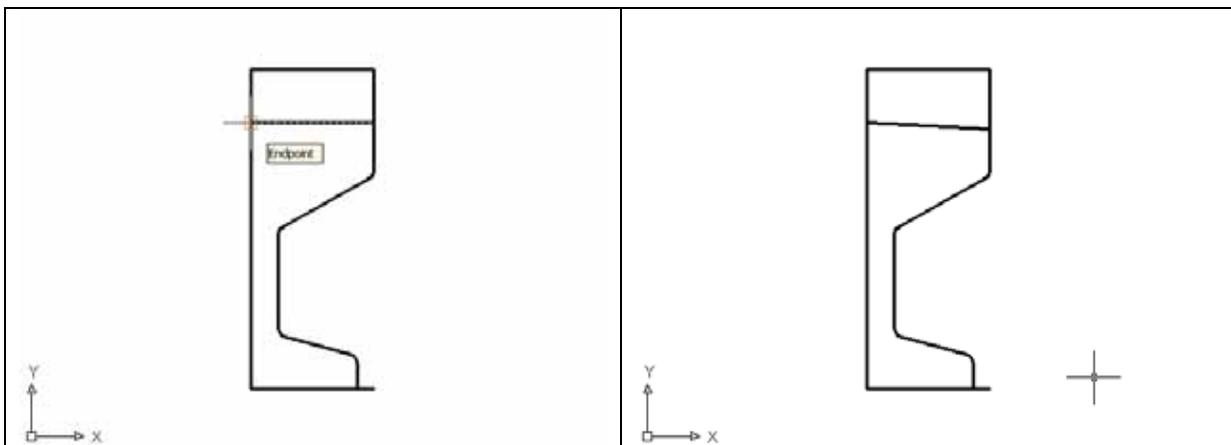


Figure 13.15 – Base Point for Rotation

Figure 13.16 – First Rotated Line

Hit **Enter** to repeat the Rotate command and pick the bottom horizontal line. The selected entities will be dotted as shown in Figure 13.17. To continue with the command, press **Enter** to select the base point of rotation. Select the Endpoint of the line as shown in Figure 13.17. Type **-3** and **Enter** for the degrees of clockwise rotation to finish this detail will appear as shown in Figure 13.18.

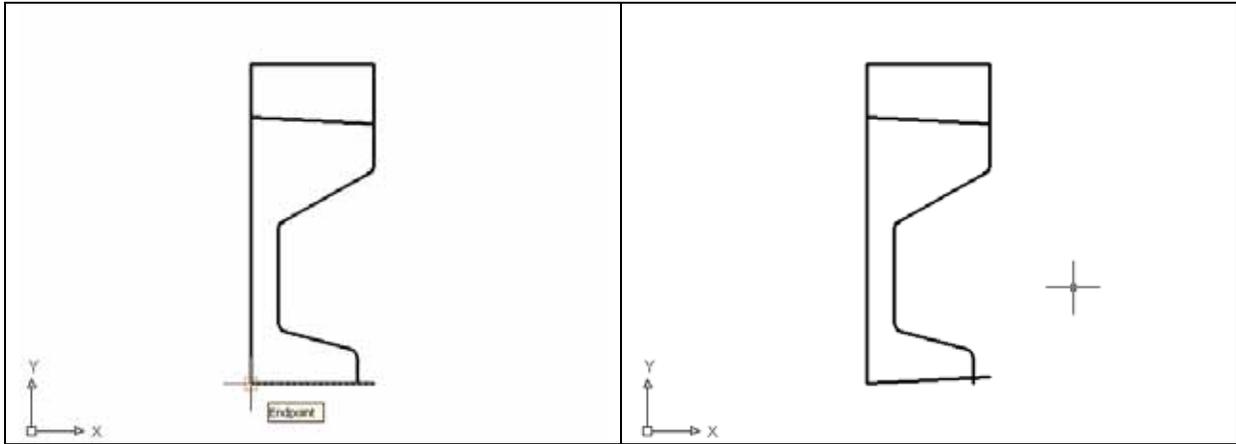
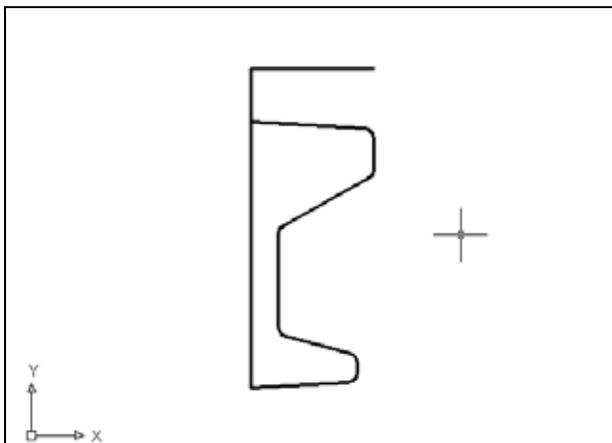


Figure 13.17 – Base Point for Rotation

Figure 13.18 – Second Rotated Line

Add Two Additional Curved Edges Using the Fillet Command

To add two more radius edges to corners, you can use the Fillet command. These two arc segments will be located on the end of the lines just rotated.

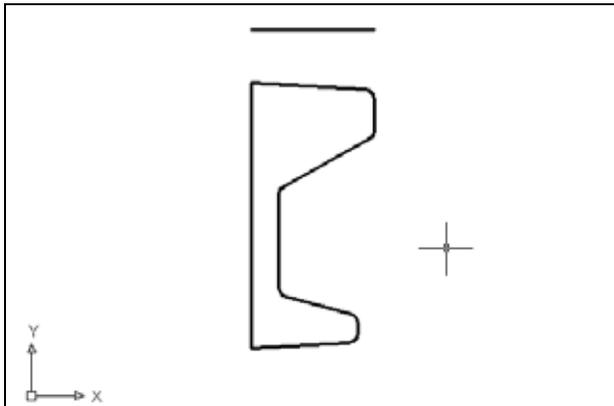


Pick the Fillet tool on the Modify panel. You do not need to set the radius since the radius measurement in the function is still set at 2. Select the two lines representing a corner where one of the rotated lines meet a vertical line and the 2 arc will appear on the edge of the perimeter as shown in Figure 13.19. There are two arcs in this boundary. Hit **Enter** to repeat the Fillet command each time, and pick twice defining the intersection that will remain.

