

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 UCS settings
- Creating center and mirror lines using the Line command
- Using the Offset command to create new lines
- Adding two lines using the Line Command
- Removing lines using the Erase command
- Adding curved edges using the Fillet command
- Rotating lines using the Rotate command
- Adding zero radius edges using the Fillet command
- Creating a 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 a circle to be extruded with a taper
- Extruding a solid from closed polylines and circles
- Moving a solid into position on the master solid
- Creating a solid array with the Array command
- Subtracting 3D solids from the master solid
- Moving the solid to the origin point
- Mirroring the solid using the 3D Mirror 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 and X axes. In this exercise you will use the Fillet command to place rounded edges on the polyline perimeter. After revolving the closed polyline perimeter into a solid, you will place a tapered and extruded solid onto the master solid, create an array from the detail, and subtract the additional solids to make holes in the master solid. You will utilize the Fillet command once more and put a rounded edge on the six mounting holes. Finally, you will use a new command, 3D Mirror, to copy the master solid across the plane and union them together to finish the challenge.

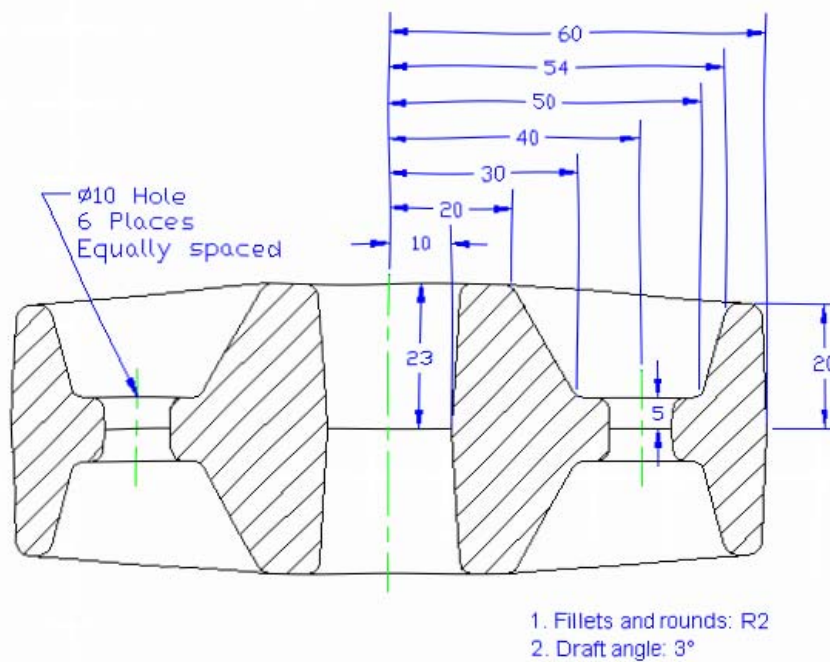


Figure 13.1 – Revolving Problem Two sketch

Remember, when you receive a sketch from a professional, 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 Two 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.

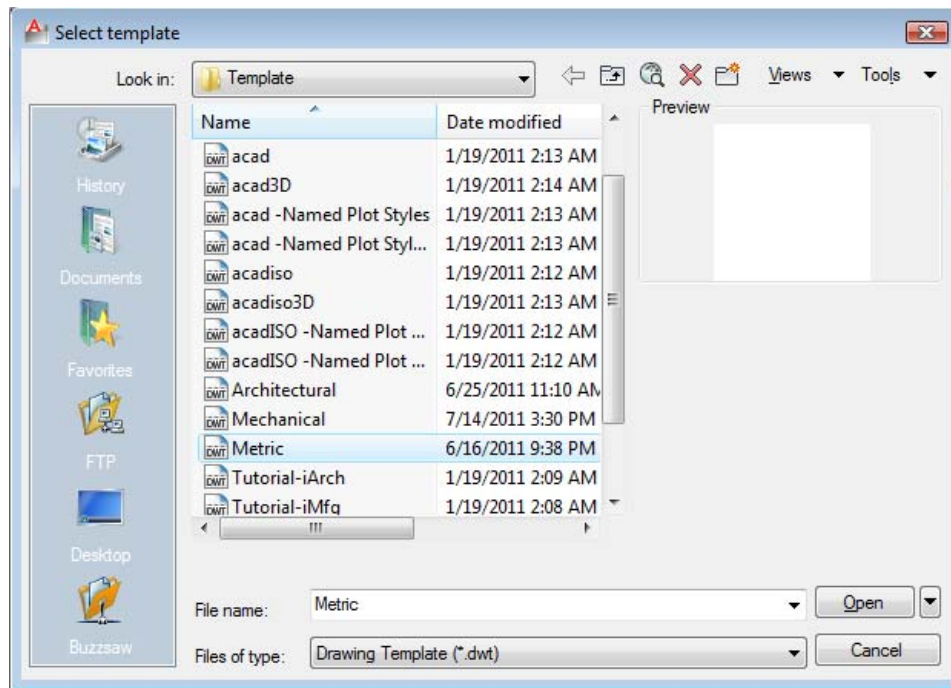


Figure 13.2 – Starting the drawing using the metric template

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 13.3) and select the “Show UCS Icon” option (Figure 13.4). You also have the choice of keeping the UCS displayed at the origin or turning it off completely.

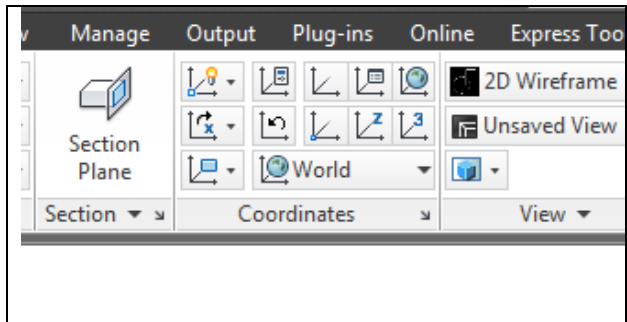


Figure 13.3 - The Coordinates panel

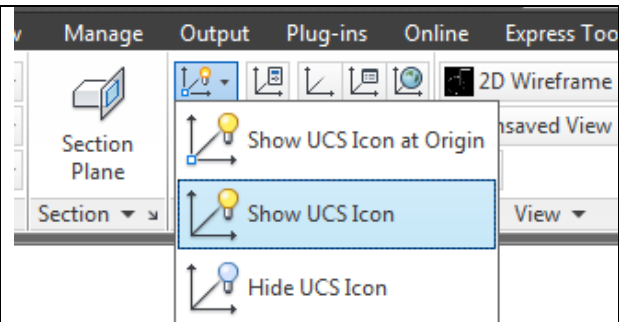
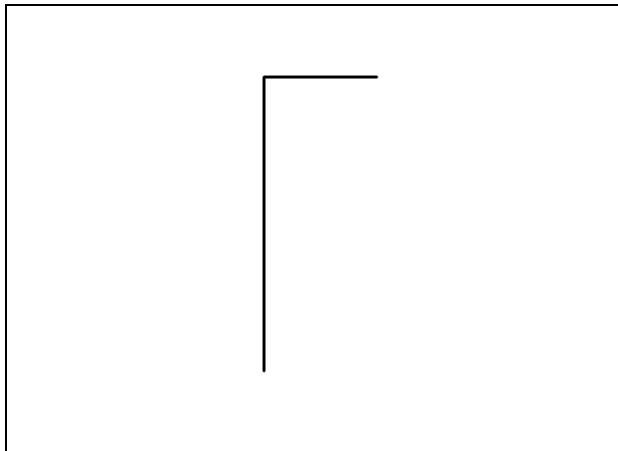


Figure 13.4 - Changing the UCS display settings

Creating 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.



