

# Chapter 12

## Revolving Problem One

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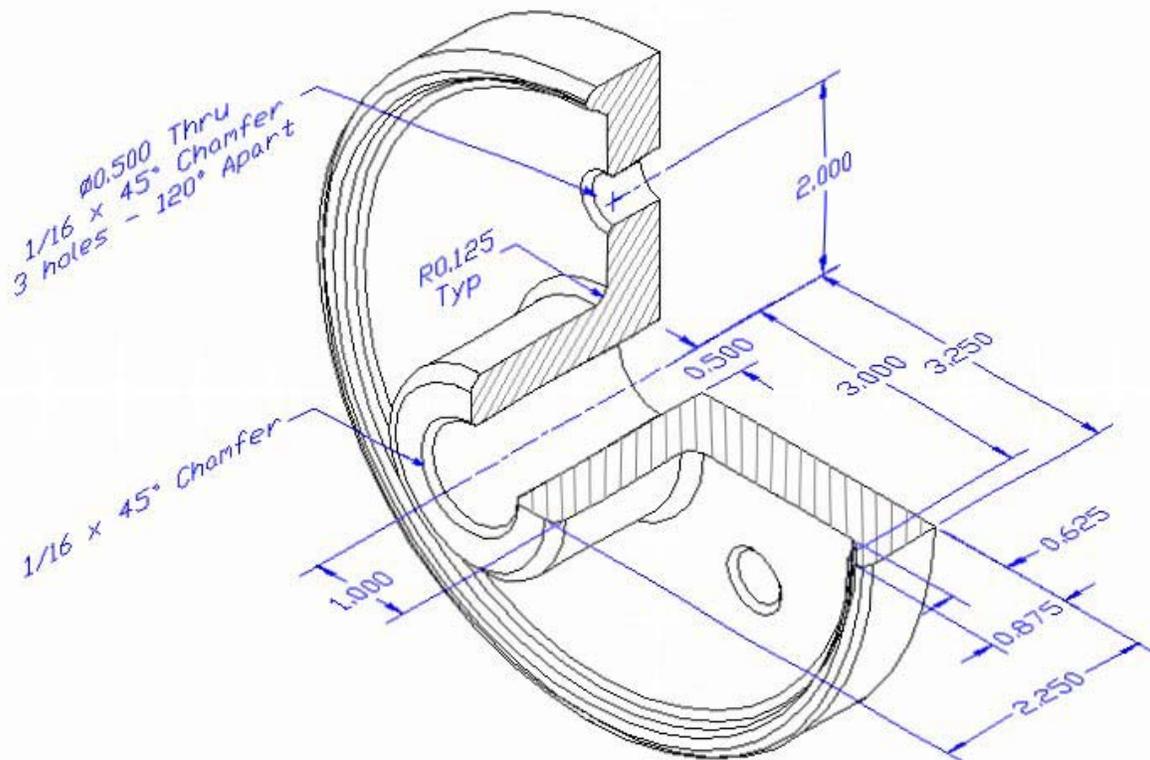
In this chapter, you will learn the following to World Class standards:

1. Sketch of Revolving Problem One
2. Starting a 3D Part Drawing
3. Modifying How the UCS Icon is Displayed
4. Draw a Center Line Using the Line Command
5. Creating a Perimeter Using the Polyline Command
6. Add Curved Edges Using the Fillet Command
7. Revolve a Solid from a Closed Polyline
8. Rotating a 3D Solid Using the Rotate3D Command
9. Remove the Center Line Using the Erase Command
10. Drawing a Solid Cylinder
11. Moving a Solid into Position on the Master Solid
12. Array a Solid Using the Array Command
13. Subtracting 3D Solids from the Master Solid
14. Hide the Solid
15. Moving the Solid to the Origin Point
16. Saving the Solid Problem

## Sketch of Revolving Problem One

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You will start the first revolving problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across the Z-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. In problem one, the master solid contains radius edges in the main shape, where you get a chance to use the Fillet tool. After revolving the Polyline perimeter, you will place a cylinder onto that main solid, array the cylinder and subtract them. Finally, you will utilize the Chamfer command and put a beveled entry on the three mounting holes and the hole in the hub.



**Figure 12.1 – Revolving Problem One Sketch**

Remember, when you receive a sketch from a professional, you need to practice quickly identifying the different shapes and deciding 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 one, you will begin the 3D drawing by selecting the Mechanical template. In Figure 12.2, the window “Select template” is loaded by selecting the New tool on the Standard Toolbar. Scroll down the list and select Mechanical.dwt. A new drawing file will open that contains the system variables and layer definitions, which you will use later in Paper Space to finish the drawing.

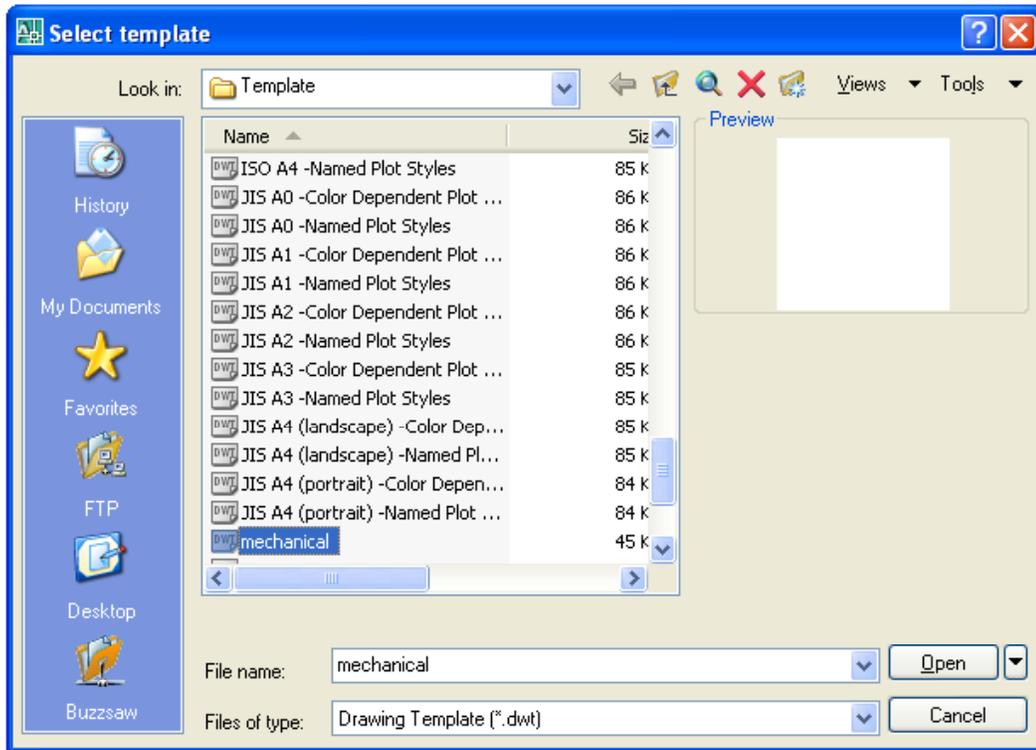


Figure 12.2 – Starting the Drawing Using the Mechanical Template

## Modifying How the UCS Icon is Displayed

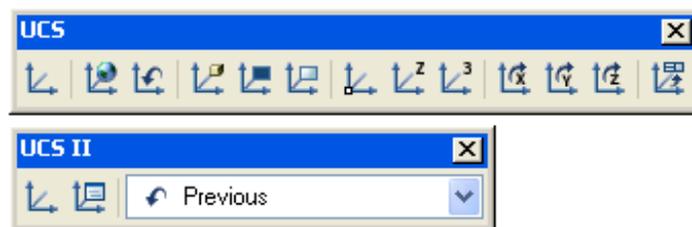


Figure 12.3 - The UCS Toolbar

When working in AutoCAD, you may notice the UCS icon moving on the graphical display showing the precise location of the origin. Many computer aided design

operators do not appreciate having additional UCS lines in with their drawing or solid, so you can set the system to show the UCS in the lower left hand corner of the display.