Figure 13.19 – Two More R2 Fillets

Add a Zero Radius Edge Using the Fillet Command

To add a zero radius edge to a corner, you can use the Fillet command to create the feature rather than chopping away line segments using the Trim command. Many computer-aided designers use the zero-radius to create sharp corners between two entities.

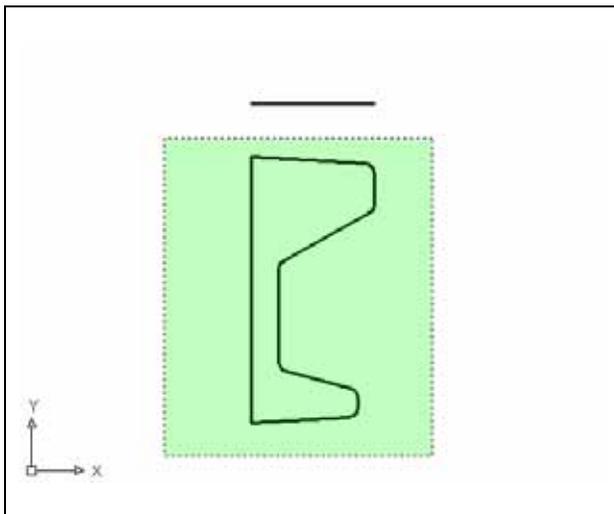


Pick the Fillet tool on the Modify panel and then type **R** and **Enter**. Type **0** and **Enter** to set the system for a zero radius arc. Select the two lines representing a corner where one of the angled lines meet a vertical line and the sharp corner will appear on the edge of the perimeter as shown in Figure 13.20. There are two zero radius corners in this boundary. Hit **Enter** to repeat the Fillet command each time, and pick twice defining the intersection that will remain.

Figure 13.20 – Using Two 0 Fillets

Creating Polylines 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 under the drop-down arrow on the Modify panel and the command line will prompt you to “select objects”.



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 13.21 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.

Figure 13.21 – Making a Complex 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 under the right drop-down arrow of the Modeling panel and the command line 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 13.22. 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 13.23 for the object. Hit **Enter** to select 360 as the default angle of revolution, and the new solid will appear as shown in Figure 13.24.

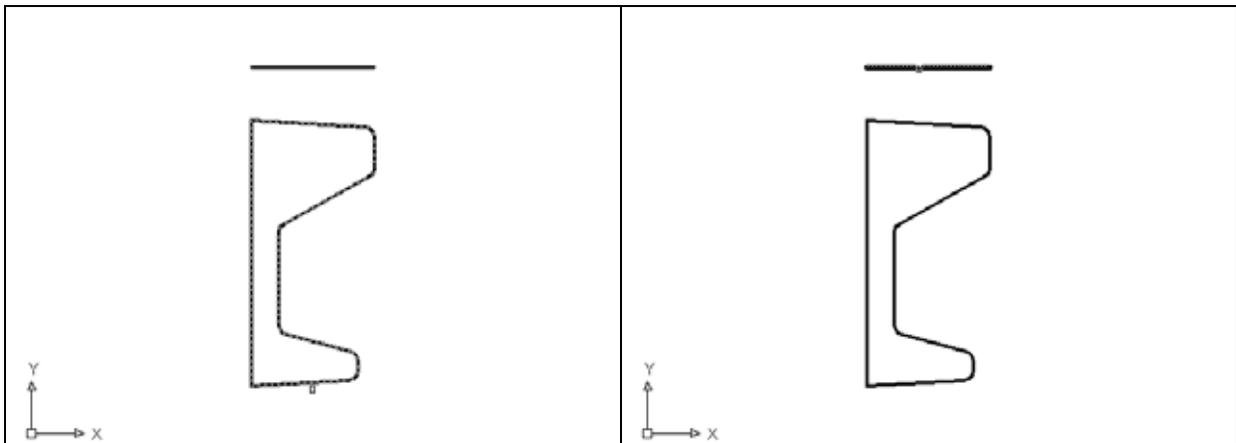


Figure 13.22 – Select the Polyline

Figure 13.23 – Select the Axis Line

After defining the axis of rotation, a new master solid will appear on the graphical display. 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 as shown in Figure 13.25.