Select any point at the upper right side of the graphical display. With the Ortho mode “on”, move the line on the graphic screen to the left and type **23** and **ENTER**. Then pull the cursor downward and type **60** and **ENTER**. Two lines will appear on your graphical display as shown in Figure 13.5. Hit **ENTER** again to exit the line command.

Figure 13.5 – The center and mirror lines

Using the Offset Command to Create New Lines

You will use the offset command to add a set of construction lines from which you will create the outline of the closed perimeter. This is not the first time you have used this technique; you offset construction lines in World Class CAD’s Fundamentals of 2D Drawing training module.



You will use the Offset command to create a new line 10 units below the horizontal centerline. This will allow you to create a line without selecting points to define its start and end points. Choose the Offset tool on the Modify panel. After picking the Offset tool, the

command line will prompt you for the offset distance, so type **10** and **ENTER**. Now select the horizontal centerline and pick a point below that line to create a new line 10 below the first. Select the new line and pick below that line again to create another. Do this process one more time to generate a third line below the centerline, hitting **ENTER** once more to exit the Offset command.

To create a new line 20 units below the last line you created, hit **ENTER** again to repeat Offset command. The command line will prompt you for the offset distance so type **20** and **ENTER**. Now select the last offset line and pick below that line to create a new line 20 below the one selected. Hit **ENTER** twice to exit and repeat the Offset command. Offset the line you just created once more, 10 units below. All offset lines should appear in the drawing as shown in Figure 13.6.

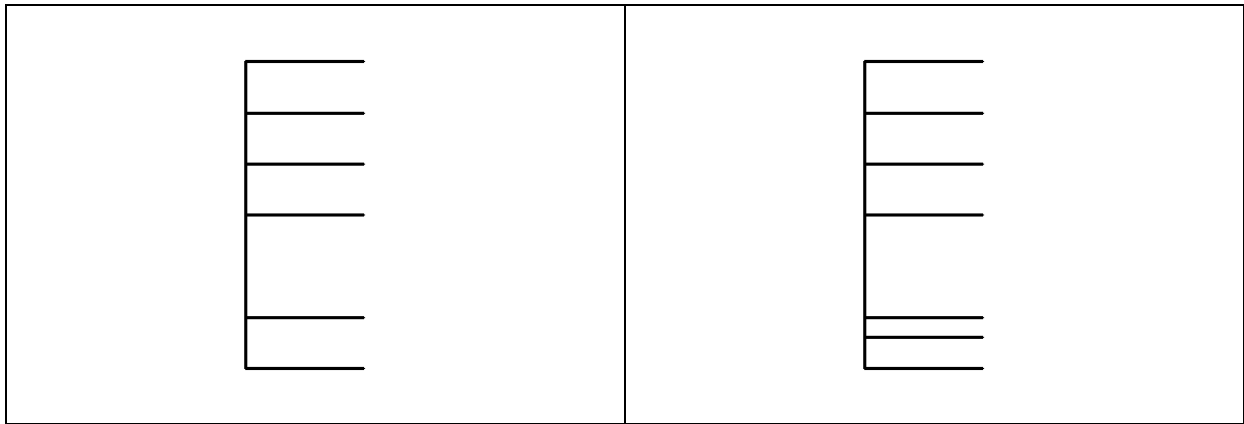


Figure 13.6 – Offset lines by 10

Figure 13.7 – Offset a line by 6

You need to create one additional horizontal line using the Offset command 6 units above the last offset line. 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 units above the one selected and hit **ENTER**. The line will appear as shown in Figure 13.7.

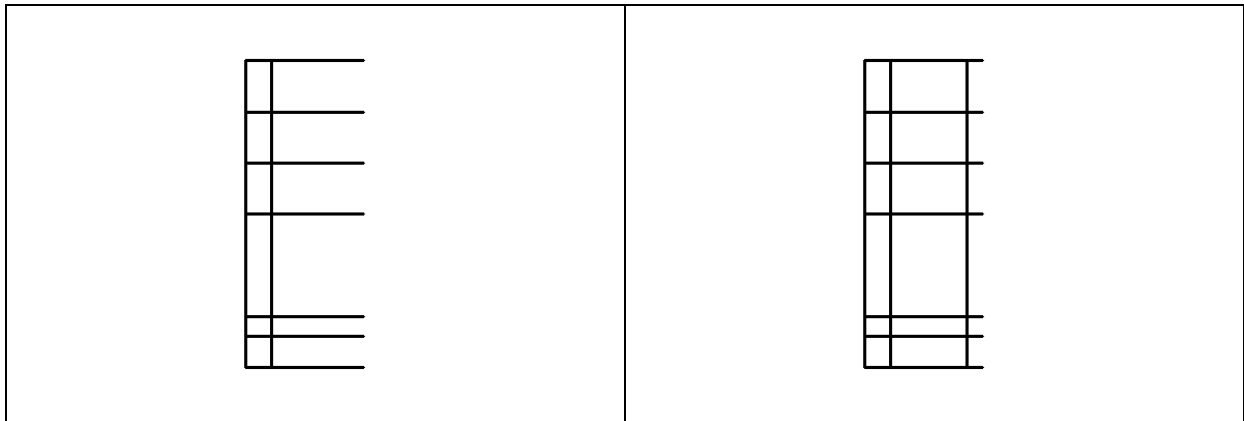
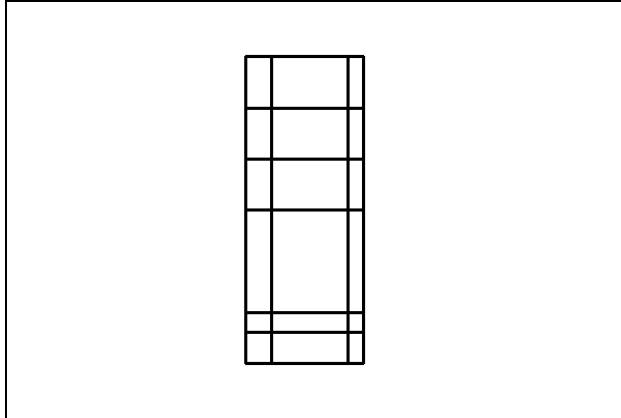


Figure 13.8 – Offset a line by 5

Figure 13.9 – Offset a line by 20



Repeat the Offset command again and offset the vertical line 5 units to the right (Figure 13.8). Offset the same original vertical line to the right again by a distance of 20 (Figure 13.9), and then once more by a distance of 23 (Figure 13.10). Next, you will draw two angled lines using intersection points in the matrix you have created as reference points.

Figure 13.10 – Offset a line by 23

Adding Two Lines Using the Line Command

You next need to place two lines using the Line tool from the Draw panel. Select two points using the intersection Osnap to make the first line as shown in Figure 13.11.