Select the Display UCS Dialog tool on the UCS II toolbar (Figure 12.3) and the UCS window will appear the graphical display. Pick the Settings tab to change the UCS Icon settings. Remove the check from “Display at UCS origin point” as shown in Figure 12.4 and press the OK button to have the system variable change. Now you can start to create the solid.

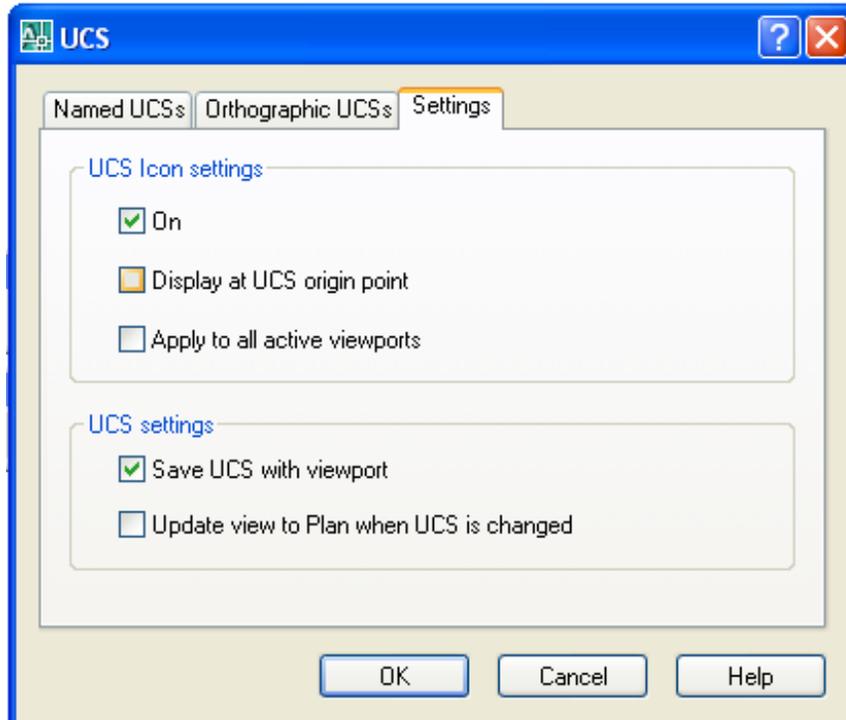
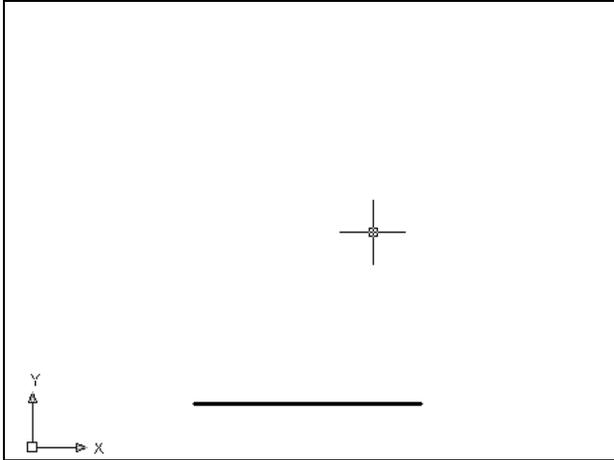


Figure 12.4 - The UCS Window, Settings Tab

## Creating a Center Line Using the Line Command

First, you need to draw a 2.25 line with the Line tool from the Draw toolbar. With your mouse, select any point at the lower right side of the graphical display.



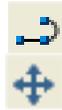


Even while in the Line command, continue with the Ortho mode On to keep the lines perfectly horizontal when drawing. Move the line on the graphic screen to the left and type “**2.25**”. Hit **Enter** to complete the function. A line will be on your graphical display as shown in Figure 12.5. You will use a relative coordinate to start the Polyline above the centerline, so you will start the line on the right and finish on the left. Your Polyline will start on the left and above, working counterclockwise.

**Figure 12.5 – Draw a Center Line**

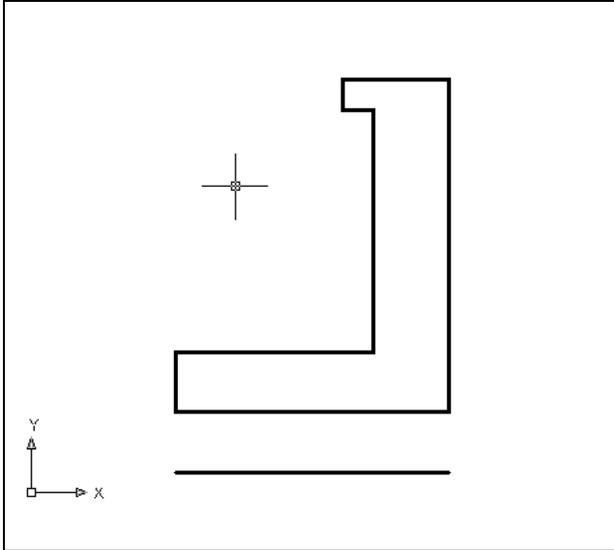
## Creating a Perimeter Using the Polyline Command

You will still need a closed Polyline to revolve a circular solid, and you will start the Polyline 0.50 above the centerline. The method to accomplish that task is to Draw the polyline on the centerline and then Move it .5 up from the centerline. Select the left endpoint of the centerline as the starting point of the polyline. The command line information is shown in Figure 12.6.



```
Command: _pline
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: 2.25
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: 2.75
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 0.875
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: .25
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: .25
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 2
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 1.625
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: c
```

**Figure 12.6 – The Polyline Command Lines for the Perimeter**



With the Ortho mode “on”, draw a **2.25** unit line to the right, a **2.75** line up, a **0.875** line to the left, a **0.25** line down, a **0.25** line to the right, a **2.0** line down, a **1.625** line to the left and then type “C” to close the Polyline. The AutoCAD program computes the last line length and angle when you use the close option. A closed entity will appear in the graphical display as shown in Figure 12.7. Now use the Move tool: select the polygon and move it .5 up from the centerline. Select the Zoom Extents tool on the Standard toolbar to have the box fill the monitor, so you can proceed to the next step in the drawing process.