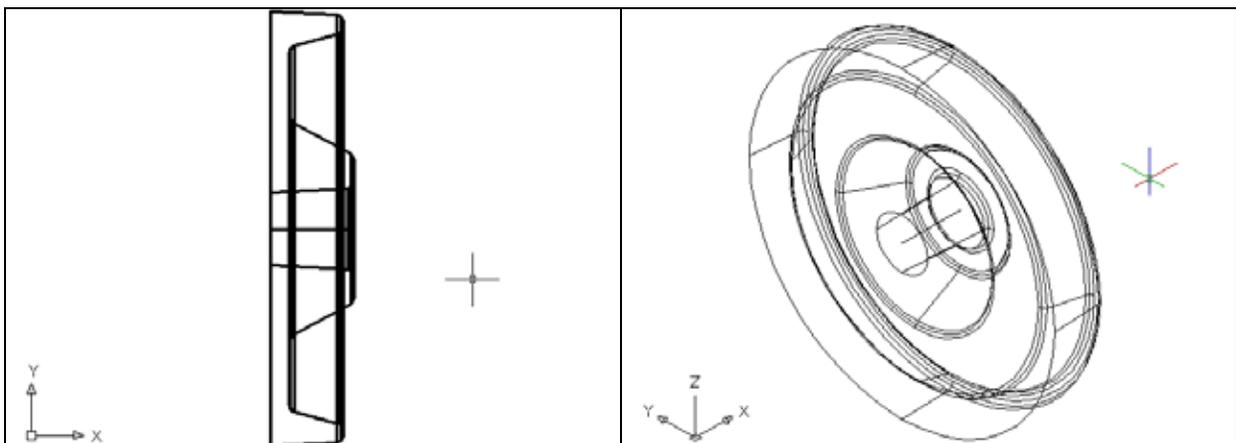
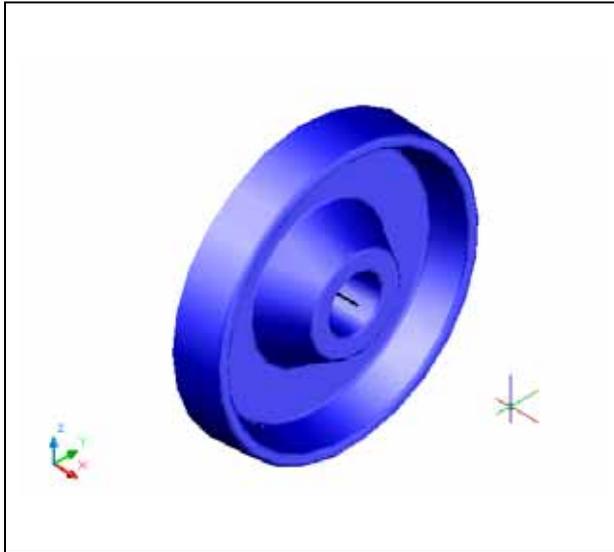


Figure 13.24 – The Revolved Solid

Figure 13.25 – SW Isometric View

Shade the 3D Solid

To view the 3D solid with holes in a better means, select one of the tools on View panel. 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 View panel. 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 View panel 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 View panel.

Figure 13.26 – Shaded Solid

Rotating a 3D Solid Using the 3D Rotate Command

Select the 3D Rotate tool on the Modify panel. At the command, “select objects”, pick the solid on the graphical display and **Enter** to proceed to the second part of the function. Select the bottom front quadrant Osnap as the point of rotation. You are going to rotate the solid part on the y-axis, so pick the green diameter from the globe as shown in Figure 13.27. Type **90** to rotate the solid as shown in Figure 13.28.

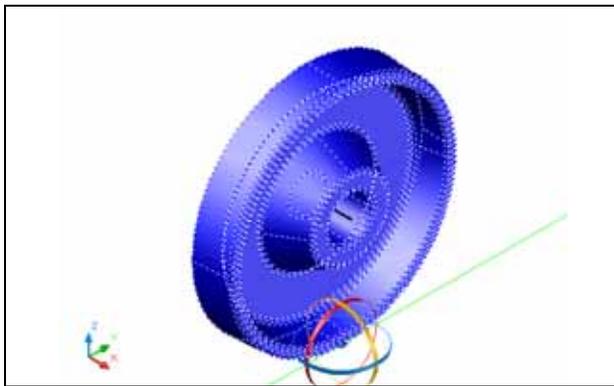


Figure 13.27 – The Rotation Axis

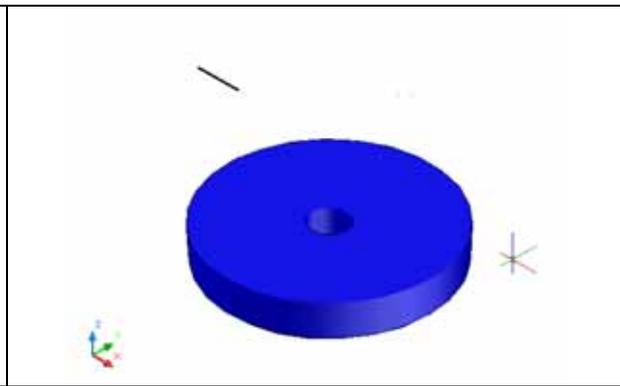
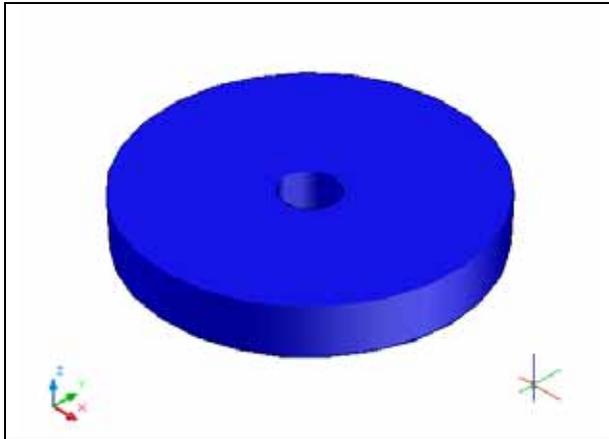


Figure 13.28 – The Rotated the Solid

Removing the Center Line Using the Erase Command

Again, do not excessively rely on the erase function because removing entities once drawn inefficient. 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. When you employ the centerline, you do have to remember to remove the entity with the Erase function.



Use the Erase tool on the Modify panel to remove the centerline that you no longer need. Select the Erase tool on the Modify panel, and select the entity. **Enter** and this will remove the hemisphere on the front side as shown in Figure 13.29.

Figure 13.29 – 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 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”.



Drawing a Circle to be Extruded at a Taper

You need to add six holes to the master solid, one hole going completely through the master solid, and a second through sixth holes that 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.



