

# Chapter 13

## Revolving Problem Two

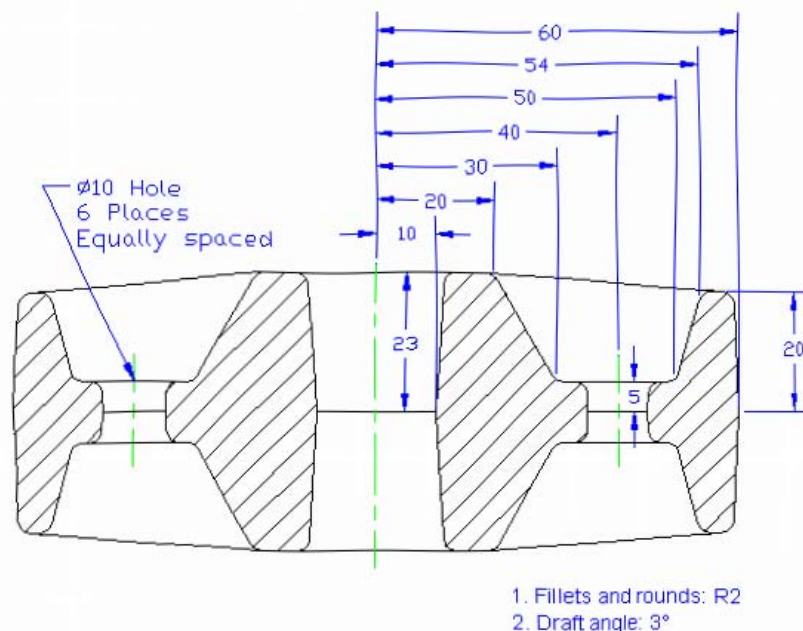
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In this chapter, you will learn the following to World Class standards:

1. Sketch of Revolving Problem Two
2. Starting a 3D Part Drawing
3. Modifying How the UCS Icon is Displayed
4. Creating Center and Mirror Lines Using the Line Command
5. Using the Offset Command to Create New Lines
6. Adding Two Lines Using the Line Command
7. Remove Lines Using the Erase Command
8. Adding Curved Edges Using the Fillet Command
9. Rotating Two Lines Using the Rotate Command
10. Adding Two Additional Curved Edges Using the Fillet Command
11. Adding Zero Radius Edges Using the Fillet Command
12. Creating a Perimeter Using the Polyline Command
13. Revolving a Solid from a Closed Polyline
14. Shading a 3D Solid
15. Rotating a 3D Solid Using the Rotate3D Command
16. Removing the Center Line Using the Erase Command
17. Drawing a Circle to be Extruded at a Taper
18. Extruding a Solid from Closed Polylines and Circles
19. Moving a Solid into Position on the Master Solid
20. Copying and Arranging a Solid Using the Array Command
21. Subtracting 3D Solids from the Master Solid
22. Moving the Solid to the Origin Point
23. Adding the Other Side Using Mirror3D Command
24. 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 the most complex shape of the component and then use simple geometric solids to subtract each detail. The exercises in revolving problem two involve you drawing just a portion of the component, revolving the section, and after adding the necessary detail, mirroring the second half and combining them together. In problem two, the perimeter contains rounded edges in the main shape that you will create using the Fillet tool. After revolving the Polyline perimeter into a solid, you will place a tapered and extruded solid onto that main solid, create an array, and then subtract the shapes. 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. You will combine the two halves using union in the final step of the problem.



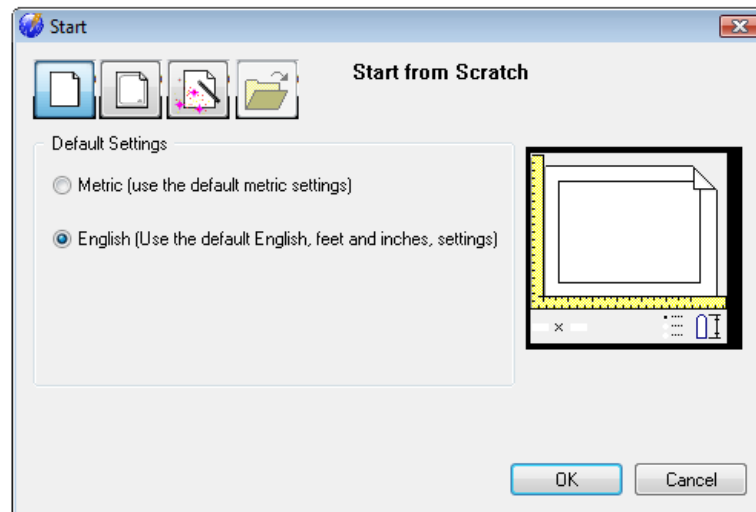
**Figure 13.1 – Revolving Problem Two Sketch**

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 by only leaving a few features to finish at the end. This problem will help you address solid parts containing complexities and sizes. All three revolving drills in the Fundamentals of 3D Drawing textbook will give you those repetitive maneuvers that will make you successful in 3D modeling.

## Starting a 3D Part Drawing

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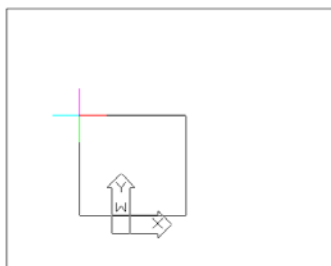
In Revolving Problem 2, you will begin the 3D drawing by selecting the New tool on the Standard Toolbar. Select the “Start from Scratch” (first button) at the top of the Start window. Choose to use English units as your Default Settings and hit OK (Figure 13.2). A new drawing file will open that contains the system variables and layer definitions which you will use later in Paper Space to finish the drawing.



**Figure 13.2 – Starting the Drawing**

## Modifying How the UCS Icon is Displayed

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When working in progeCAD, you may notice that the UCS icon has left its position in the corner of the display. This is because the default setting places the UCS at the origin of model space if the origin is in view. Many computer aided design operators do not appreciate having additional UCS lines in with their drawing or solid, so you can change the system to show the UCS in the lower left hand corner of the display at all times.

**Figure 13.3 - The UCS Toolbar**

Select Tools from the Menu Bar and then select Drawing Settings to pull up the Drawing Settings window. Under the Display tab, choose to Change settings for Display. In the UCS Icon section select the On radial button and then hit OK. Now the UCS icon will always appear in the corner of the display.

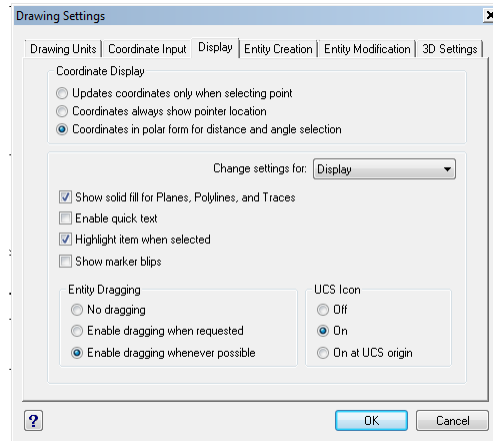
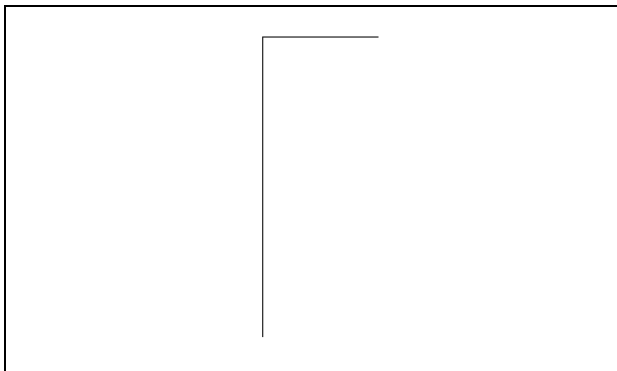


Figure 13.4 - Drawing Settings window, UCS Icon

## Creating a Center and Mirror Lines Using the Line Command

To begin this problem, you will draw a 23 line from right to left and then 60 down to create a center and mirror line.



