Chapter 12

Revolving Problem One

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. **Drawing a Center Line Using the Line Command**
5. **Creating a Perimeter Using the Polyline Command**
6. **Adding Curved Edges Using the Fillet Command**
7. **Revolving a Solid from a Closed Polyline**
8. **Rotating a 3D Solid Using the Rotate3D Command**
9. **Drawing a Solid Cylinder**
10. **Moving a Solid into Position on the Master Solid**
11. **Creating and Placing Solids Using the Array Command**
12. **Subtracting 3D Solids from the Master Solid**
13. **Shading the Solid**
14. **Moving the Solid to the Origin Point**
15. **Saving the Solid Problem**

Sketch of Revolving Problem One

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 revolution problem one, the master solid contains a rounded edge in the main shapes that you will create with the Fillet tool. After revolving the Polyline perimeter, you will place a cylinder onto that main solid, copy and arrange new cylinders using the Array tool, and then 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.

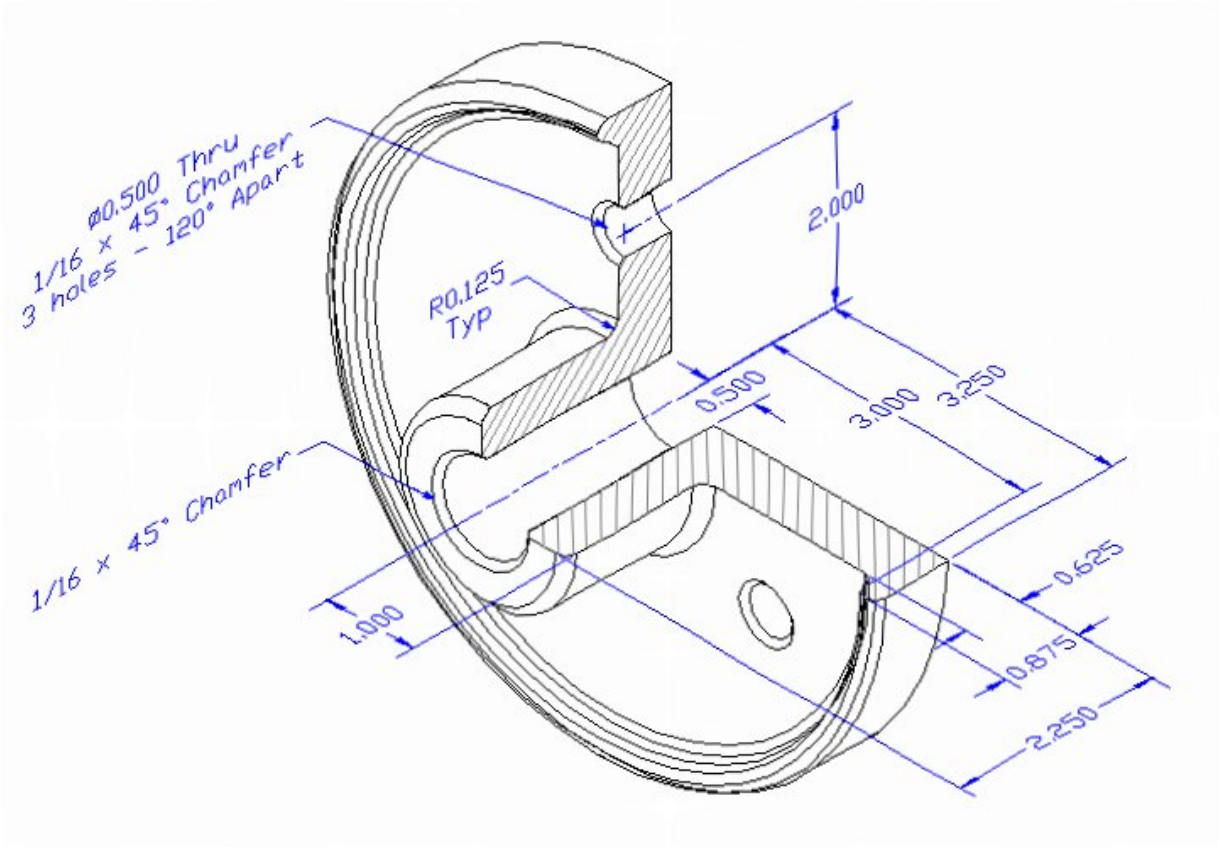


Figure 12.1 – Revolving Problem One Sketch

Remember, when you receive a sketch from a professional, you need to practice 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, with a few features left to finish the exercise. This problem will help you address solid parts containing different 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

In Revolving Problem 1, 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 12.2). 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.