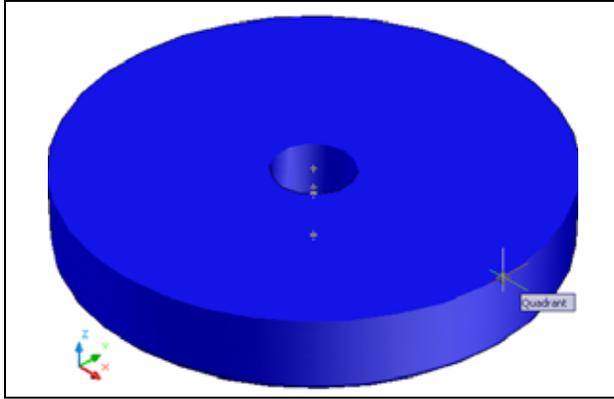


Figure 13.31 – Center of a Circle

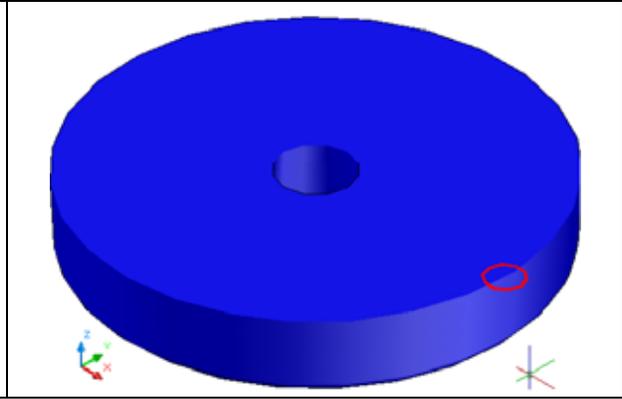


Figure 13.32 – The Circle

On the Draw panel, pick the Center, Radius tool. Specify the center point for the bottom of the cylinder as the Quadrant of the master solid as shown in Figure 13.31. Type “D” for diameter and hit **Enter** to input the sketch’s measurement of 10. Hit **Enter** and the circle will appear in the graphical display as shown in Figure 13.32.

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 Polylines and Circles at a time with the Extrude command. 

Select the Extrude tool on the Modeling panel and the command line will prompt you to “**select objects**”. Pick the circle as shown in Figure 13.33 and you will see the selected entity as a dotted selection. The command line will return with “1 found”, so hit **Enter**. You will be prompted to “Specify height of extrusion or [Direction/Path/Taper angle]”. Type “T” and **enter** to change the angle of taper. Type “3” and **Enter** for the taper angle, and then “-5” and **Enter** for the height. A more complex 3D solid will appear in the graphical display as shown in Figure 13.34.

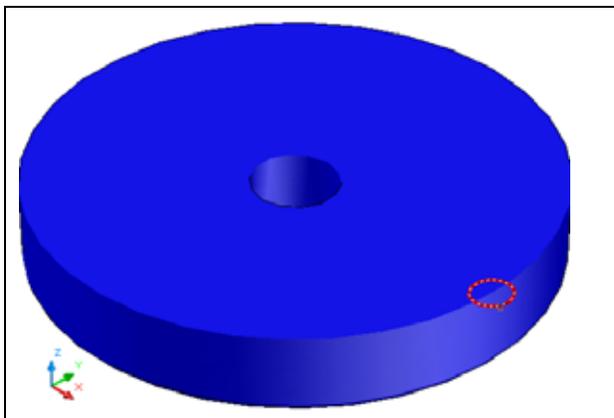


Figure 13.33 – Select for Extrusion

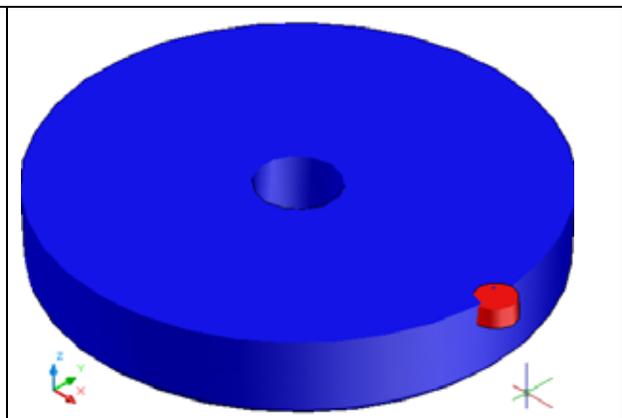


Figure 13.34 – Extrude the Circle

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.

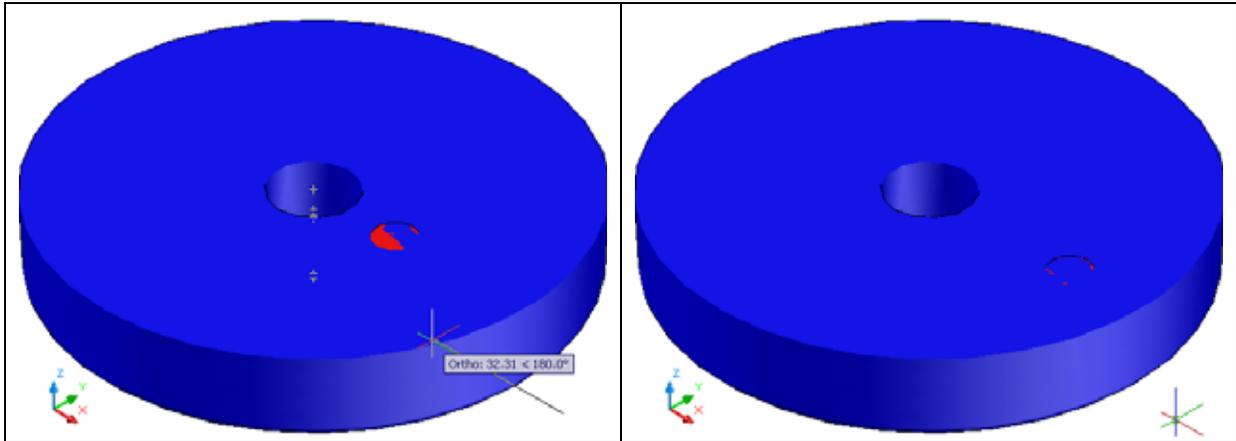


Figure 13.35 – Moving the Solid

Figure 13.36 –The Solid in Position

To move the cylinder, select the Move tool on the Modify panel. 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 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 13.35. For the “second point of displacement”, you need to move the cursor to the left and with the Ortho “On”, type **20.0** and **Enter**. The cylinder will be ready to polar array as shown in Figure 13.36.