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, move the line on the screen to the left and type **23** and **ENTER**, 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.

Figure 13.5 – Draw Two Lines

## 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 2D Drawing training module.

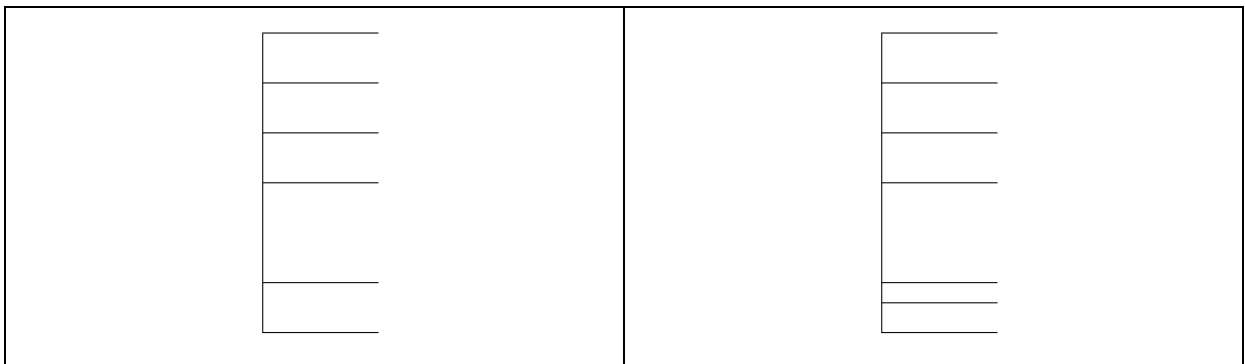


To create a new line 10 units below the horizontal centerline you will use the Offset command, which will allow you to create a line without selecting points to define the starting and endpoints. Choose the Offset tool on the Modify toolbar. After picking the Offset tool,

the command line will prompt you for the offset distance. Type **10** and hit **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 pick below that line to create another. Do this process once more 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, so type **20** and **ENTER**. Now select the last line you just created and pick below it to place the new line.

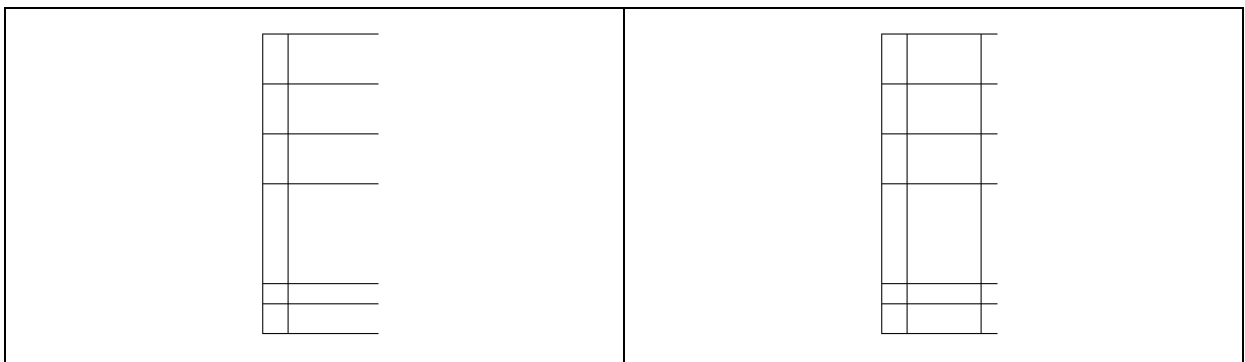
To create another horizontal line 10 below the last line, hit **ENTER** to repeat Offset command. The command line will prompt you for the offset distance, so type **10** and hit **ENTER**. Now select the last line and pick below that line to create a new line 10 below the one selected. All the lines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



**Figure 13.6 – Offset Lines by 10 and 20**

**Figure 13.7 – Offset a Line by 6**

To create the last horizontal line 6 units above the last line, hit **ENTER** to repeat Offset command. The command line will prompt you for the offset distance, so type **6** and **ENTER**. Now select the last line and pick above it to create a new line 6 units above. The line will appear as shown in Figure 13.7.

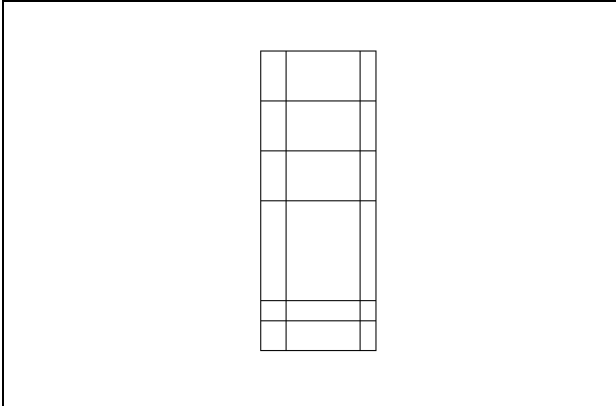


**Figure 13.8 – Offset a Line by 5**

**Figure 13.9 – Offset a Line by 20**

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

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

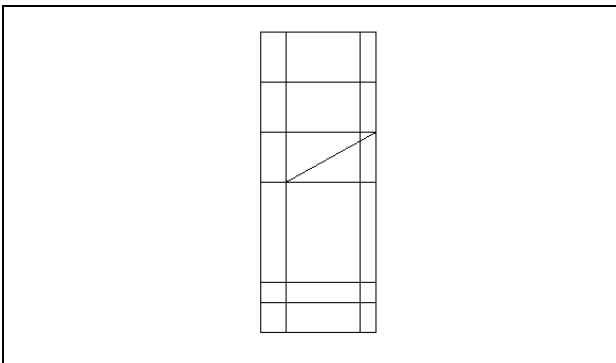


For the last offset line, create a new vertical line 23 units to the right of the vertical mirror line. Hit **ENTER** to repeat the Offset command. The command line will prompt you for the offset distance, so type **23** and **ENTER**. Now select the vertical mirror line and then pick to the right of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

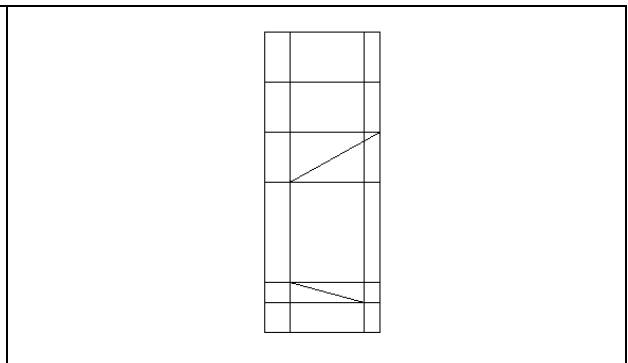
**Figure 13.10 – Offset a Line by 23**

## Adding Two Lines Using the Line Command

Next you need to place two lines using the Line tool from the Draw toolbar. With your mouse, select points using the Intersection Esnap 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**