**Figure 12.7 – Polyline for the Perimeter**

## 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 Polyline entity by placing the arc in the 2D perimeter.



Pick the Fillet tool on the Modify toolbar and then type **R** and **Enter**. Type **0.125** and **Enter** to set the system for a 1/8 radius arc. Select the vertical line as shown in Figure 12.8, and then select the horizontal line as shown in Figure 12.9 and the 0.125 arc will appear on the edge of the 2D Polyline as shown in Figure 12.10.

There are five arcs in this Polyline. You may notice that there is an Arc option in drawing a Polyline, but you would have to calculate the length of the lines, by subtracting the radius. Like most designers, you will come to believe that the computer-aided software should handle the mathematics whenever possible; therefore, you will place chamfers and fillets on the 2D Polyline or 3D Solid as a secondary step.

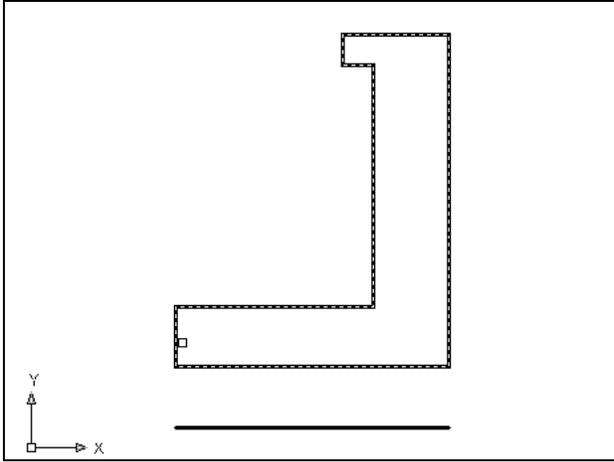


Figure 12.8 – Pick the First Edge

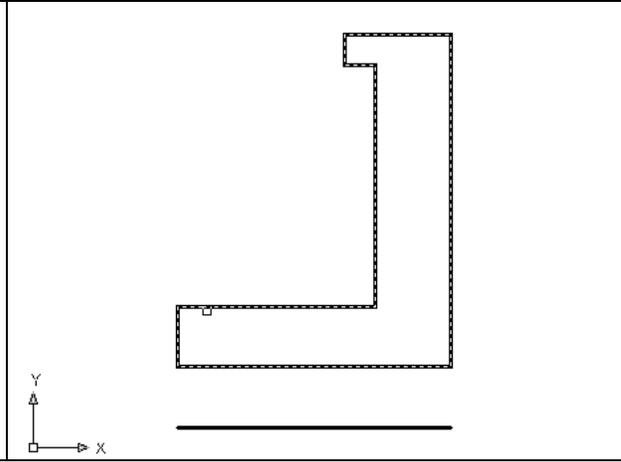


Figure 12.9 – Select the Second Edge

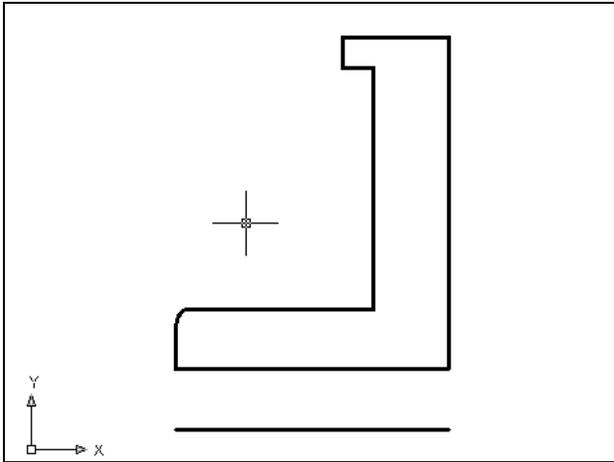


Figure 12.10 – 0.125 Arc on the Polyline

Repeat the Fillet command by pressing **Enter** on the keyboard. The next arc is just to the right of the first and the radius is still set, so select the horizontal line and then the vertical line and another arc will appear as shown in Figure 12.11. The third curved segment as shown in Figure 12.12 is above the second, so continue to repeat the command and adding fillets. You may want to zoom closer to make the selections for the fourth arc as shown in Figure 12.13 and the fifth fillet as shown in Figure 12.14

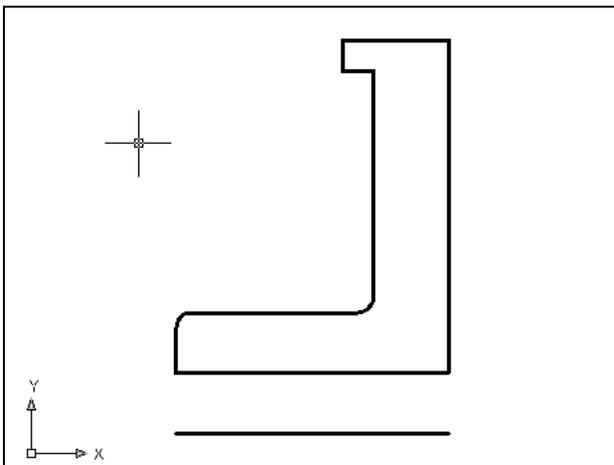


Figure 12.11 – The Second Fillet

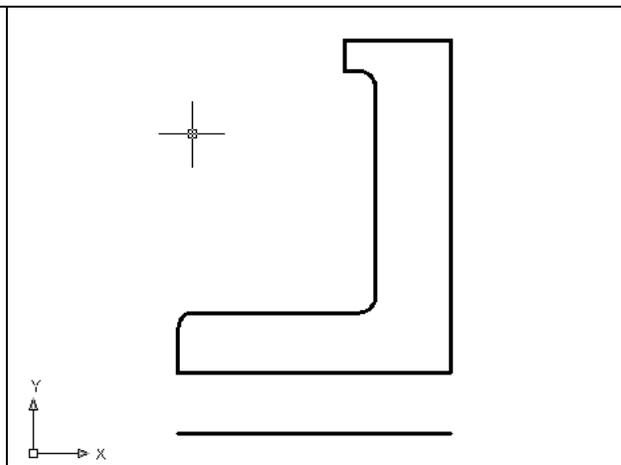
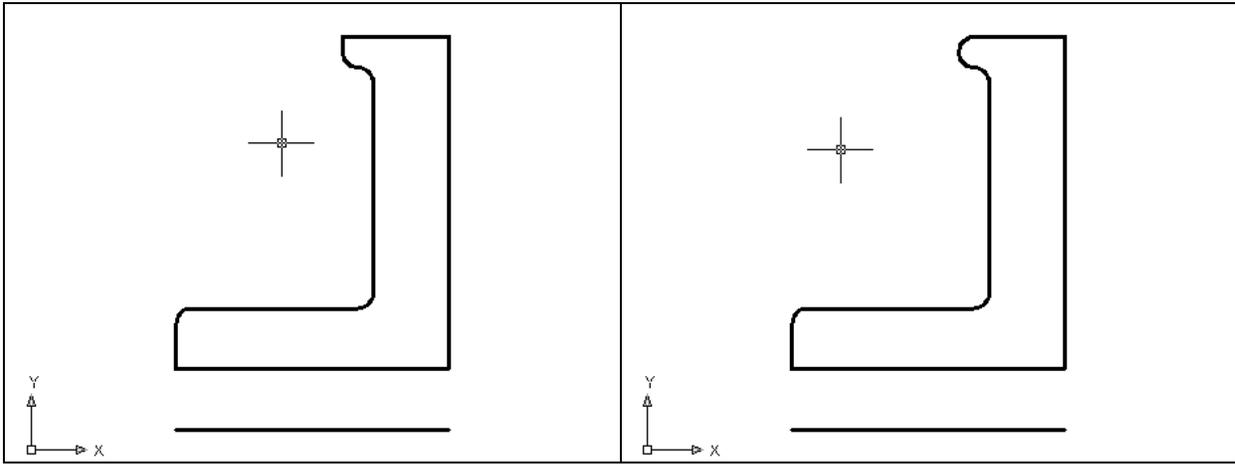