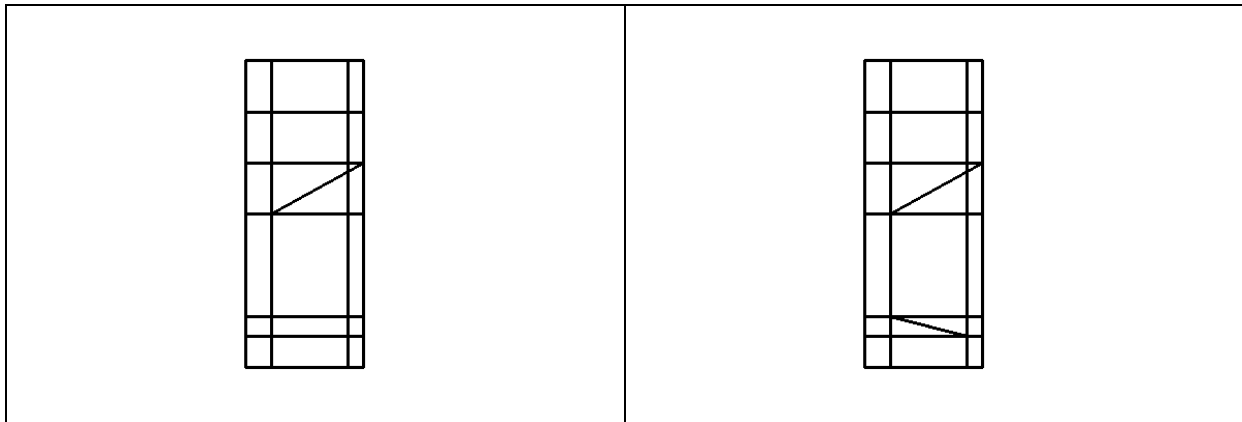


Figure 13.11 – Adding first angled line

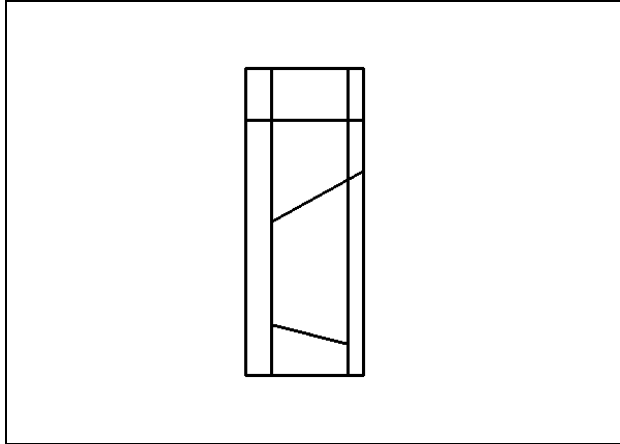
Figure 13.12 – Adding second angled line

Hit ENTER twice to repeat the Line command. Select the two intersections Osnap points so that you place a line as shown in Figure 13.12. Hit ENTER once more to end the Line command.

Removing Four Lines Using the Erase Command

Again, do not excessively rely on the erase function because removing drawn entities is not inherently efficient.



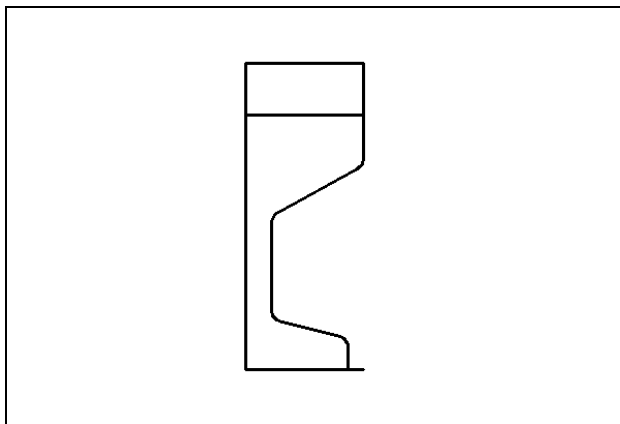


Now that the two angled lines are present, you can eliminate the four construction lines used to create the intersection Osnap points that aided you in placing the two lines. Use the Erase tool on the Modify panel to remove the four horizontal lines that you no longer need. Select the Erase tool on the Modify panel and select the four entities that contain endpoints of the angled lines. Hit **ENTER** to remove the four lines as shown in Figure 13.13. You will use the Fillet tool to join and trim the remaining lines into the perimeter.

Figure 13.13 – Removing four lines

Adding Curved Edges Using the Fillet Command

Now you will use the Fillet command to join and trim lines with a rounded edge as shown in Revolving Problem Two's sketch.



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 unit radius arc. Also, type **M** and **ENTER** so that you will be able to place multiple fillets with this radius measurement. Select an angled line and its intersecting line to place a 2 unit radius on the corner between the two lines as shown in Figure 13.14. There are four rounded edges to place on this perimeter.

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 middle horizontal line. The selected entity will be dotted as shown in Figure 13.15.



To continue with the command, hit **ENTER** to select the base point of rotation. Select the left endpoint Osnap of the line as shown in Figure 13.15. Type **-3** and **ENTER** for the degrees of clockwise rotation as shown in Figure 13.16.

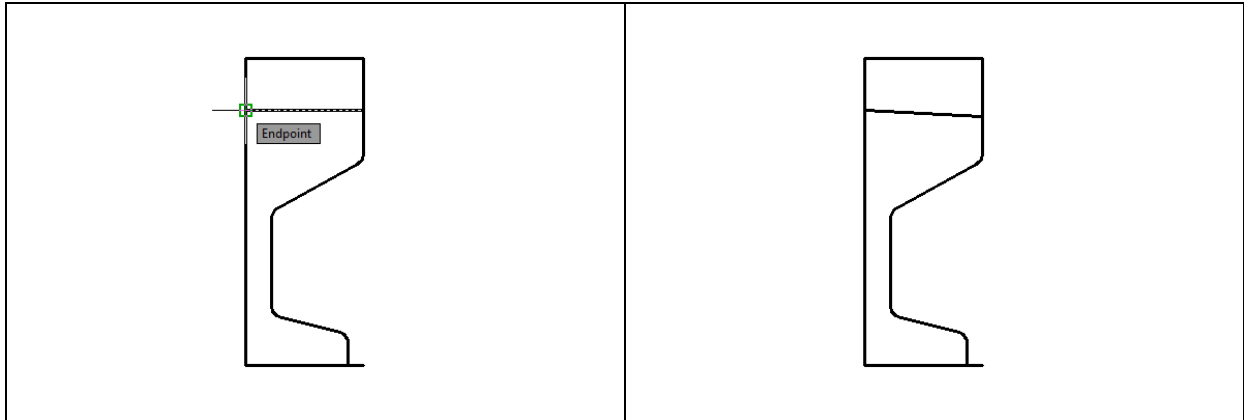


Figure 13.15 – Base point for rotation

Figure 13.16 – First rotated line

Hit **ENTER** to repeat the Rotate command and select the bottom horizontal line. The selected entity will be dotted as shown in Figure 13.17. Hit **ENTER** to select the base point of rotation. Select the left endpoint Osnap of the line as shown in Figure 13.17. Type **3** and **ENTER** for the degrees of clockwise rotation as shown in Figure 13.18.