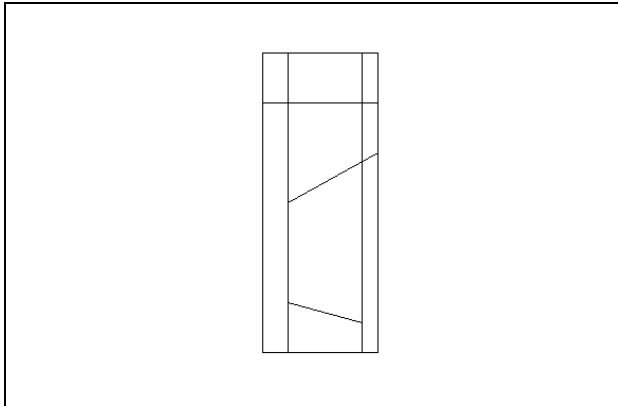
**Figure 13.12 – Add Second Angled Line**

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

## Removing the Four Lines Using the Erase Command

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Again, do not excessively rely on the erase function because removing entities once drawn will lower your efficiency. Now that the two angled lines are present, you can eliminate four construction lines



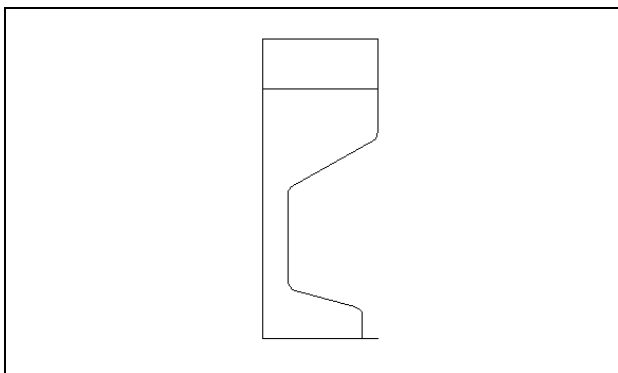
Use the Erase tool on the Modify toolbar to remove the four horizontal lines that you no longer need. Select the Erase tool on the Modify toolbar, and select the four horizontal lines that served as the starting and ending points of the angled lines. Hit **ENTER** and this will remove the four lines as shown in Figure 13.13. All of the remaining lines will change length as you use the 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

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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 in this exercise you will modify the lined matrix by placing the arc in the 2D perimeter.



Pick the Fillet tool on the Modify toolbar and then type **R** and **ENTER**. Type **2** and **ENTER** to set the system for a 2-unit radius arc. Select the two lines representing a corner where one of the angled lines meet a vertical line and the arc will appear on the edge of the perimeter as shown in Figure 13.14. There are four arcs in the perimeter. Hit **ENTER** to repeat the Fillet command each time.

**Figure 13.14 – Adding Four Fillets**

## Rotating Two Lines 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 toolbar. Select the first offset line you drew as shown in Figure 13.15. To continue with the command, hit **ENTER** to select the base point of rotation. Select the Endpoint of the line as shown in Figure 13.15. Then type **-3** and **ENTER** for the degrees of counterclockwise rotation to finish this detail. The rotation will appear as shown in Figure 13.16.

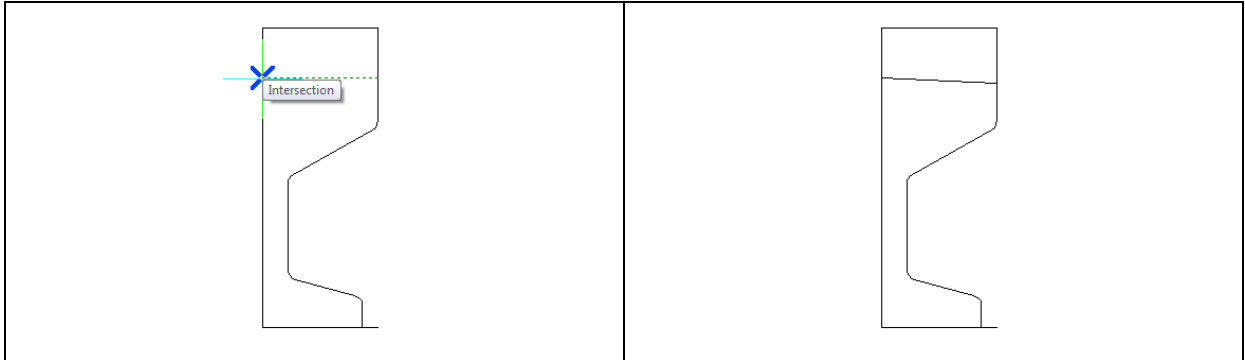


Figure 13.15 – Base Point for Rotation

Figure 13.16 – First Rotated Line

Hit **ENTER** to repeat the Rotate command and pick the bottom horizontal line as shown in Figure 13.17. To continue with the command, hit **ENTER** to select the base point of rotation. Select the Endpoint of the line as shown in Figure 13.17. Then type **3** and **Enter** for the degrees of counter clockwise rotation to finish this detail. The rotation will appear as shown in Figure 13.18.