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.



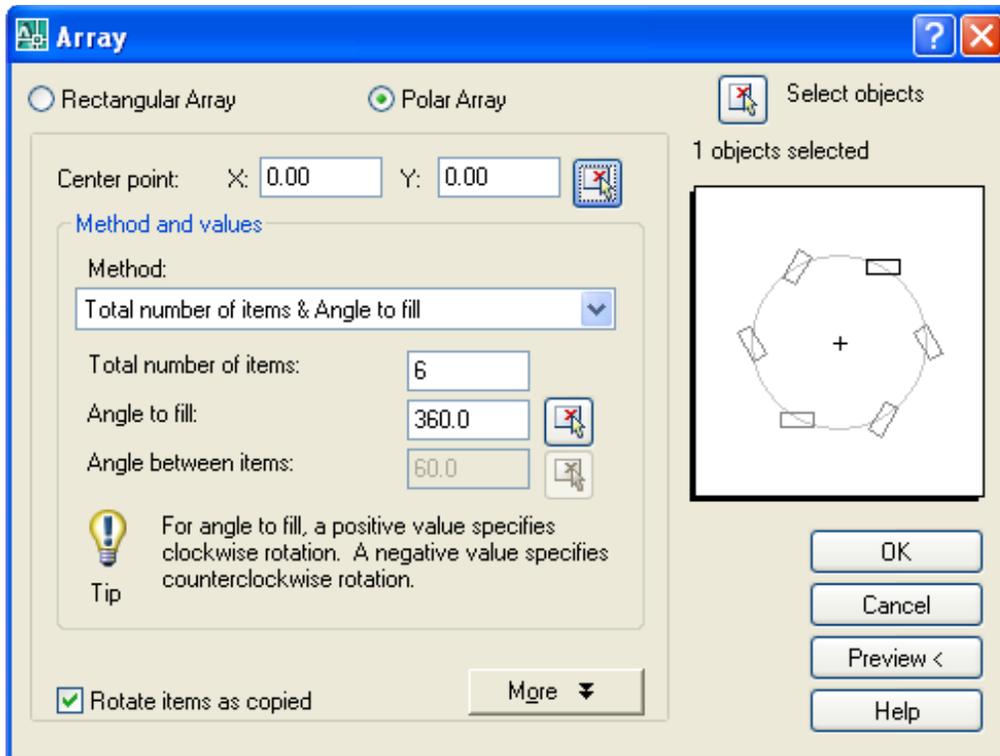


Figure 13.37 – The Array Window

Select the Array tool on the Modify panel under the drop-down arrow, which will cause the Array window as shown in Figure 13.37 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 13.38 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 13.39.

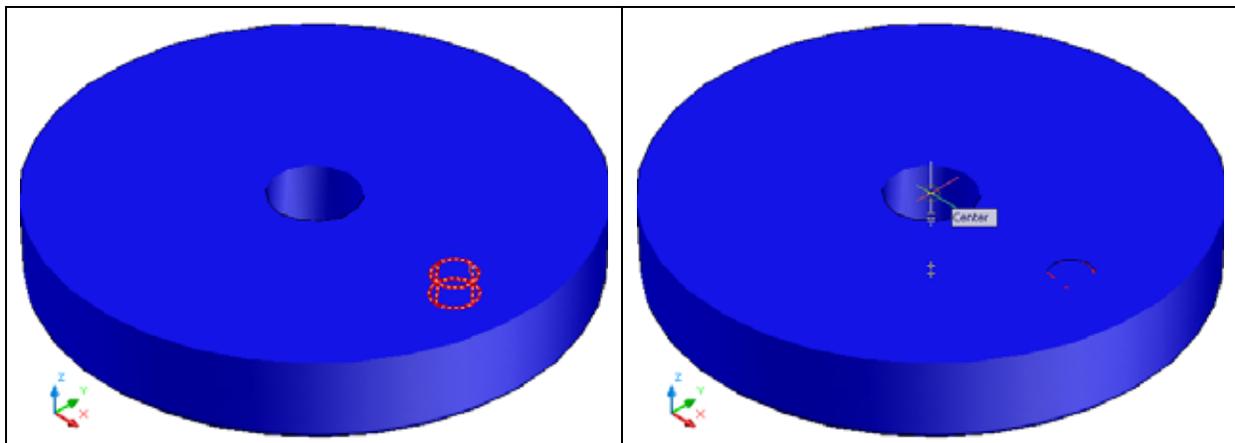
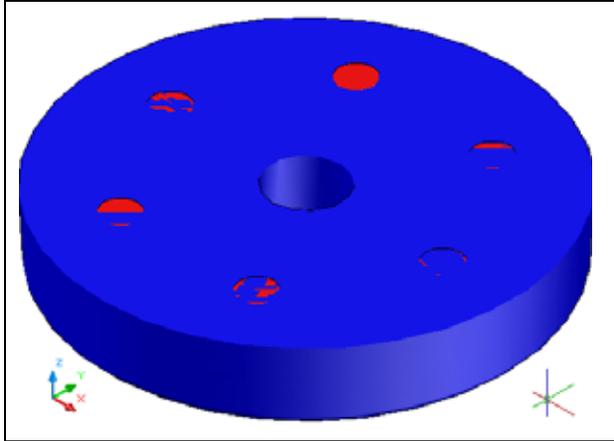


Figure 13.38 – Select Cylinder

Figure 13.39 – Center of the Polar Array



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 “6” 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 13.40, press the Accept button and the array will be completed. If the Circular problem does not look like Figure 13.40, press the Modify button repeat the array process.