Figure 12.12 – The Third Fillet

Remember that you can use the Fillet command in 2D or 3D mode. When you apply the tool in the 2D mode, you must select two lines to describe the radius. When you are utilizing the

tool in the 3D manner, you only select the edge, which is represented by a single tessellation line. Either method will result in the same result. Because of the many commands in the AutoCAD program, you can accomplish similar tasks using different approaches.



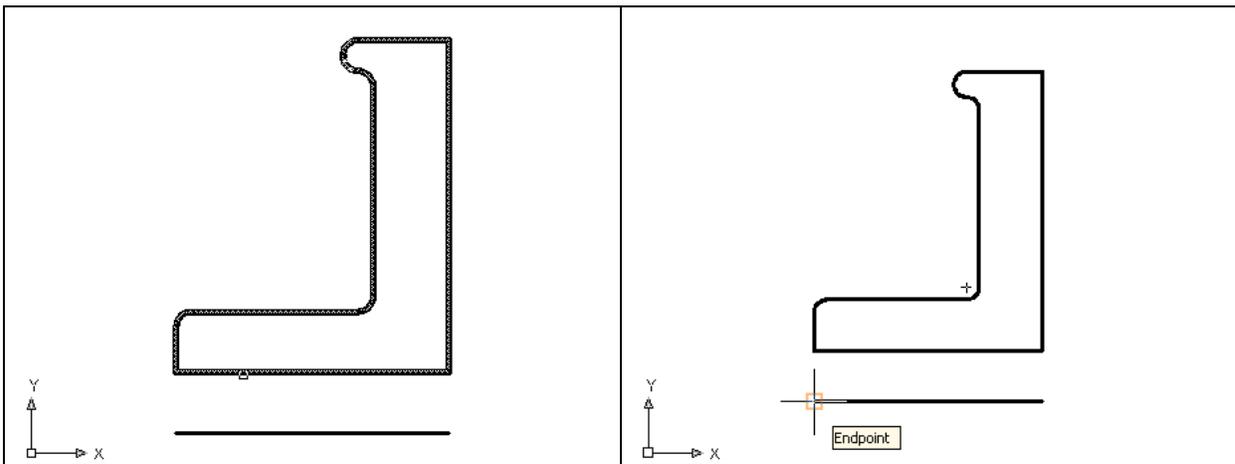
**Figure 12.13 – The Fourth Fillet**

**Figure 12.14 – The Fifth Fillet**

## Revolving a Solid from a Closed Polyline

Once you add the last fillet, 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 objects”. Pick the perimeter of the Polyline and the command line will return with “1 found” as shown in Figure 12.15. Next, you choice two points to describe the axis of rotation. Pick the Endpoint of the centerline as shown in Figure 12.16 for the first point. Pick the Endpoint of the centerline as shown in Figure 12.17 for the second point.



**Figure 12.15 – Select the Polyline**

**Figure 12.16 – The First Point of the Axis**

You will then be prompted to “Specify angle of revolution or [SStart angle] <360>:”. Hit

enter to rotate the part 360 degrees. After defining the angle of revolution, a new master solid will appear on the graphical display. On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane and the vertical Z-axis will rise upward as shown in Figure 12.19.

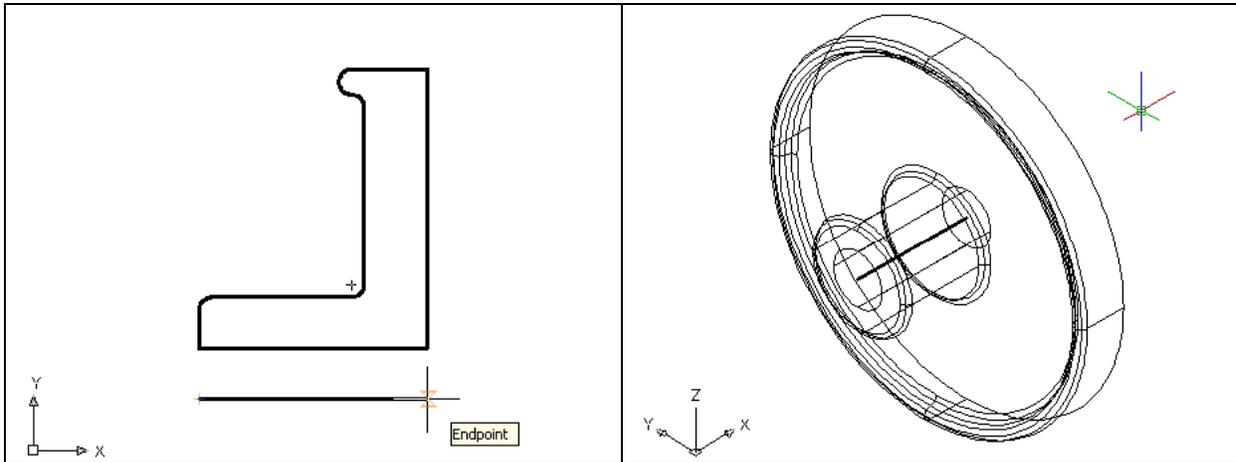


Figure 12.17 – Second Point of the Axis

Figure 12.18 – The Revolved Solid

## Rotating a 3D Solid Using the Rotate3D Command

Select the Modify on the Menu Bar, then 3D Operations and pick Rotate 3D from the list of four commands. 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 back quadrant of the part as your point of rotation. You are going to rotate the solid part on the y-axis, so pick any green diameter from the globe. Type **-90** to rotate the solid as shown in Figure 12.20.