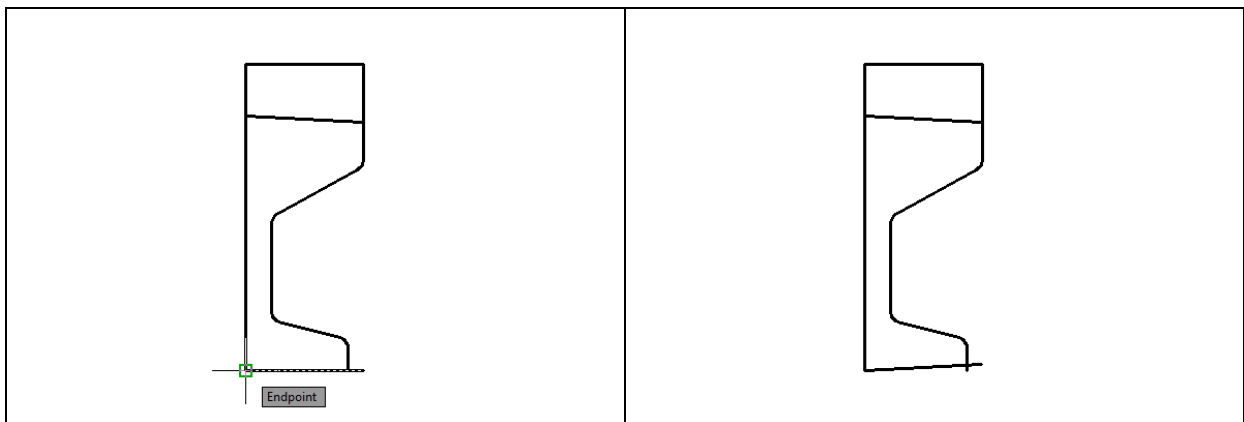


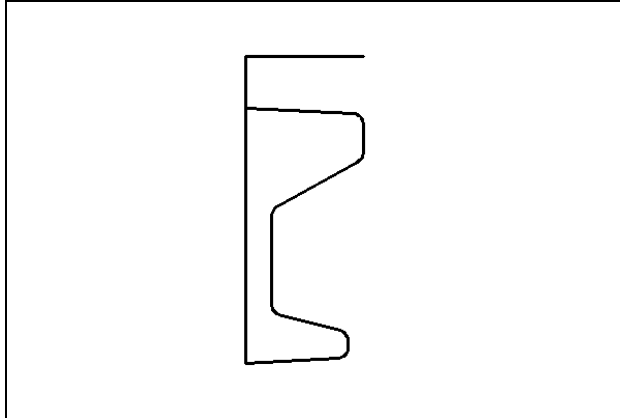
Figure 13.17 – Base point for rotation

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 that you just rotated.



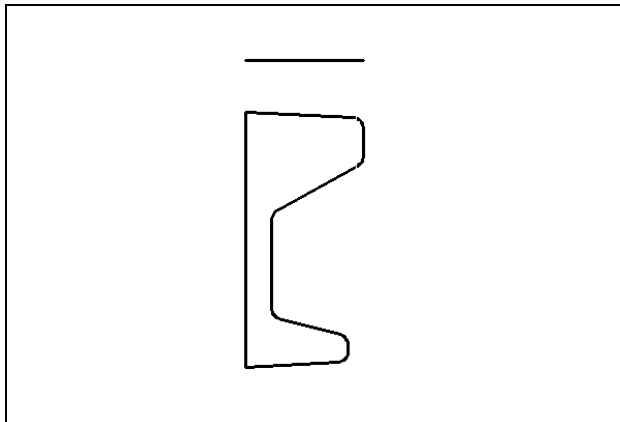


Pick the Fillet tool on the Modify panel. You do not need to set the radius because the radius measurement in the function is still set at 2, but do type **M** and **ENTER** so that you can place both fillets. Select the rotated line and its intersecting line to place a 2 unit radius on the corner between the two lines. The two rounded edges should appear as shown in Figure 13.19.

Figure 13.19 – Two more rounded edges

Add a Zero Radius Edge Using the Fillet Command

You can use the Fillet command to create non-rounded corners between two entities as well. Many computer-aided designers use the zero-radius to create sharp corners between two entities rather than using the Trim and/or Extend commands.



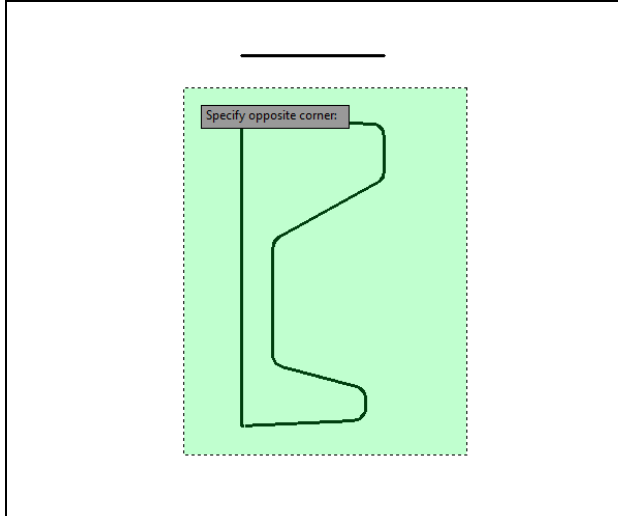
Pick the Fillet tool on the Modify panel and then type **R** and **ENTER**. Type **0** and **ENTER** to set the system for a zero radius arc. Select both lines that create the top left corner of the perimeter and the extended vertical line will be trimmed away as shown in Figure 13.20.

Figure 13.20 – Using a 0 radius fillet

Creating Polylines Using the Edit Polyline Tool

Use the Edit Polyline command to merge the lines and arc segments 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 13.21). The command line should read “**14 total**”. Hit **ENTER** to join all of the entities into a closed shape, and then hit **ESCAPE** to exit the Edit Polyline tool.

Figure 13.21 – 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 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. Hit **ENTER** to select 360 as the default angle of revolution, and the new solid will appear as shown in Figure 13.24.

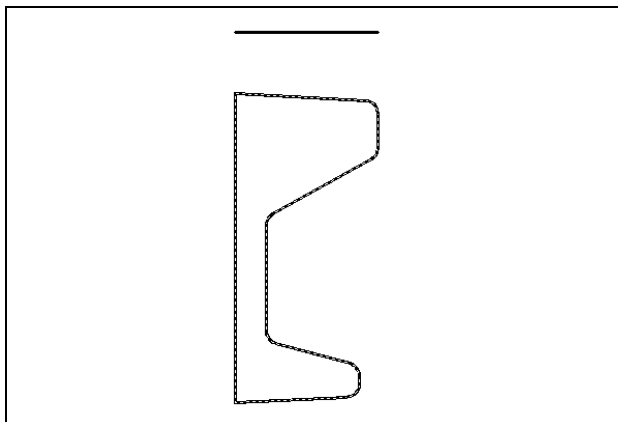


Figure 13.22 – Select the polyline

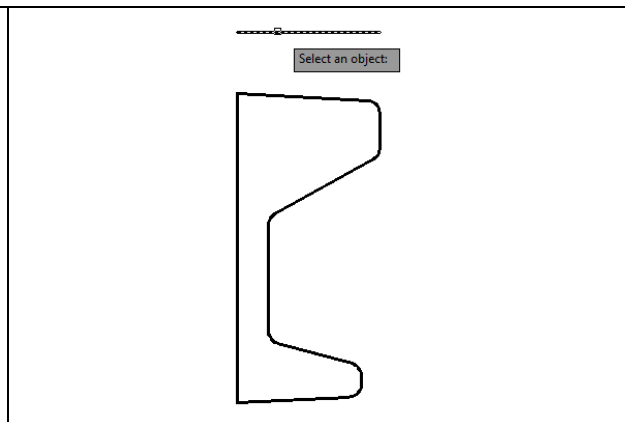


Figure 13.23 – Select the axis Line

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 all three axes as shown in Figure 13.25.