Figure 13.40 – The Arrayed Cylinders

Subtracting 3D Solids from the Master Solids

To remove the cylinders from the master solid, select the Subtract tool on the Solid Editing panel.

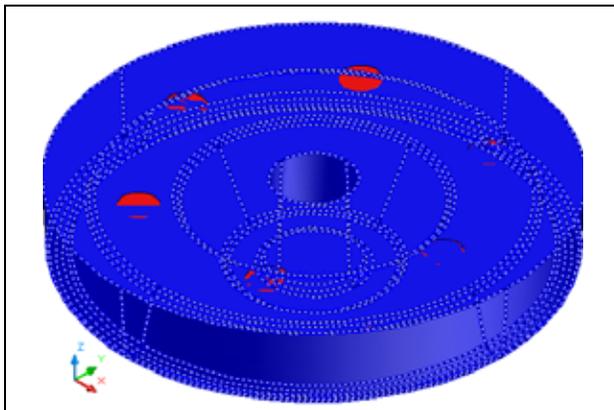


Figure 13.41 – The Master Solid

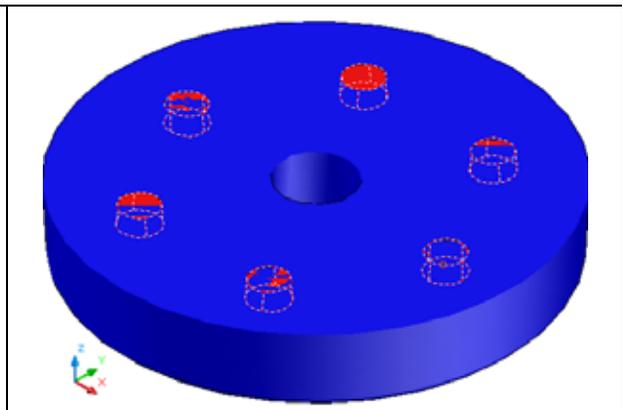


Figure 13.42 – Cylinders to be Subtracted

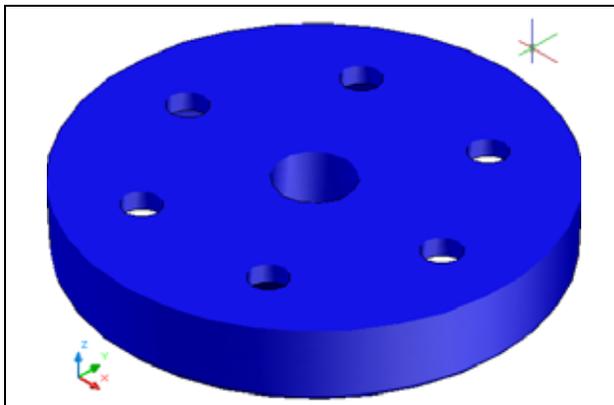
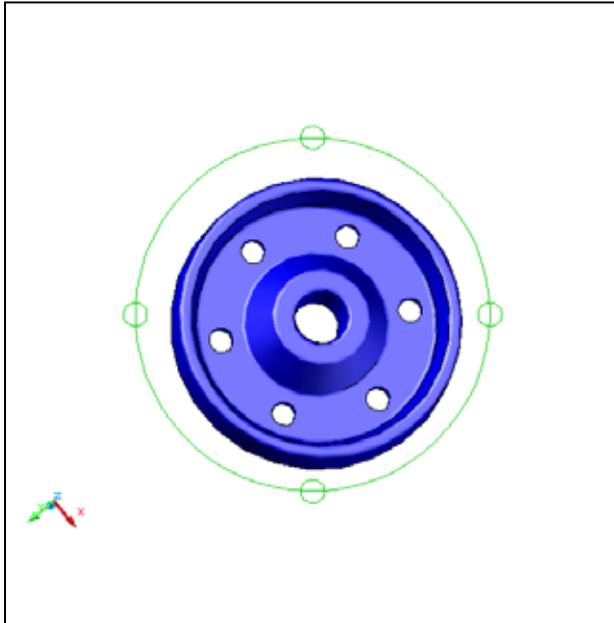


Figure 13.43 – Master Solid with Holes

AutoCAD will prompt you with “Select solids and regions to subtract from”. Select the master solid as shown in Figure 13.41 and the system will respond with “1 found”. You need to hit **Enter** to proceed to the next step, which is to “Select solids and regions to subtract”. Pick the six cylinders as shown in Figure 13.42 and the software application will counter with “6 found”. Hit **Enter** and a region exactly the shape of the six cylinders will disappear from the larger one.

Using the 3D Orbit Tool to Change to a Non-standard View

Whenever you wish to view an entity from a perspective other than the front, top, right or any other primary orthographic direction, the 3D Orbit function will be the tool you may choose to quickly acquire the correct observation.



Select the Free Orbit tool under the drop-down arrow next to the Orbit tool from the Navigation panel on the View tab. A green oval appears on top of the graphical display. A small sphere with two circular lines around the ball appears in the middle of the green orbit ring. Press down the left mouse button and move the master solid in any direction. By placing the cursor in the small circle at the quadrant, you can select a vertical or horizontal orbit that has the ball with the circular line going in a vertical or horizontal path. Placing the cursor outside the ring and selecting with the mouse will allow for a rotation matching the path of the orbit ring. Move the solid into a position as shown in Figure 13.44.