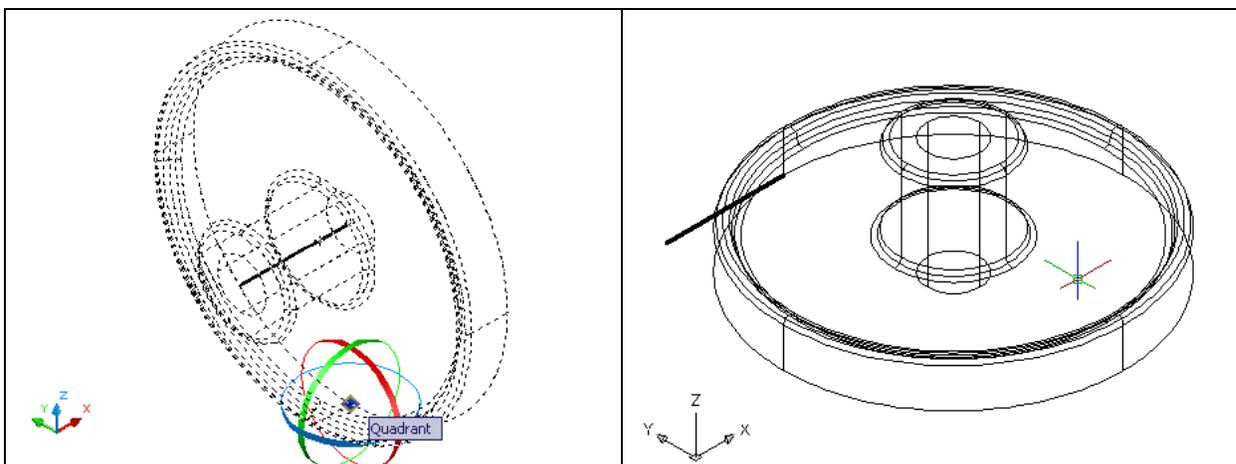
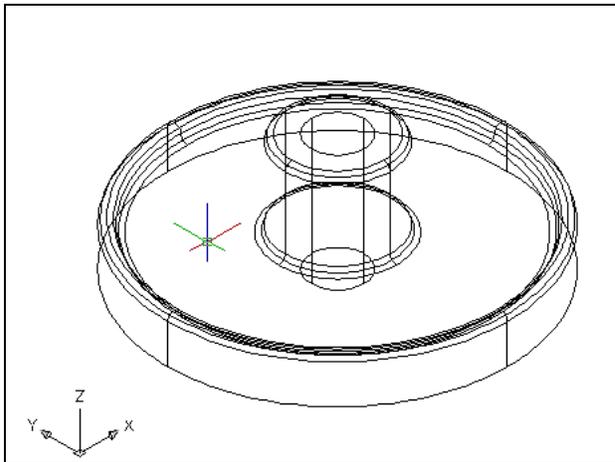


Figure 12.19 – The Rotation Axis

Figure 12.20 – The Rotated the Solid

## Removing the Center Line Using the Erase Command

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



Although you can pick what side of the solid you wish to retain in the Slice function, you will find the new command easier to add to your toolbox if you just slice the solid and leave both sides present. Use the Erase tool on the Modify toolbar to remove the centerline that you no longer need. Select the Erase tool on the Modify toolbar, and select the entity. **Enter** and this will remove the line as shown in Figure 12.21.

Figure 12.21 – Erase the Center Line

## Drawing a Solid Cylinder

You need to add three holes to the master solid: one 0.5 diameter hole going completely through the master solid, and a second and third 0.5 diameter hole that will be a copy of the first after you perform a polar array.

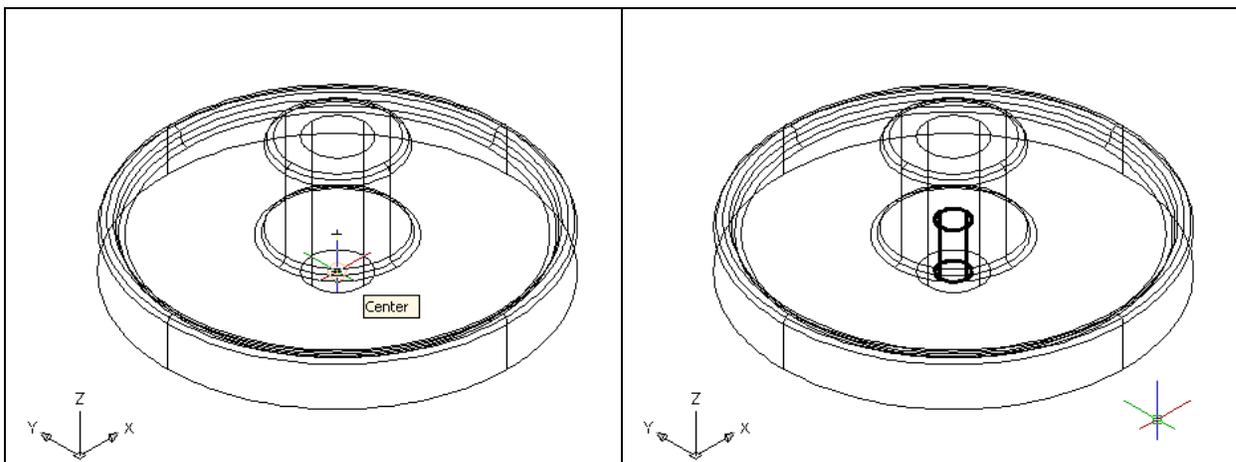


Figure 12.22 – Center of a Solid Cylinder      Figure 12.23 – The Solid Cylinder

Pick the Cylinder tool on the Solids toolbar. Specify the center point for the bottom of the cylinder as the Center of the master solid as shown in Figure 12.22. You need to type “D” for diameter and **Enter** to input the sketch’s measurement of **0.5**. **Enter** to the next prompt, which is “specify the height of the cylinder”. Type **0.875** for the elevation and hit **Enter**, and

the cylinder appears in the graphical display growing in the positive direction as shown in Figure 12.23.