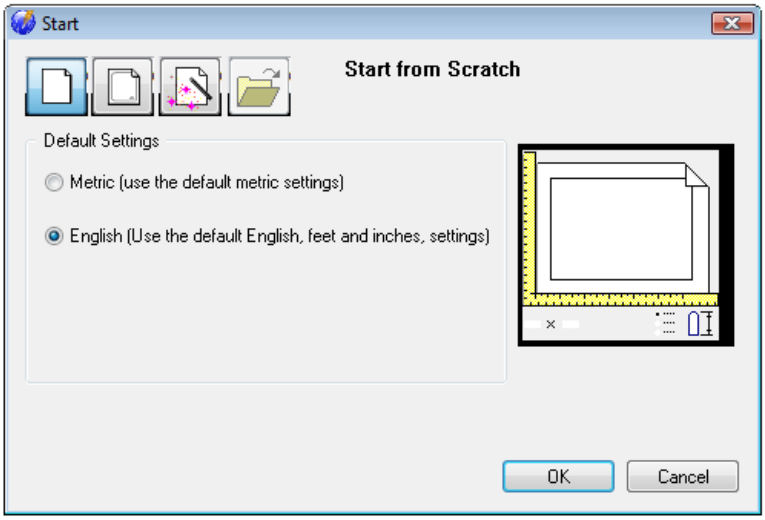
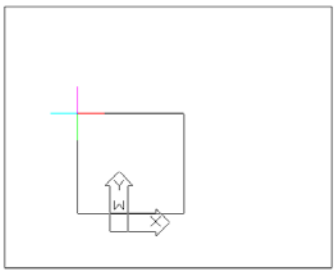


Figure 12.2 – Starting the Drawing

Modifying How the UCS Icon is Displayed



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 addition 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 12.3 - 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.

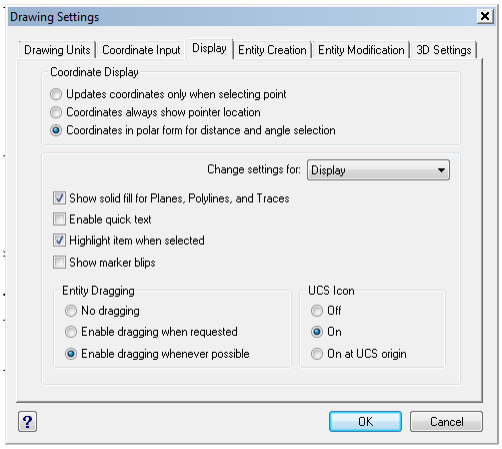
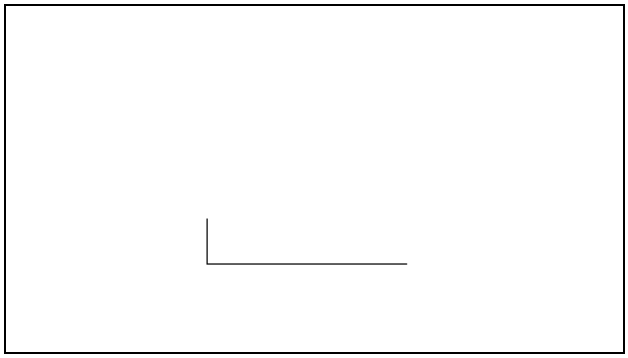


Figure 12.4 - Drawing Settings window, UCS Icon

Creating a Center Line Using the Line Command

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



With the Ortho mode on, move the line on the graphic screen to the left and type "**2.25**" and **ENTER** at the keyboard to complete the function. Next drag the line up on the screen; type 0.5 and ENTER to create a construction line as shown in Figure 12.5. This line will allow you to place the polyline in the correct position in the next step of the problem. Hit Escape to exit the line command.