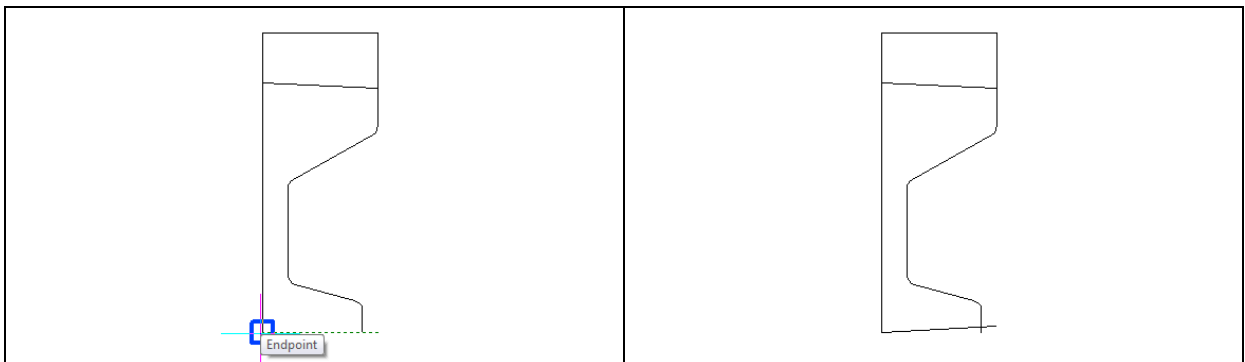


Figure 13.17 – Base Point for Rotation

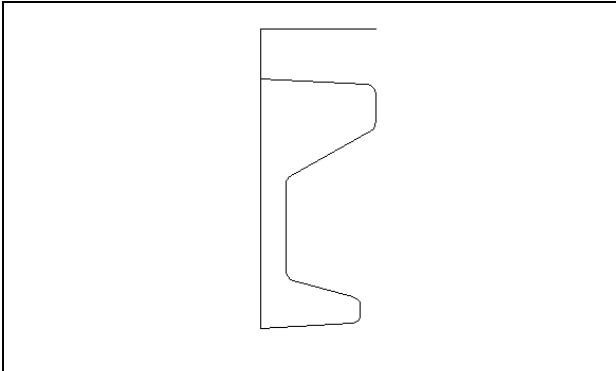
Figure 13.18 – Second Rotated Line



## Add Two Additional Curved Edges Using the Fillet Command

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



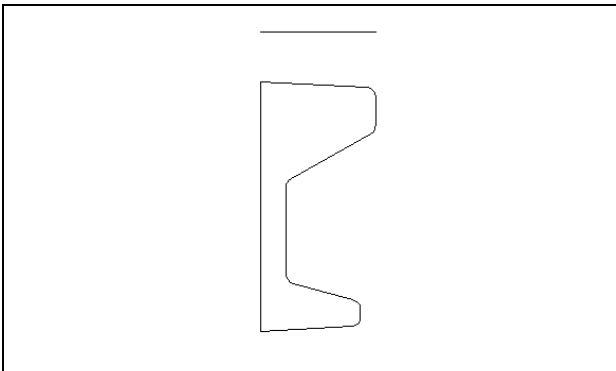
Pick the Fillet tool on the Modify toolbar. You do not need to set the radius since the radius measurement in the function is already set at 2. Select the two lines representing a corner where one of the rotated lines meets a vertical line and the arc will appear on the edge of the perimeter as shown in Figure 13.19. There are two arcs in this boundary. Hit **ENTER** to repeat the Fillet command each time.

Figure 13.19 – Two More Fillets

## Add a Zero Radius Edge Using the Fillet Command

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You can use the Fillet command to create a zero radius edge on a corner 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 toolbar and then type **R** and **ENTER**. Type **0** and **ENTER** to set the system for a zero radius arc. Select the top side and left side and the sharp corner will appear on the top left corner 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, and place another corner on the bottom left corner of the perimeter.

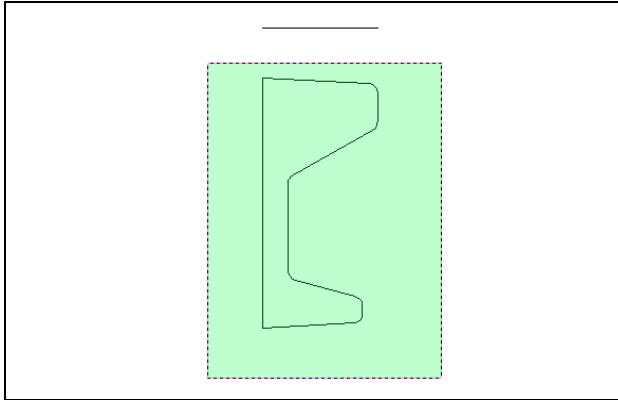
Figure 13.20 – Placing Two 0 degree Fillets

## Creating Polylines Using the Edit Polyline Tool

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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 then select one of the objects.





Type J and ENTER to go to the Join feature of the Edit Polyline tool. You will be prompted to “Select entities to join.” Use a selection window to select all of the lines in and arcs in the perimeter and hit ENTER. The command line will respond with “Polyline is now closed. 12 vertices added to polyline.” Hit Escape to exit the Edit Polyline command

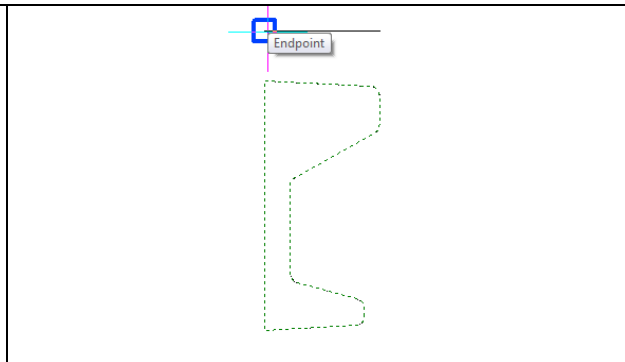
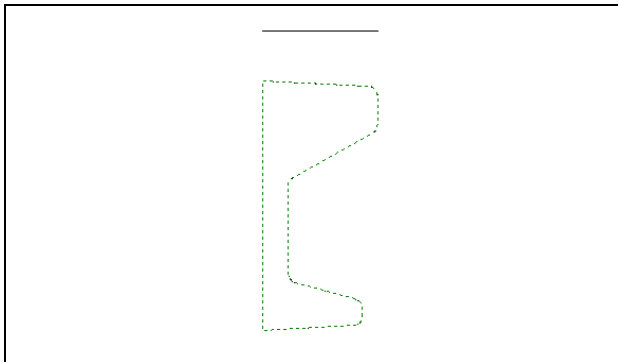
**Figure 13.21 – Making a Complex Polyline**

## 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 on the Solids toolbar and the command line will prompt you to “Select entities:”. Pick the perimeter of the Polyline and the command line will return with “1 found” as shown in Figure 13.22. Next, choose two points to describe the axis of rotation. Pick the Endpoint of the centerline as shown in Figure 13.23 for the first point. Pick the Endpoint of the centerline as shown in Figure 13.24 for the second point.



**Figure 13.22 – Select the Polyline**

**Figure 13.23 – The First Point of the Axis**

For the “Specify angle of revolution <360>:” just hit ENTER to revolve the polyline the default value of 360 degrees. 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 with the Z-axis rising vertically as shown in Figure 13.25.

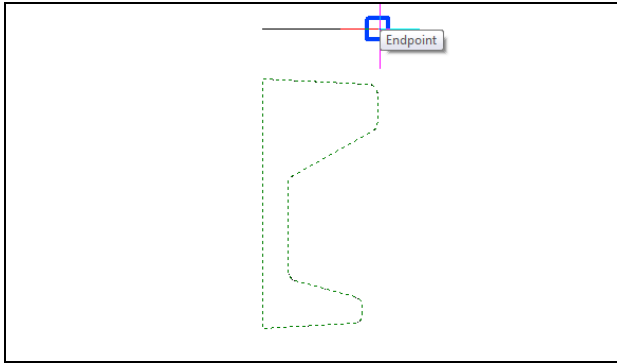


Figure 13.24 – Second Point of the Axis

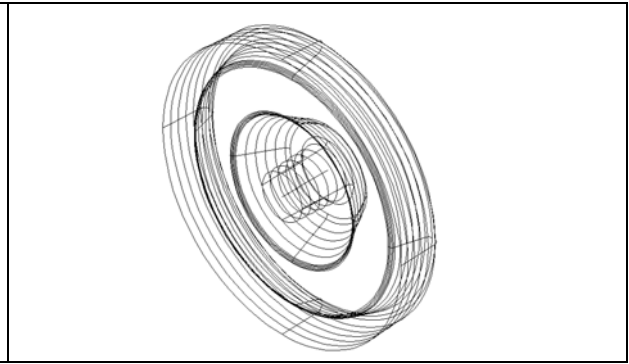


Figure 13.25 – The Revolved Solid

## Shade the 3D Solid

To view the 3D solid in a more lifelike way, select one of the Shade tools on the Shade 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.