Figure 13.24 – The revolved solid

Figure 13.25 – SW isometric view

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 13.26. The tessellation lines associated with the Hide command are no longer interfering with your ability to read the drawing.

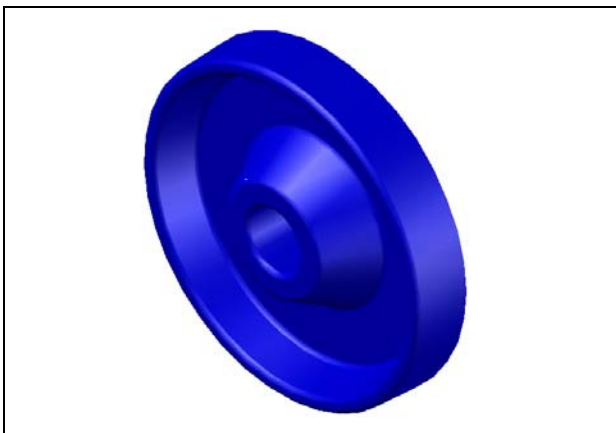
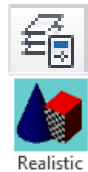


Figure 13.26 – Shaded solid

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 13.26 shows the part in the Northeast isometric view. Change to this view so that you can rotate the master solid.

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**.

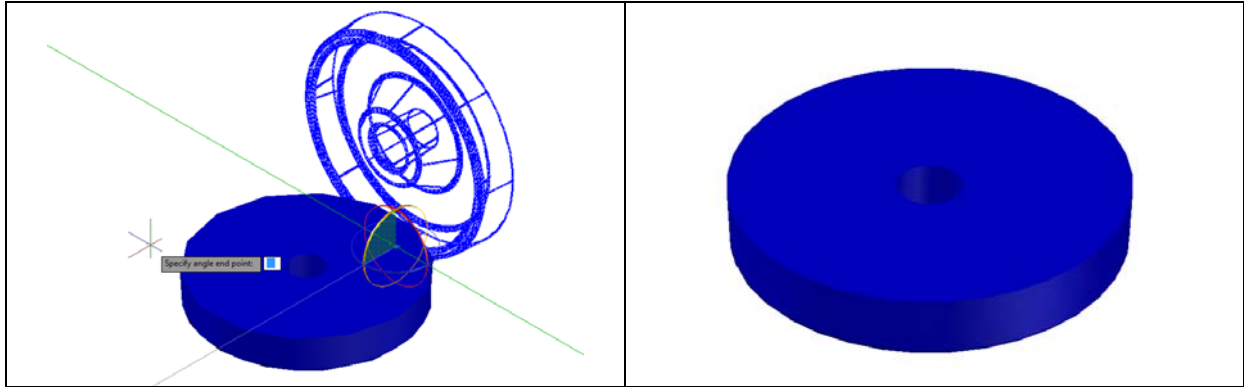


Figure 13.27 – The angle end point

Figure 13.28 – 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 13.27). With the Ortho mode “on”, specify an angle start point and then use the cursor to move the solid 90 degrees clockwise (Figure 13.27). Finish the rotation by clicking with the left mouse button and the solid will appear rotated on its base as shown in Figure 13.28.

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 a Circle to be Extruded at a Taper

You need to add six holes to the master solid. You will create one by extruding a circle, and then create the other five with a polar array. These openings will not be straight; rather they will have a taper, which is a common aspect to cast parts.



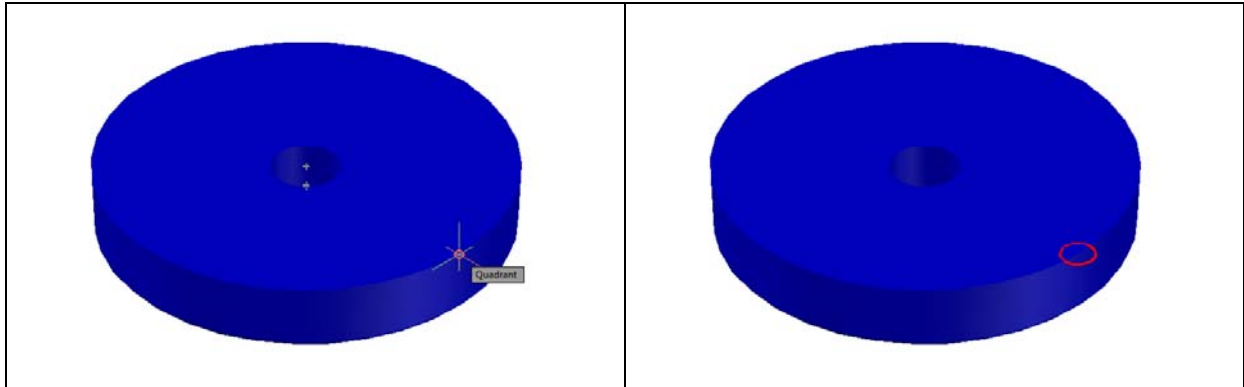



Figure 13.29 – Quadrant Osnap

Figure 13.30 – The circle

Pick the Circle command on the Draw panel. Specify the center point for the bottom of the cylinder as the quadrant Osnap of the master solid as shown in Figure 13.29. 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 13.30.

Extruding a Solid from a Circle

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”. Select the circle and the command line will return with “1 found”, so hit **ENTER**. Before specifying the height of the extrusion, you need to add the angle of taper by typing “T” and **ENTER** and then 3 and **ENTER**. Now when you move your cursor down into the master solid to show the direction of extrusion (with Dynamic Input toggled “on”) you will be able to see the taper, especially if you pull the extrusion to the bottom of the screen. Type “5” and **ENTER** for the height of the extrusion (Figure 13.31). The circle will expand 5 units into the Z axis as shown in Figure 13.32.

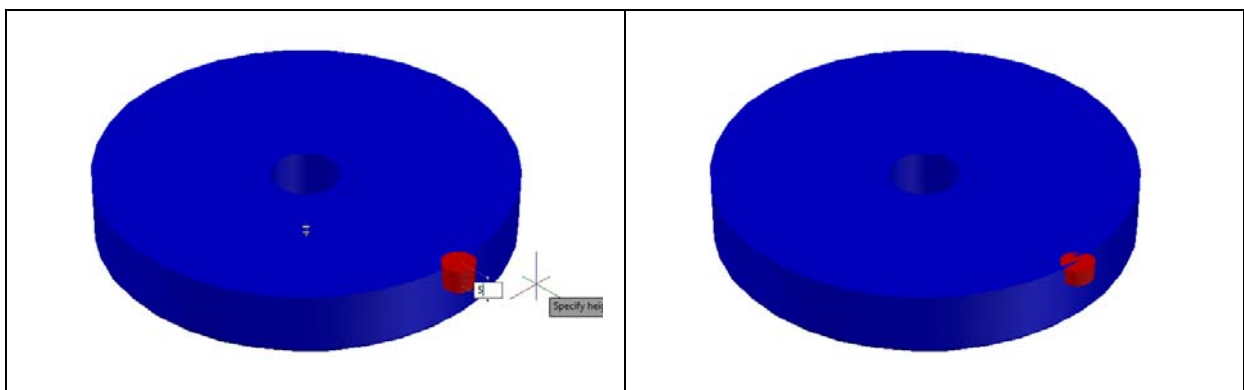


Figure 13.31– Select for extrusion

Figure 13.32 – The extruded 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.