## 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.

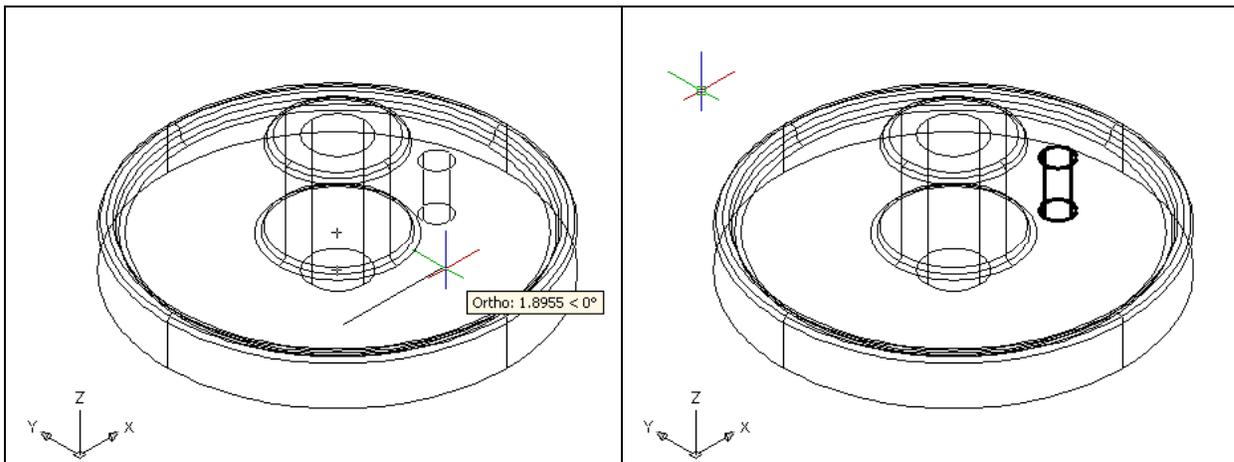


Figure 12.24 – Moving the Cylinder

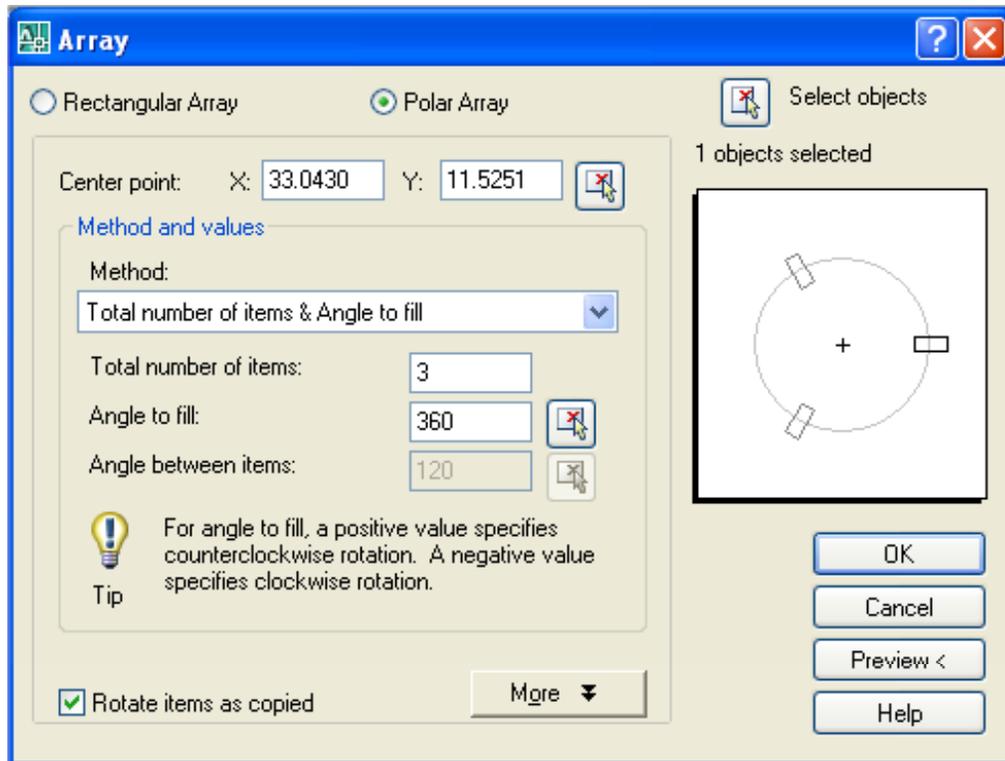
Figure 12.25 –The Cylinder in Position

To move the cylinder, you select the Move tool on the Modify toolbar. 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 12.24. For the “second point of displacement”, you need to move the cursor to the right with the Ortho “On”, type **2.0** and **Enter**. The cylinder will be ready to polar array as shown in Figure 12.25.

## Using the Array Command to Create Circular Patterns

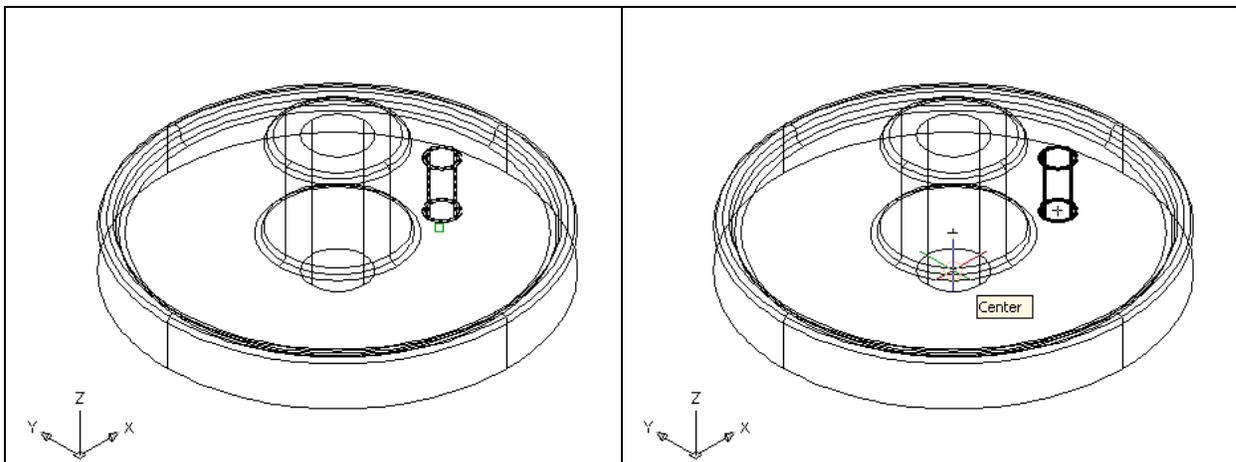
You now need to copy and rotate the solid cylinder, spacing each one 120 degrees apart. Use the Array function with the polar option as the tool for this operation.





**Figure 12.26 – The Array Window**

Select the Array tool on the Modify toolbar, which will cause the Array window as shown in Figure 12.26 to appear. Pick the Polar radial button, then the Select Objects button that will allow you to select the 0.50 diameter cylinder as shown in Figure 12.27 and hit **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 12.28.