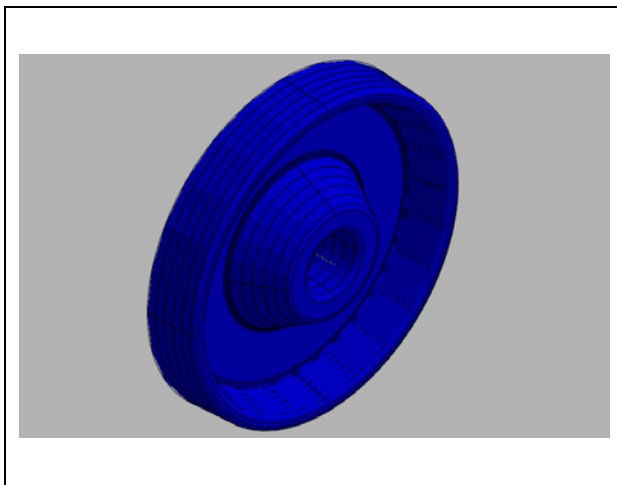
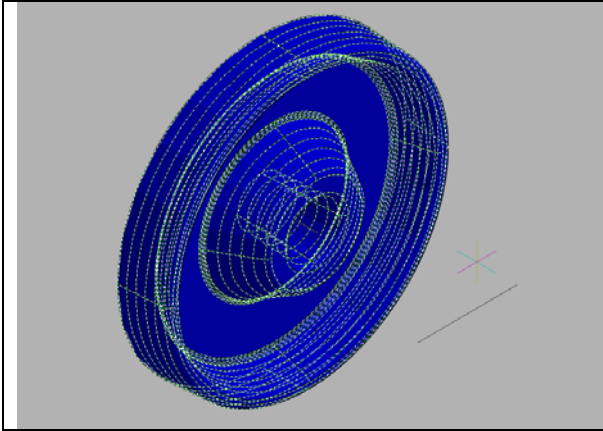


Figure 13.26 – Shade the Revolved Solid

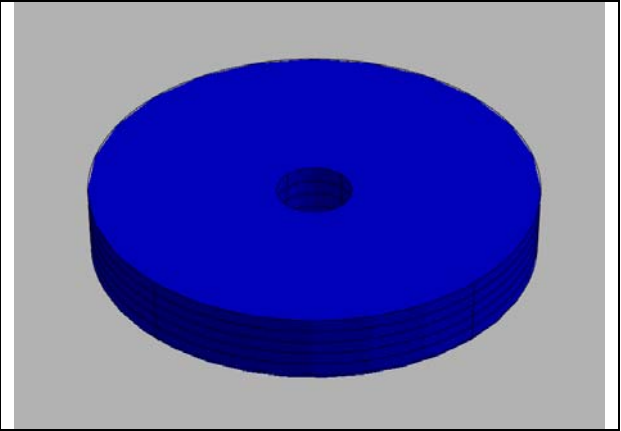
Use the Southeast Isometric view to see the front of the part. Select a Shade tool on the Shade toolbar. The part is now a solid blue color as shown in Figure 13.26. The tessellation lines associated with the Hide command are no longer interfering with your ability to read the drawing. Your customers will most likely prefer this mode to the Hide command because they will be able to visualize the product clearly. If you ever desire to return to standard viewing, choose the 2D Wireframe tool on the Shade toolbar to return the drawing to the previous settings.

## 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 entities to rotate:”, 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 any point on the display and then with the Ortho mode still “on” pick a second point drawing a line on the y-axis as shown in Figure 13.27. Type **90** and **ENTER** to rotate the solid as shown in Figure 13.28. You can also erase the Centerline that did not get rotated.



**Figure 13.27 – The Rotation Axis**



**Figure 13.28 – The Rotated the Solid**

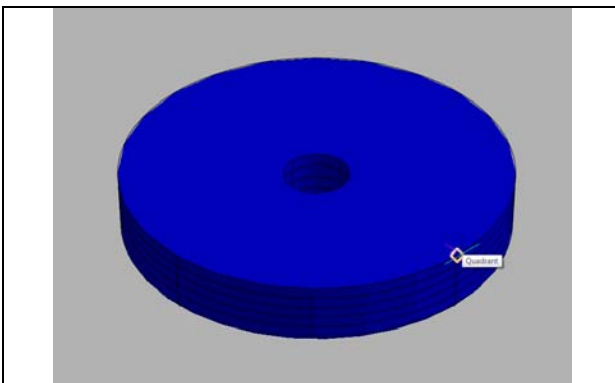
## Creating a New Layer

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 entities on that 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 Explore 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”.

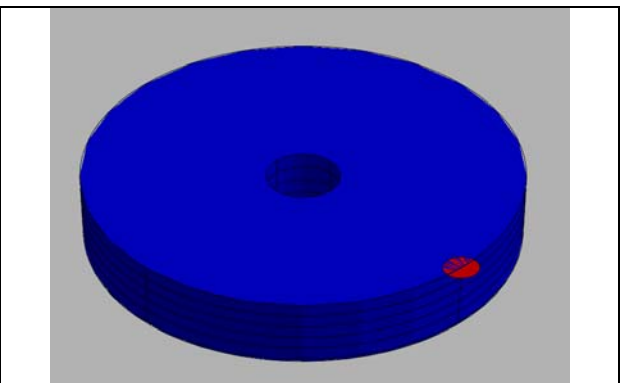


## Drawing a Circle to be Extruded at a Taper

You need to add six holes to the master solid. You will draw one hole going completely through the master solid, and the second other five holes will be a copied from the first after you perform a polar array. These openings will not be straight, but will be at an angle for ease of casting. Make sure to switch to the Cylinder layer before starting the Circle command.



**Figure 13.31 – Center of a Circle**



**Figure 13.32 – The Circle**

On the Draw toolbar, pick the Circle 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

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Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Solids toolbar. You can extrude multiple Polylines and Circles at a time with the Extrude command.



Select the Extrude tool on the Solids toolbar and the command line will prompt you to “**Select entities:**”. Use a selection window to select the circle. The command line will return with “**Entities in set: 1**”. Hit **ENTER** to specify the height or path of the solid. Type “**-5**” for the height of the extrusion. Type **-3** as the angle of taper and hit **ENTER**. The cylinder will extend down into the z axis 5 units with a 3 degree taper as shown in Figure 13.34.

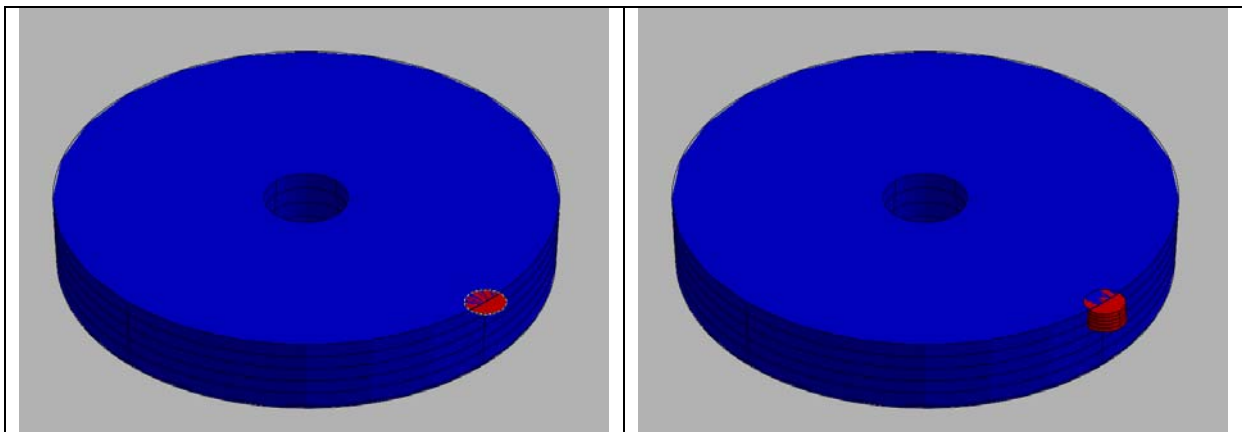


Figure 13.33 – Selecting for Extrusion

Figure 13.34 – The Extruded Circle

## Moving 3D Solids into Position on the Master Solid

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The Move command is actually a three-dimensional function in progeCAD Professional. The way you can determine whether a command is three dimensional in nature is to attempt to use the function outside the Z-plane. You can use move to relocate any entity to another point (X,Y,Z) in Model Space.