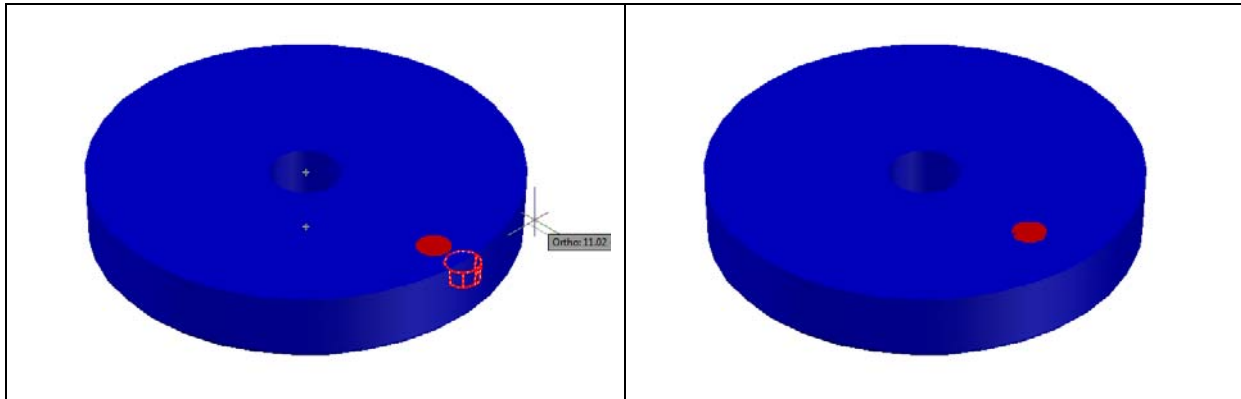


Figure 13.33 – Moving the solid

Figure 13.34 –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 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 13.33. For the “second point of displacement”, move the cursor to the right with the Ortho “On”, type **20.0** and **ENTER**. The cylinder will be ready for its polar array as shown in Figure 13.34.

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 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 cylinder and hit **ENTER** (Figure 13.35). 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 13.36). Enter **6** for the number of items, and when prompted to specify the angle to fill type **360** and **ENTER** (Figure 13.37). Five more cylinders should appear so that each is 120 degrees from the other as shown in Figure 13.38.

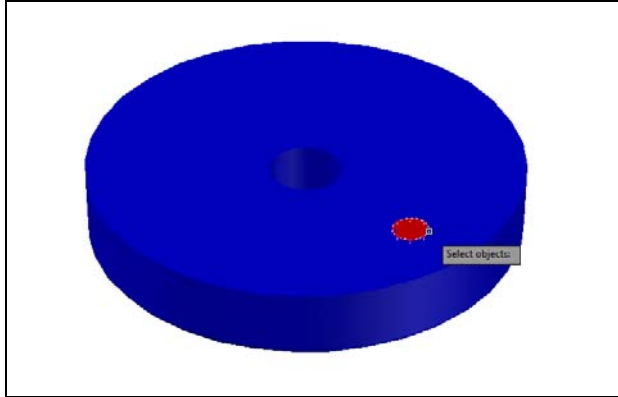


Figure 13.35 – Selecting the circle

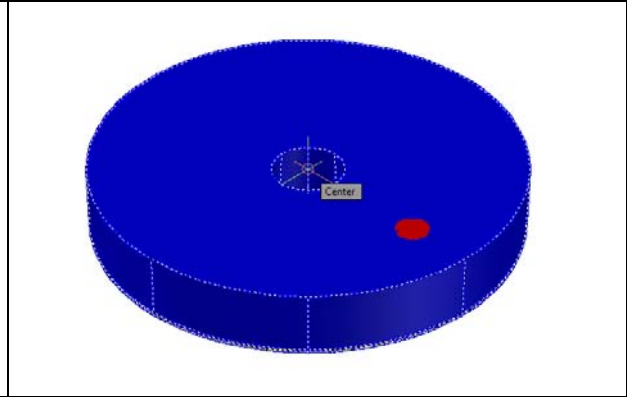


Figure 13.36 – The center of the array

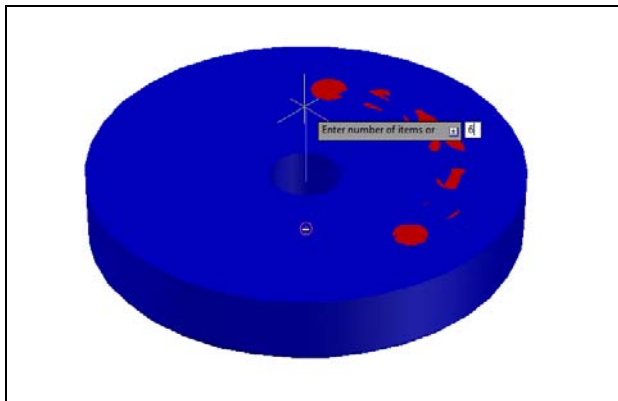


Figure 13.37 – Specifying the number of items

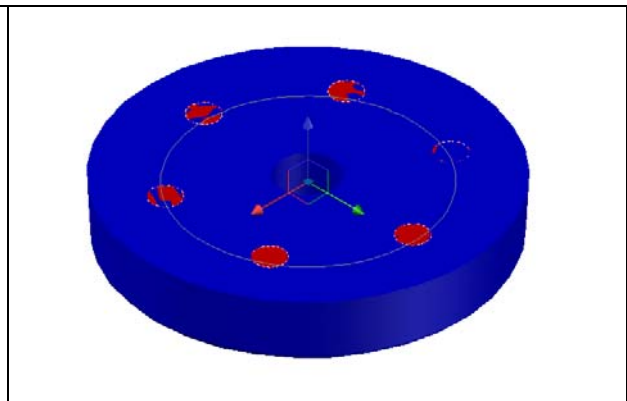


Figure 13.38 – The finished array, selected items

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.