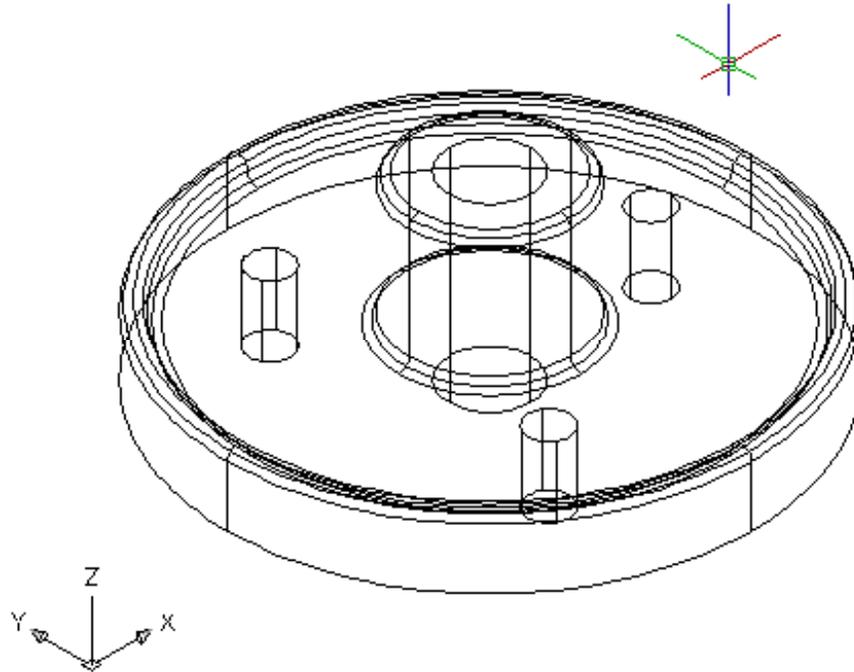


**Figure 12.27 –Selecting Objects**

**Figure 12.28 – Center of the Polar Array**

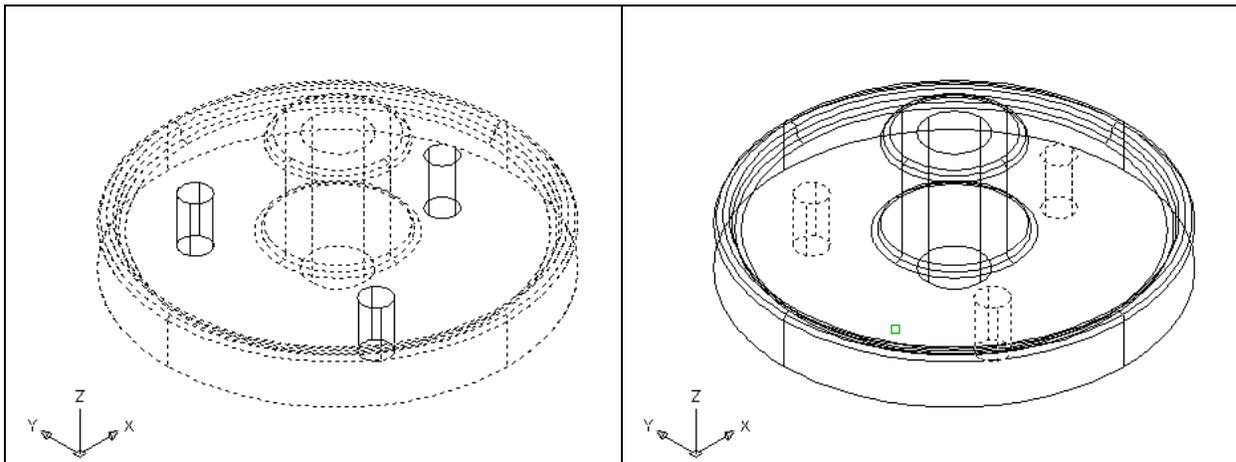
Pick the center of the part by selecting the Center of the master solid that is the heart of the part. Change the total number textbox to “3” and select the Preview button. The array will be shown on the display screen, along with an Array Accept, Modify or Cancel window. If the

Circular problem looks like Figure 12.29, press the Accept button and the array will be completed. If the Circular problem does not look like Figure 12.29, press the Modify button repeat the array process.



**Figure 12.29 – Subtracting Solids**  
**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 12.30 – Subtract Solids**

**Figure 12.31 – Subtracting Solids**

Then the AutoCAD program will prompt you to “Select solids and regions to subtract from”.

Select the left master solid as shown in Figure 12.30 and the system will respond with “1 found”. Hit **Enter** to proceed to the next step, which is “Select solids and regions to subtract”. Pick the three cylinders as shown in Figure 12.31 and the software application will counter with “3 found”. Hit **Enter** and a region exactly the shape of the three cylinders will disappear from the larger one.

## Add a Beveled Edge Using the Chamfer Command

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To add a beveled edge to a corner, you can use the Chamfer command to create the feature. In many parts, adding a broken edge will help the worker during assembly.

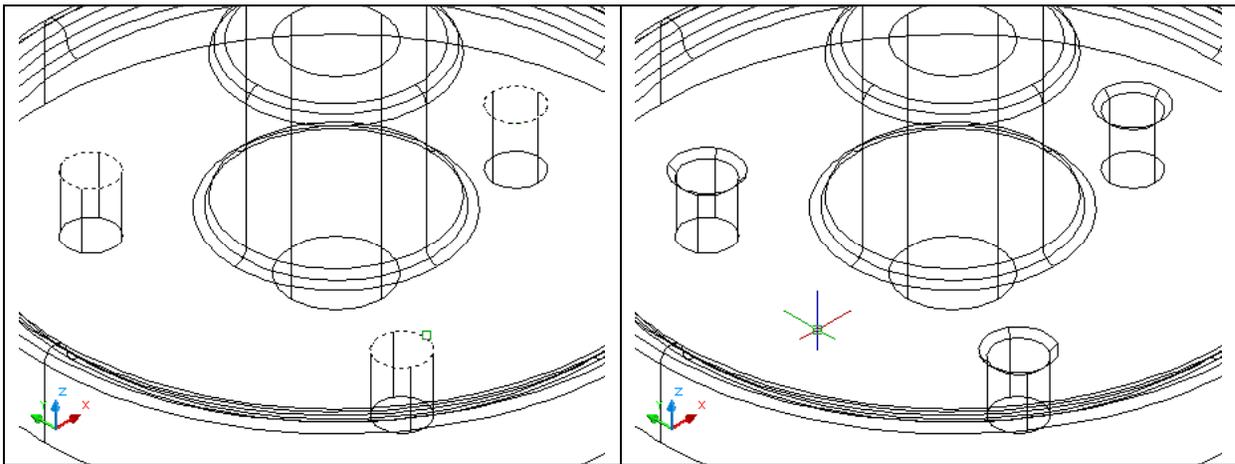


Figure 12.32 – Selecting Edges

Figure 12.33 – Three Chamfered Edges

Pick the Chamfer tool on the Modify toolbar and select the top tessellation line on the edge of a 0.5 cylinder. Hit **enter**, and then type “**0.0625**” for the offset distance. Hit **Enter** and type “**0.0625**” for the second offset distance. Hit **Enter**, and then select the top edge of each 0.5 cylinder. Hit **Enter** and the 0.0625 by 45-degree chamfer will appear on the edge of the each hole as shown in Figure 12.33.

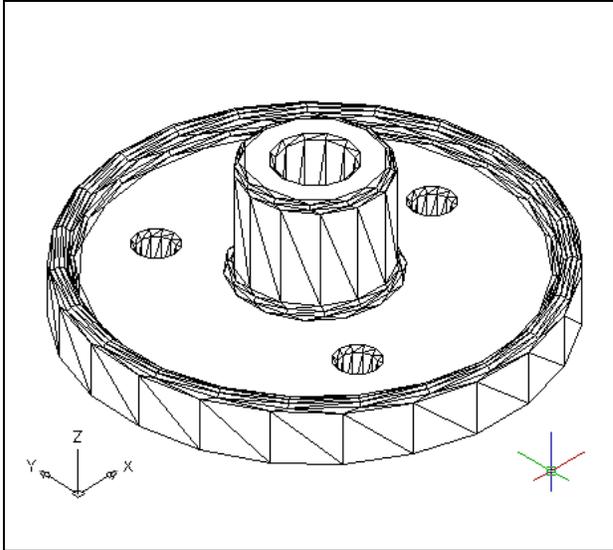
Repeat the Chamfer command by pressing **Enter** on the keyboard. Select the top tessellation line on the inner edge of the center hole. **Enter** and the 0.0625 by 45-degree chamfer will appear on the edge of the each hole as shown in Figure 12.34.

## Hide the 3D Solid

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In Figure 12.33, you will notice that the tessellation lines representing the solid’s outline on the far edge of the part are visible. In a real world solid part, you would not be able to see through solid unless the material was transparent. You can use the Hide command to remove the hidden lines from your graphical display.