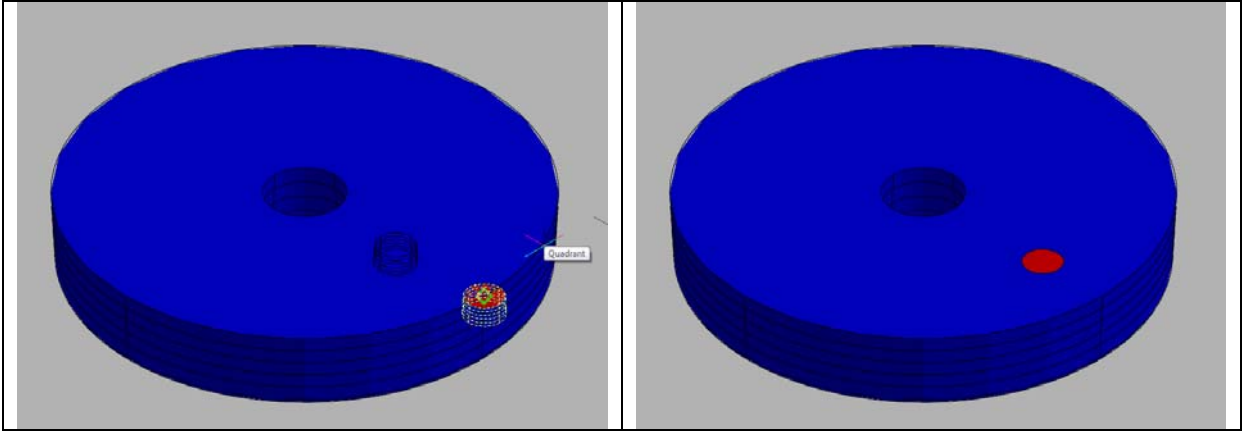


Figure 13.35 – Moving the Solid

Figure 13.36 –The Solid in Position

To move the cylinder, you select the Move tool on the Modify toolbar. Once you choose the Move command, you will be prompted to “Select entities to move:”. Pick the extruded circle and the command line will respond with “**Entities in set: 1**”, so hit ENTER to go the second part of the command. Next, you need to choose a “Vector/<Base point>:”, so select a point anywhere on the graphical display. Again with the Ortho on, move your cursor towards the center of the part and type 20 and ENTER at the command line. The extruded circle will move 20 units in the direction of your cursor. 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 into six, spacing each one 60 degrees apart. Use the Array function with the polar option as the tool for this operation.

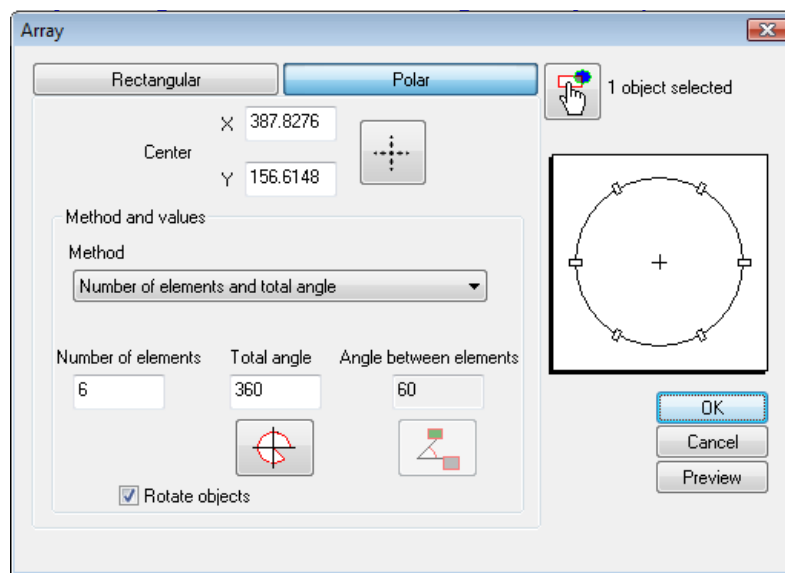
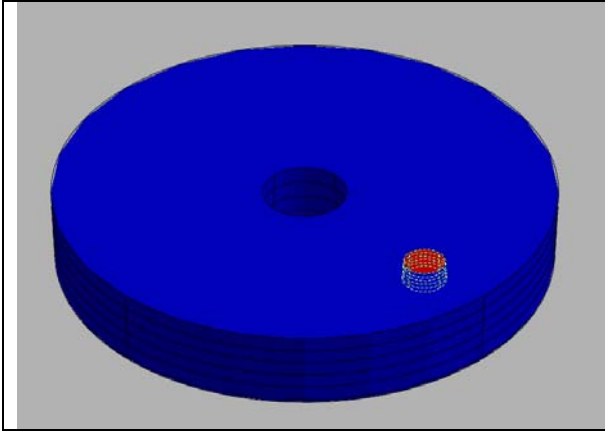
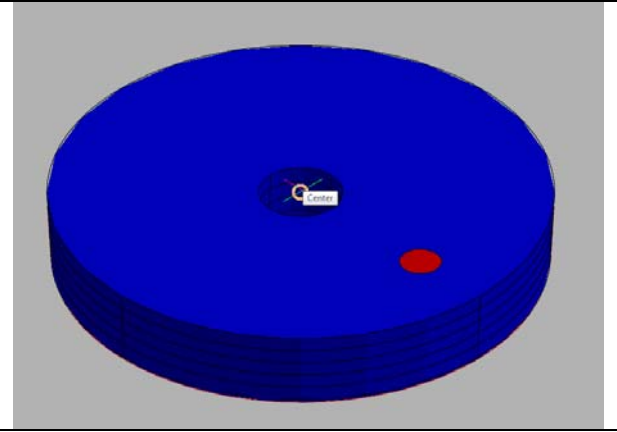


Figure 13.37 – The Array Window

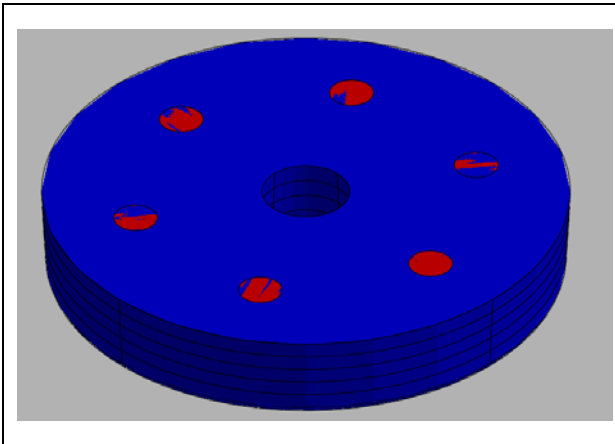
Select the Array tool on the Modify toolbar, which will bring up the Array window as shown in Figure 13.37. Pick the Polar option, and then the Select Objects button that will allow you to select the tapered cylinder as shown in Figure 13.38. Hit **ENTER** to return to the Array window. Next select the Pick Center Point button which will bring you back to the drawing to select the center of the part as shown in Figure 13.39.