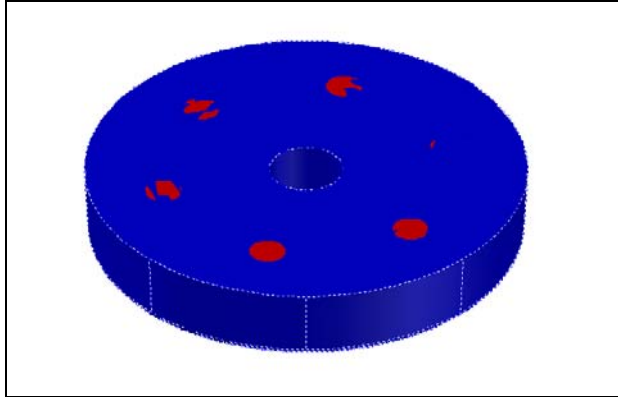


Figure 13.39 – The master solid

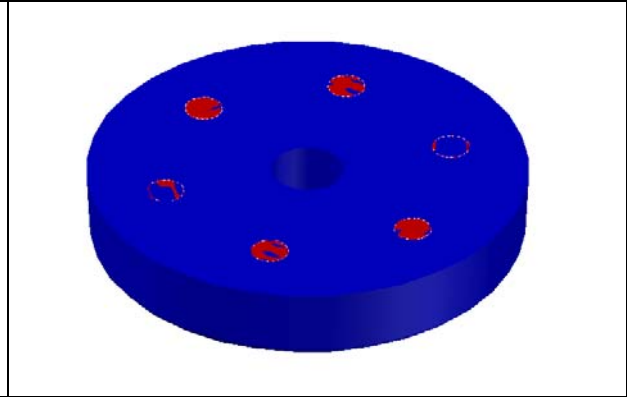


Figure 13.40 – Cylinders to be subtracted

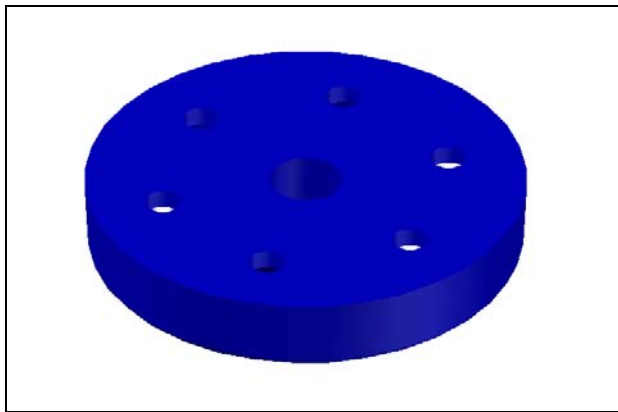


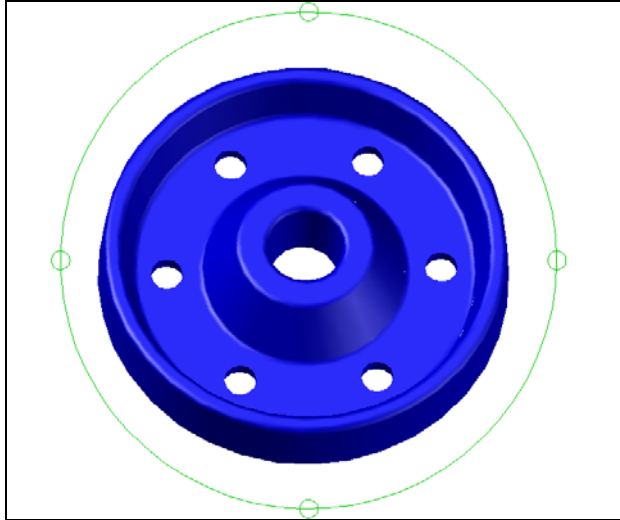
Figure 13.41 – Master solid with holes

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 13.39). 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 “**6 found**” (Figure 13.40). Hit **ENTER** and the shape of the six cylinders will disappear from the master solid as shown in Figure 13.41.

Using the Free 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 Free Orbit function will be the tool you may choose to quickly acquire the correct observation.





Select the Free Orbit tool by expanding the Orbit menu on the Navigation bar. A green circle appears on top of the graphical display. Click and drag with the left mouse button to change the orientation of the view any way you choose. Use the Free Orbit tool to rotate the solid into a position as shown in Figure 13.42. This will allow you to easily place the rounded edges on the six holes.

Figure 13.42 – Free Orbit into view

Add Curved Edges Using the Fillet Command

Use the Fillet command to add a radius edge to the corner of the six holes. Select the Fillet tool on the Modify panel and select the top tessellation line on the edge of a hole.

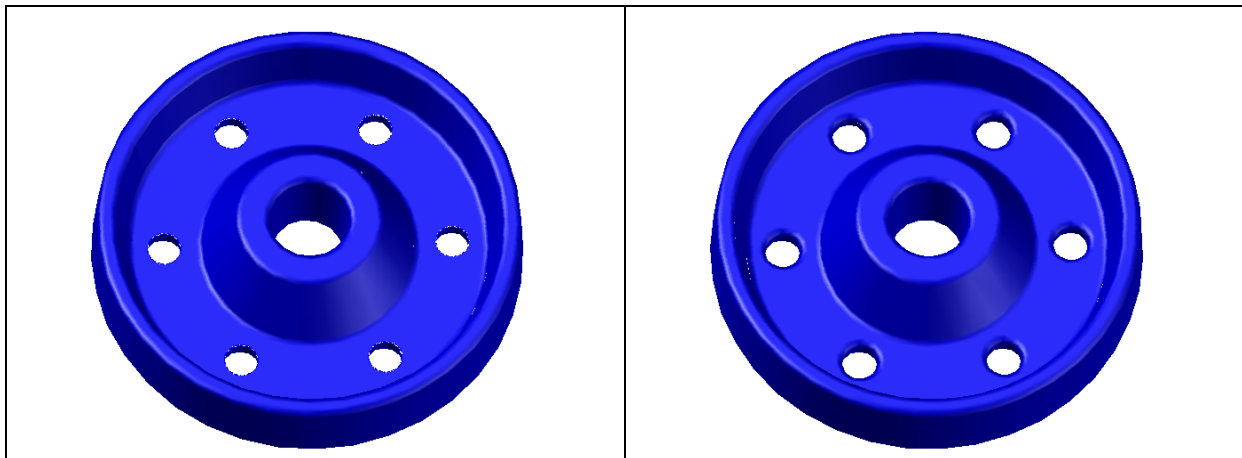


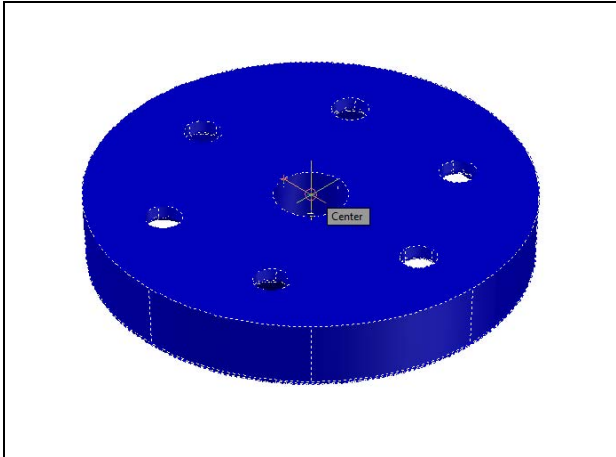
Figure 13.43 – The top edges of the holes

Figure 13.44 – Fillets placed on the holes

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 unit radius arc. Also, type **M** and **ENTER** so that you will be able to place all six fillets with this radius measurement. Select the top edge of one of the holes, and then hit **ENTER** to approve the radius measurement of 2. Continue by selecting each of the six holes as shown in Figure 13.43. Hit **ENTER** once more to place the rounded edges (Figure 13.44).

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 13.45, 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 Two 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 13.45 – 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 13.46 and hit ENTER to proceed to the second part of the function.