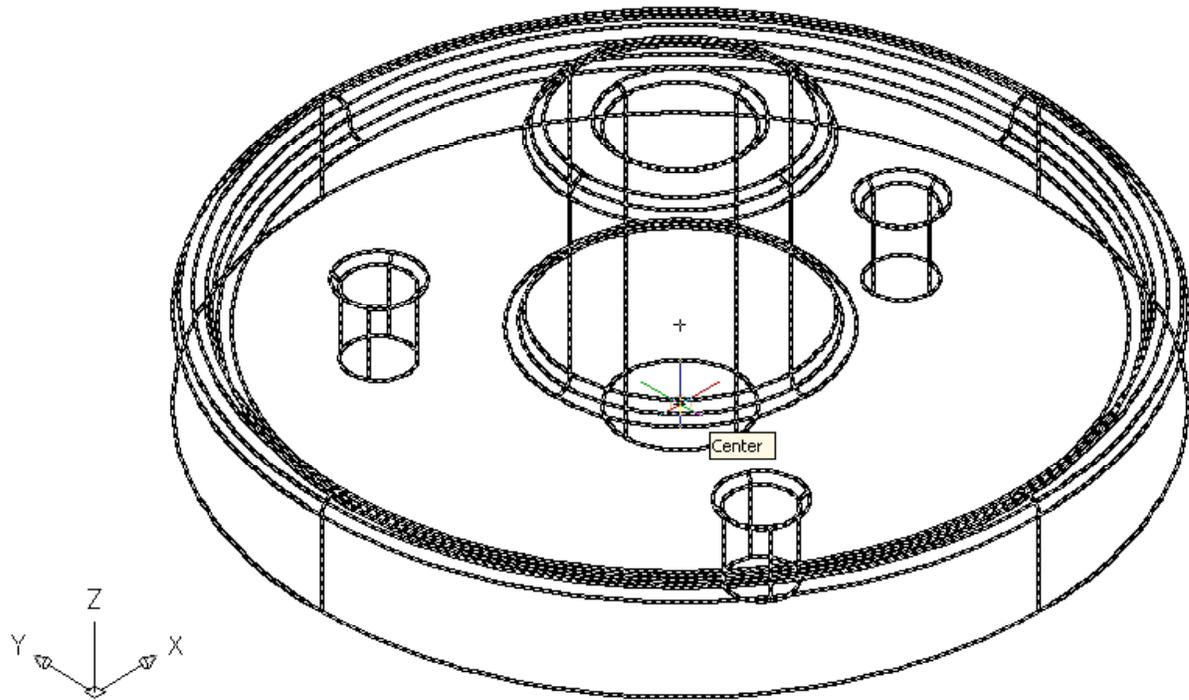
Select the Hide tool on the Render toolbar. This is a very simple command, because moments after picking the tool, the tessellation lines representing the back of the part are removed from view. If you wish to return the graphical display to the previous setup where all the tessellation lines are seen, then choose the second tool on the Visual Styles toolbar, called 3D Wireframe. By choosing the 3D wireframe tool, the solid part will return to the previous appearance. This command does not alter the mass of the solid or any of the dimensions, but is available for the computer aided designer as a viewing option.

Figure 12.34 – Hide the Solid

## Moving the Solid to the Origin Point

To move revolving problem one to the drawing origin of 0,0,0 on the x, y and z-axis, pick the Move tool on the Modify toolbar and select the 3D solid and Enter. At the prompt, “**specify base point or displacement**”, select the endpoint shown in Figure 12.35 at the bottom center of the revolving problem one, which will become the problem one’s insertion point into an assembly drawing. At the “second point of displacement”, type **0,0,0** and **Enter**. Problem one 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.

**\* World Class CAD Challenge 03-11 \* - 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.**

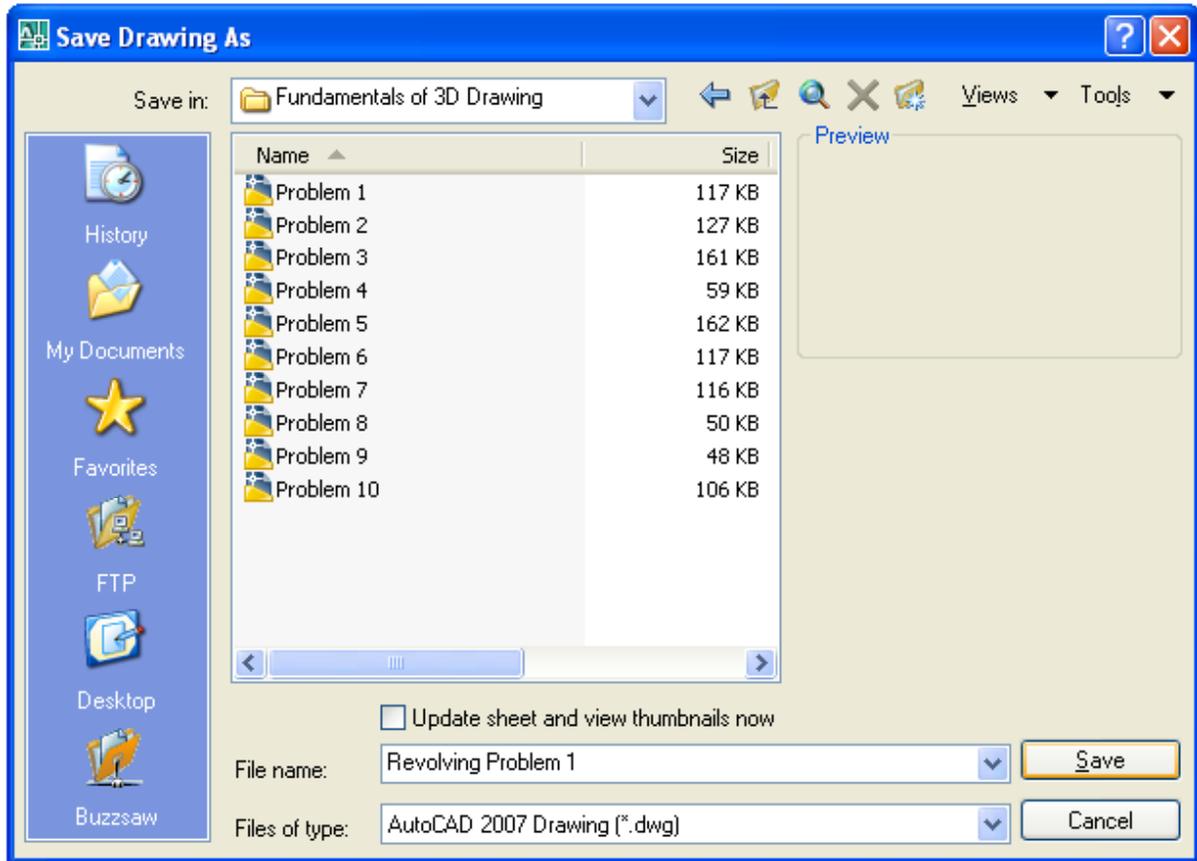


**Figure 12.35 – Moving the Finished Solid to the Origin**

## **Saving the Solid Problem**

To save Revolving Problem 1 in your Fundamentals of 3D Drawing Folder, select the Save tool on the Standard toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. At the File Name textbox, type “Revolving Problem 1” and press the Save button to save the drawing. (See Figure 12.36)





**Figure 12.36 – Saving Revolving Problem One**

**\* 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.**