Figure 13.44 – 3D Orbit into View

Add a Curved Edges Using the Fillet Command

To add a radius edge to the corner of the six holes, you can use the Fillet command to create the feature

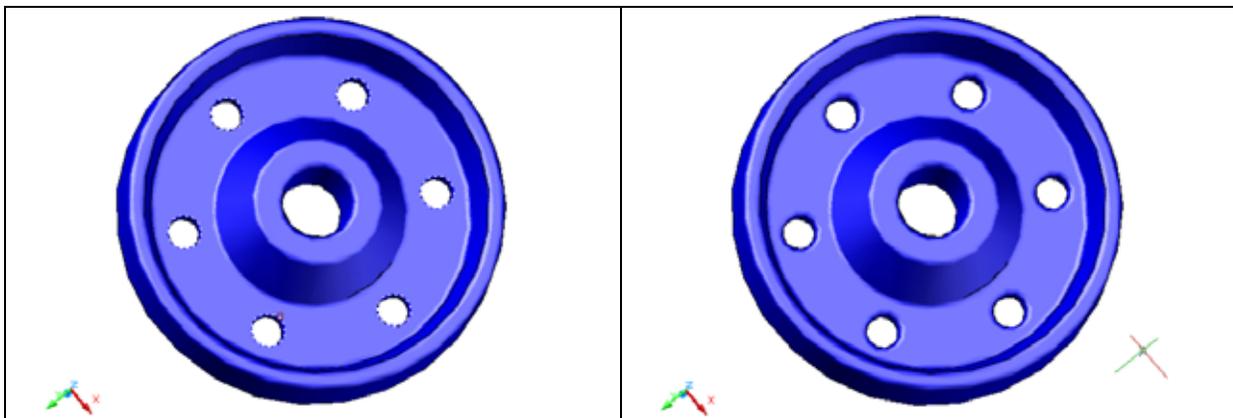


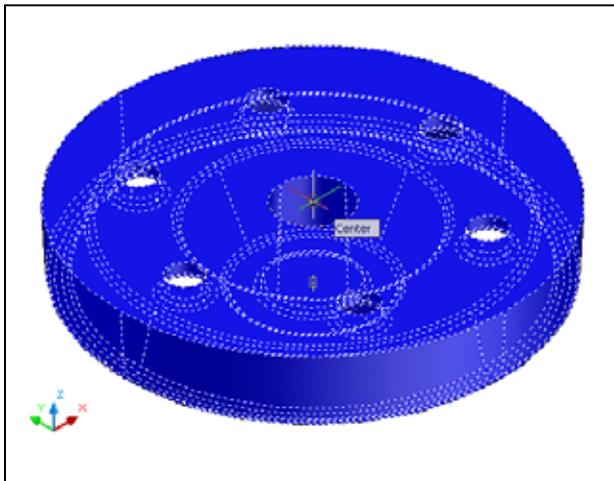
Figure 13.45 – Pick the Top of the Holes

Figure 13.46 – Fillets on the Holes

Select the Fillet tool on the Modify panel and select the top tessellation line on the edge of a hole. The fillet radius is still set at 2 so you will not need to change the radius. Select the top tessellation lines on all six holes as shown in Figure 13.45. Hit **Enter** and the curved radius will appear on the edge of the each hole as shown in Figure 13.33.

Moving the Solid to the Origin Point

To move revolving problem two to the drawing origin of 0,0,0 on the x, y and z-axis, select the SW Isometric viewpoint, and then pick the Move tool on the Modify panel, select the 3D solid, and hit Enter.



At the prompt, “**specify base point or displacement**”, select the endpoint shown in Figure 13.47 at the top center of the revolving problem two, which will become problem two’s insertion point into an assembly drawing. At the “second point of displacement”, type **0,0,0** and **Enter**. Problem two 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 13.47 – Moving to the Origin

Adding the Other Side Using 3D Mirror Command

Select the 3D Mirror command on the Modify panel. At the command, “select objects”, pick the solid on the graphical display as shown in Figure 13.48 and push **Enter** to proceed to the second part of the function.

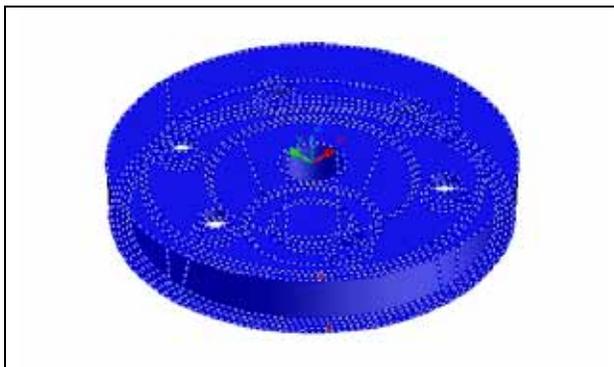


Figure 13.48 – Select the Master Solid

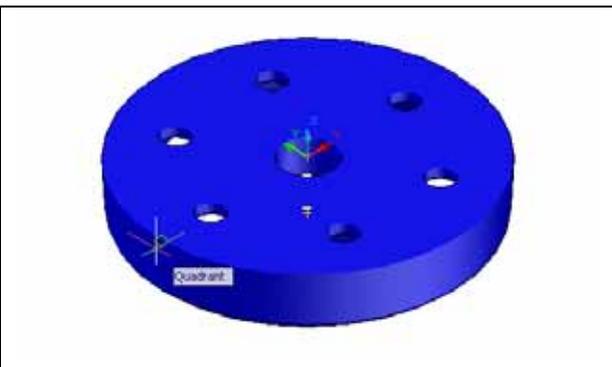


Figure 13.49 – Identify the Mirror Plane

Type **XY** signifying you are going to mirror the solid part on the XY-plane. Pick the Quadrant point on the display as shown in Figure 13.49. Press **Enter** once more to accept the default, not to delete the old side.