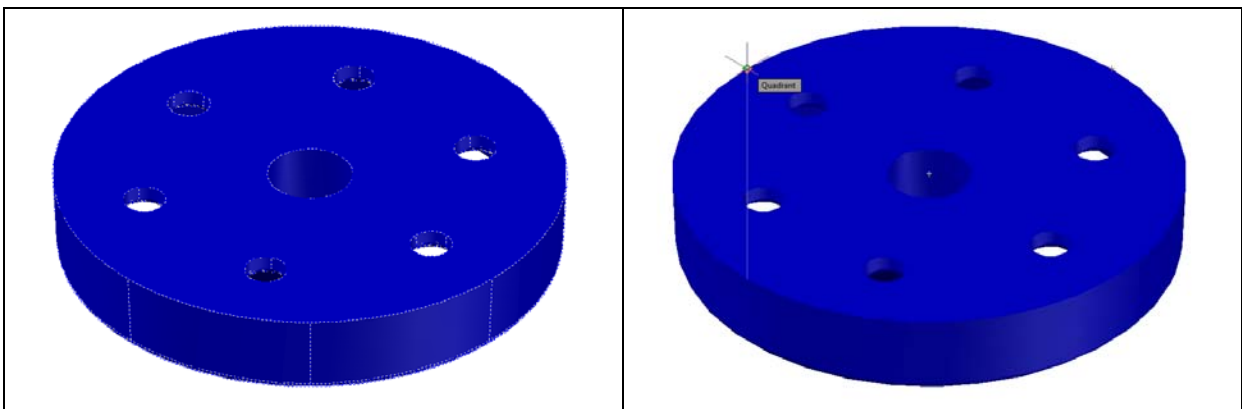


Figure 13.46 – Selecting the master solid

Figure 13.47 – Specifying the mirror plane

Type **XY** signifying you are going to mirror the solid part on the XY-plane. Select a quadrant of the top face of the master solid as shown in Figure 13.47. Hit **ENTER** once more to accept the default, not to delete the old side. Use the Union tool to bring both halves together.

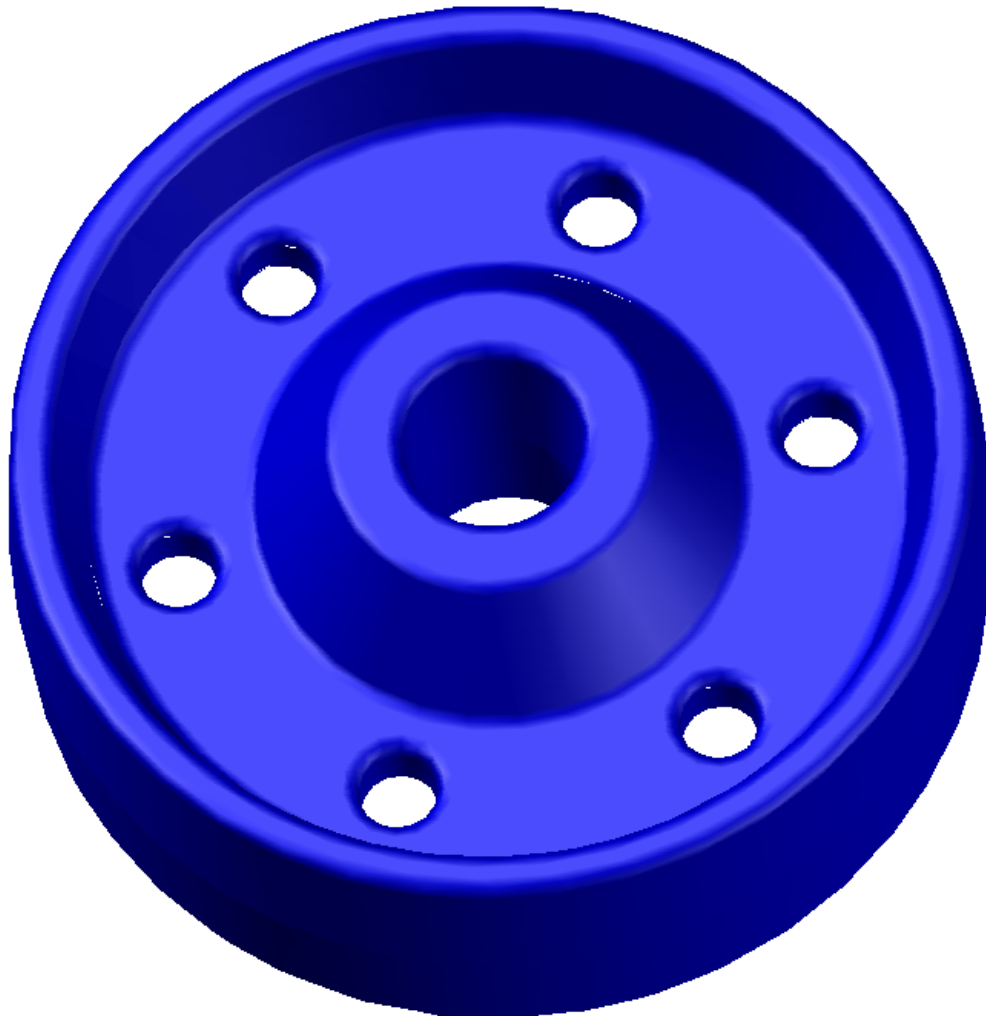


Figure 13.48 – The finished solid

Saving the Solid Problem

To save Revolving Problem Two 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 2” in the File Name textbox and hit the Save button to save the drawing (Figure 13.49).



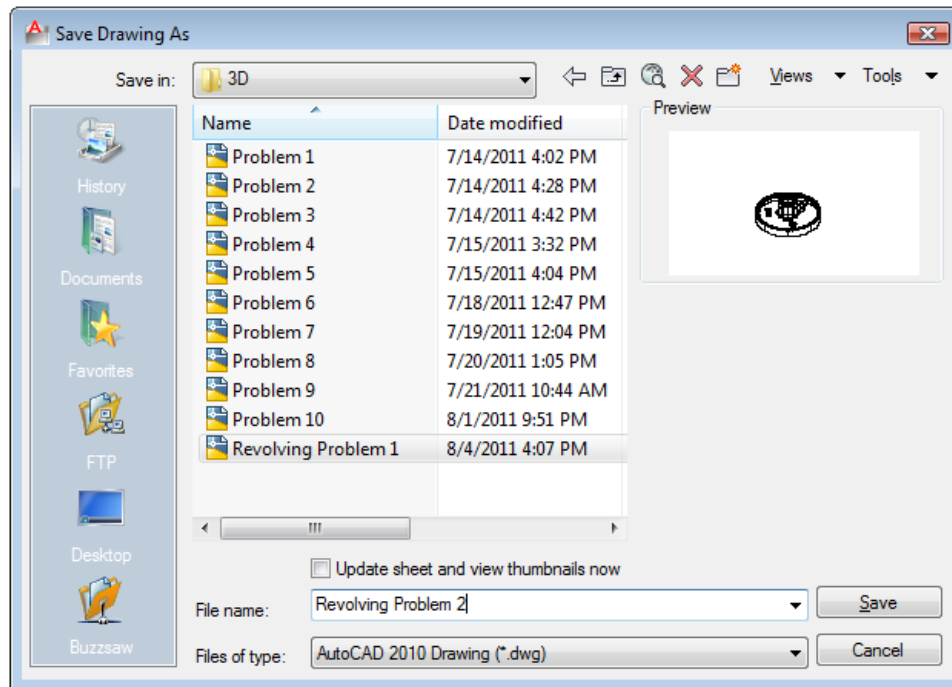


Figure 13.49 – Saving Revolving Problem Two

*** 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 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.**