**Figure 13.38 –Center of the Polar Array**



**Figure 13.39 – Subtract Solids**



**Figure 13.40 – Six Solids to Subtract**

Change the “Number of Elements” textbox to “6”. Making sure the total angle reads 360 degrees and the Rotate Objects check box is selected, hit 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.

## Subtracting 3D Solids from the Master Solids

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





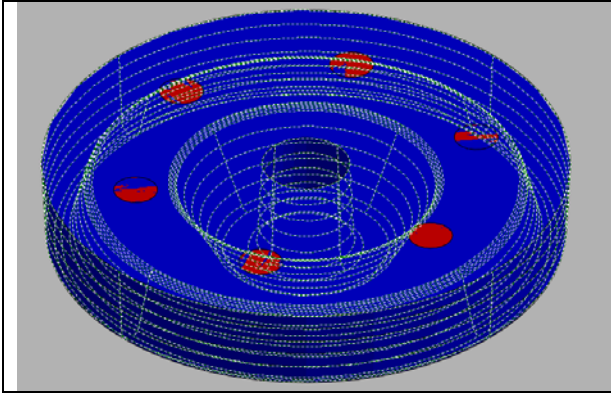


Figure 13.41 – Subtract Solids

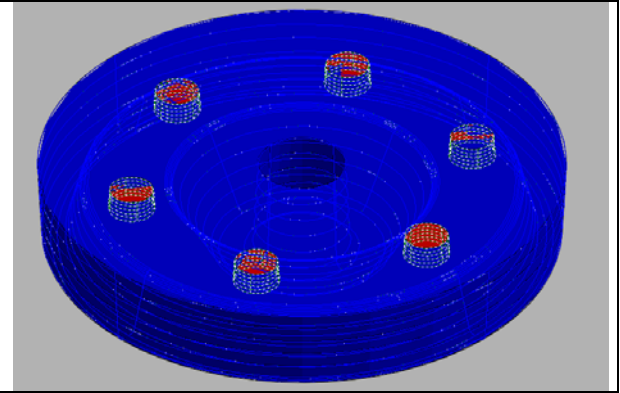


Figure 13.42 – Subtracting Solids

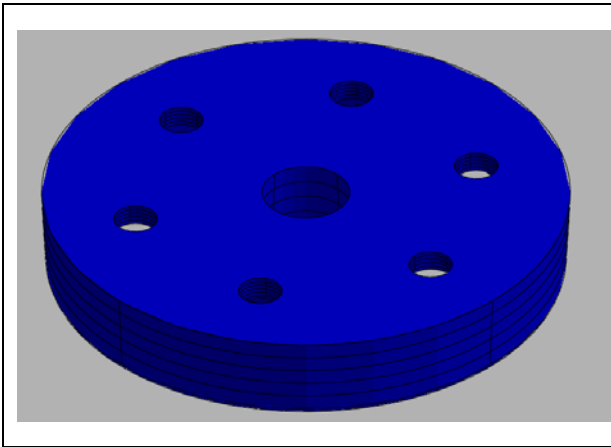


Figure 13.43 – Master Solid with Holes

The command line will prompt you to “Select ACIS object to subtract from:”. Select the larger master solid and the system will respond with “Entities in set: 1” (Figure 13.41). Hit **ENTER** to proceed to the next step, which is to “Select ACIS objects to subtract:” (Figure 13.42). Pick the tapered cylinders inside of the master solid and the command line will reply with “Entites in set: 6”. Hit **ENTER** and a region exactly the shape of the cylinders will be removed from the main solid.

## 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 a feature you can use to change your view.

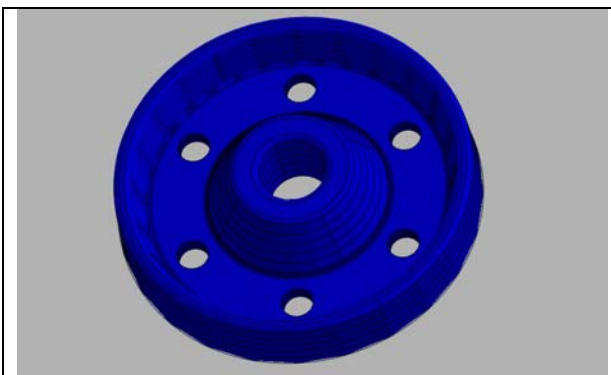


Figure 13.44 – 3D Orbit into View

Holding CTRL on the keyboard, click and drag with the left mouse button to rotate the part in view. Move the solid into a position as shown in Figure 13.44. Use Zoom Extents if necessary to maximize your view of the part.



## Add a Curved Edges Using the Fillet Command

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To add a rounded edge to the top of the six holes, use the Fillet command.

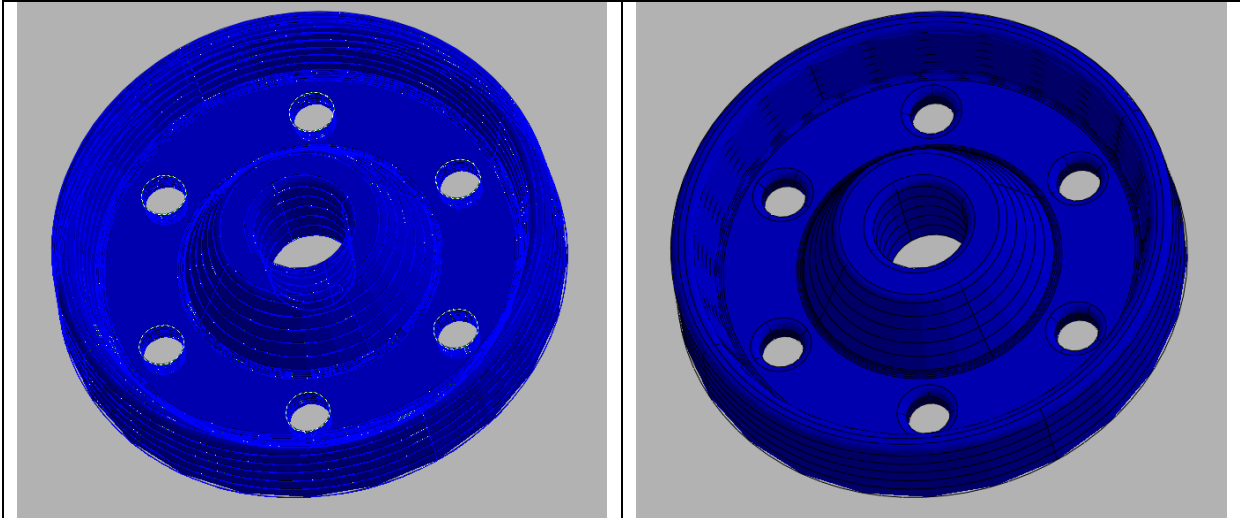


Figure 13.45 – Pick the Top of the Holes

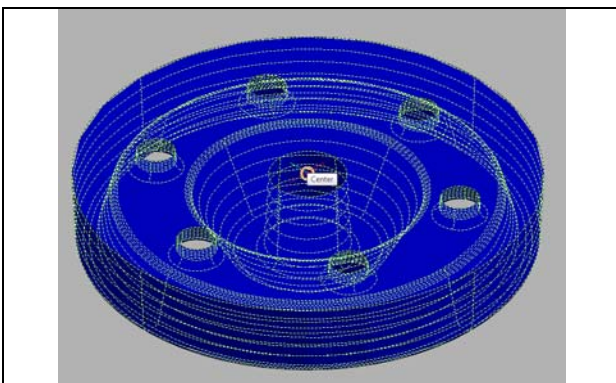
Figure 13.46 – Fillets on the Holes

Pick the Fillet tool on the Modify toolbar 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 edge will appear on the edge of the each hole as shown in Figure 13.46.