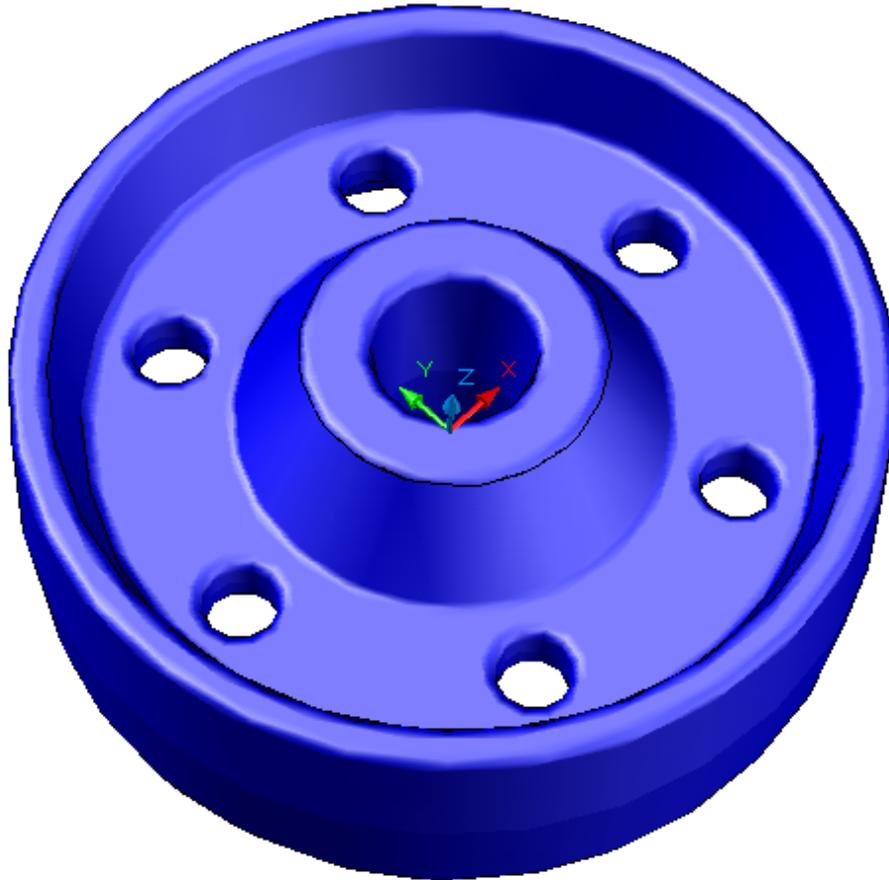


Figure 13.50 – The Finished Solid

To finish the second revolve problem, union the two halves together.

Saving the Solid Problem

To save Revolving Problem 2 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 “Revolving Problem 2” and press the Save button to save the drawing. (See Figure 13.37)



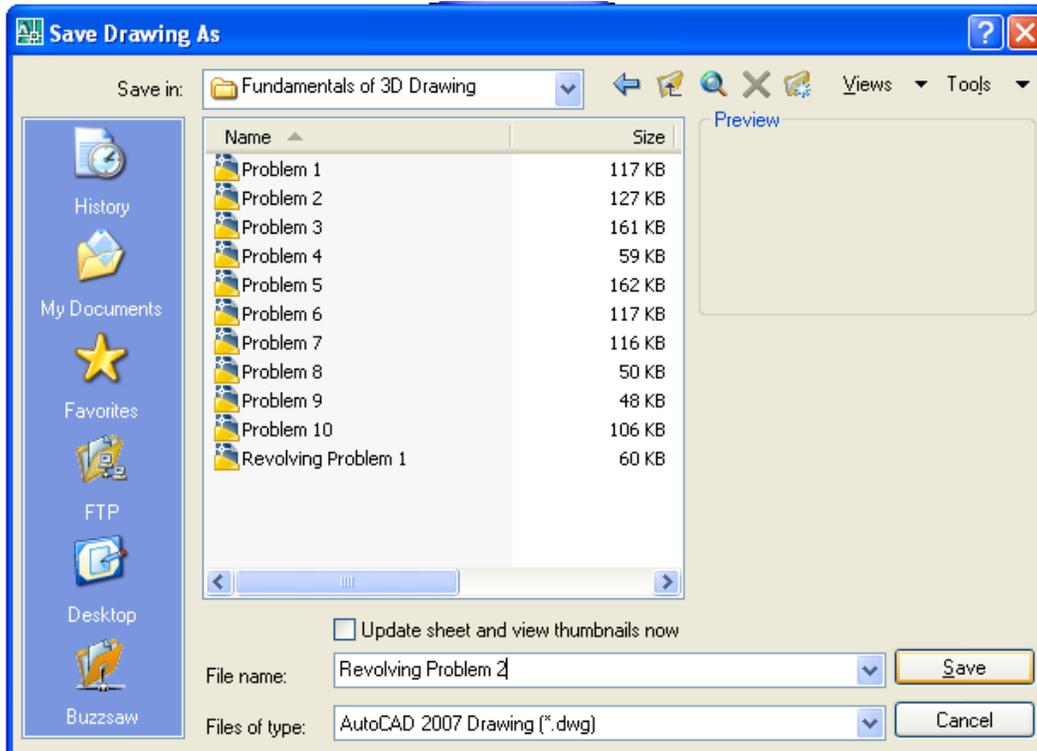


Figure 13.51 – Saving Revolving Problem Two

*** World Class CAD Challenge 03-12 * - 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.**