## Moving the Solid to the Origin Point

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Switch to the Southwest Isometric View. To move revolving problem two 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.



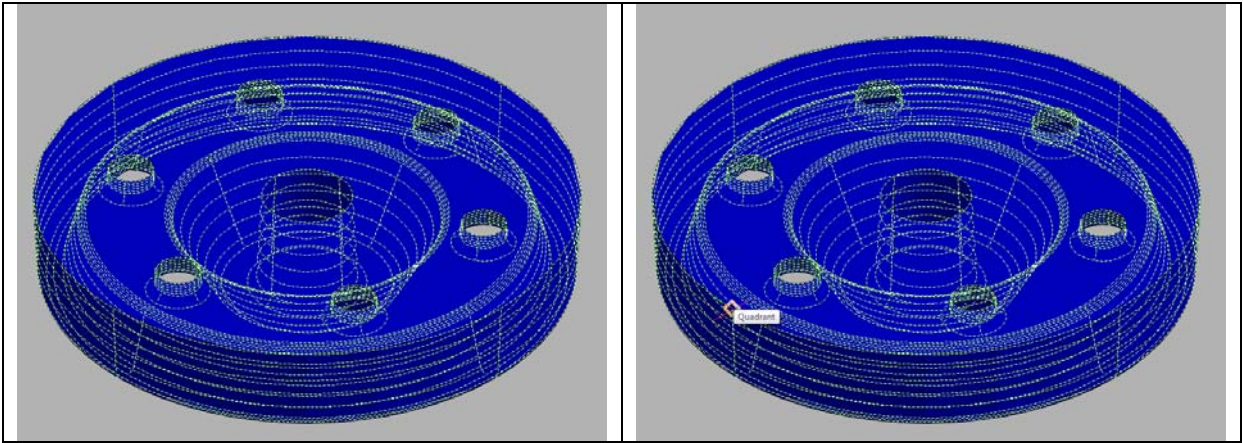
When prompted to select a “**Vector/<Base point>:**,” select the endpoint shown in Figure 13.47 at the top center of the revolving problem two, which will become its insertion point if it were inserted into an assembly drawing. For the prompt to select a “**Displacement point:**,” type **0,0,0** and hit ENTER. Problem one will move to the new origin point. Try using Zoom Extents if the part completely escapes your viewing area.

Figure 13.47 – Moving to the Origin

## Adding the Other Side Using Mirror3D Command

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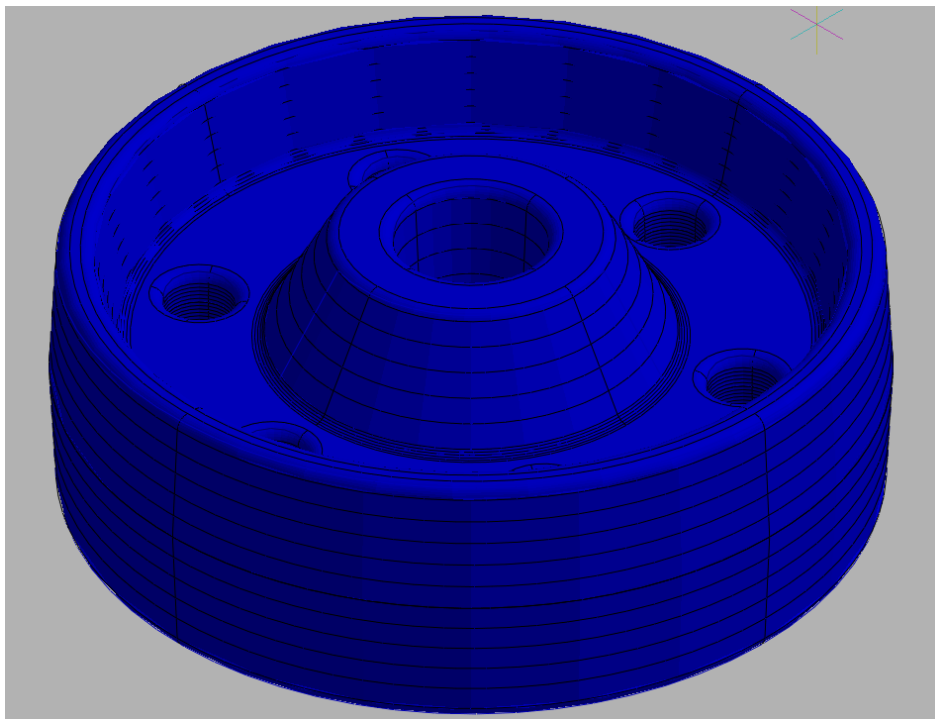
Select **Modify** on the Menu Bar, then **3D Operations** and pick **Mirror3D** from the list of commands. At the command “Select entities:”, pick the solid on the graphical display as shown in Figure 13.48 and hit **ENTER** to proceed to the second part of the function.



**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 as the point on the XY Plane. Hit **ENTER** once more to keep the original solid.

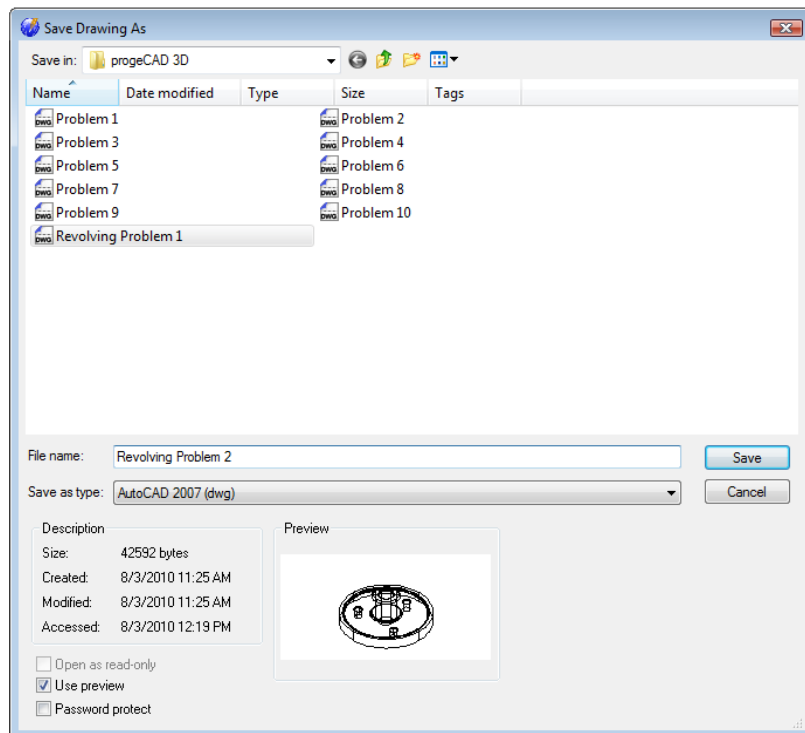


**Figure 13.50 – Moving to the Origin**

**\* World Class CAD Challenge 101-12 \*** - 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.

## Saving the Solid Problem

To save Revolving Problem 2, 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 2” and select the Save button to save the drawing (Figure 13.51).



**Figure 13.51 – Saving Revolving Problem Two**

**\* World Class CAD Challenge \*** - Report your best times to World Class CAD at [www.worldclasscad.com](http://www.worldclasscad.com) to obtain your world class ranking.