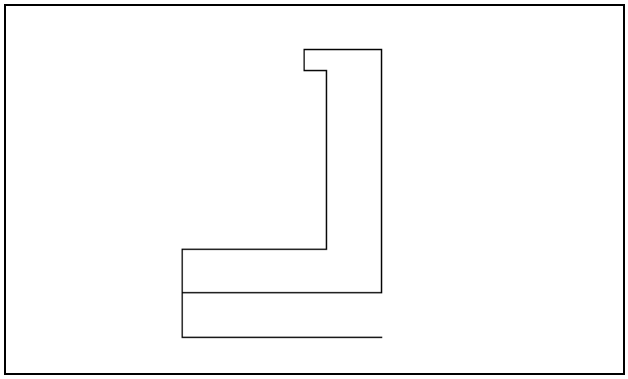
Figure 12.5 – The Center and Construction Lines

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. Use the Endpoint Esnap of the second construction line you drew as the starting point of the polyline. The command line information is shown in Figure 12.6.

Command : _POLYLINE
ENTER to use last point/Follow/<Start of polyline>:
Arc/Distance/Follow/Halfwidth/Width/<Next point>: **2.25**
Arc/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **2.75**
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **.875**
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **.25**
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **.25**
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **2**
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **1.625**
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **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. A closed entity will appear in the graphical display as shown in Figure 12.7. Zoom Extents 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 rounded 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 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. 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 included in the Polyline tool, but you would have to calculate the length of the lines by subtracting the radius. Like most designers, you will come to trust the software with mathematics whenever possible; therefore, you will place chamfers and fillets on the 2D Polyline or 3D Solid as a secondary step.

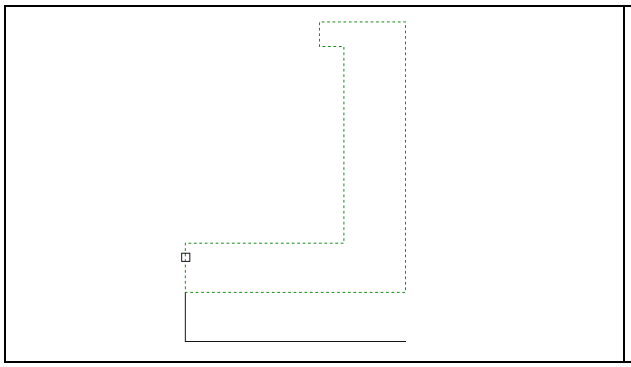


Figure 12.8 – Pick the First Edge

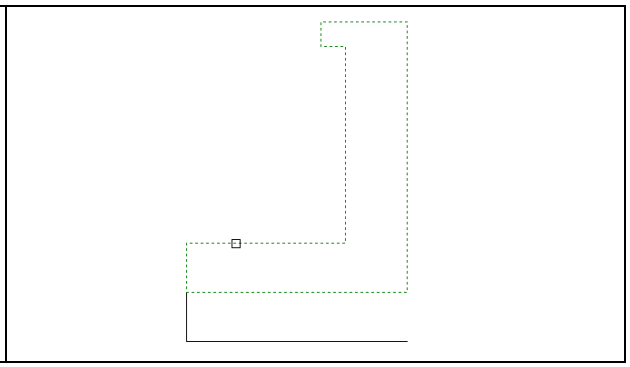


Figure 12.9 – Select the Second Edge

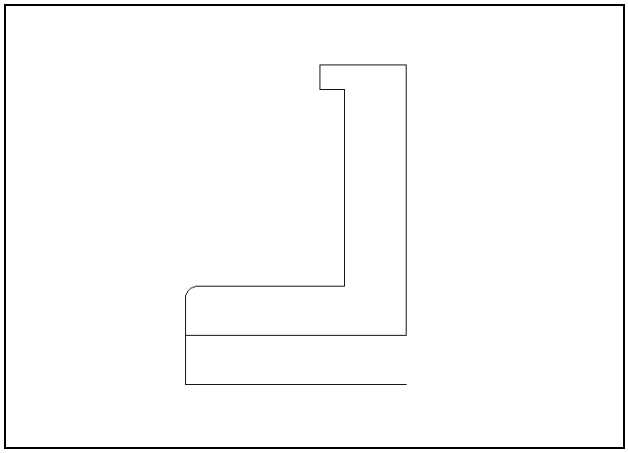


Figure 12.10 – 0.125 Arc on the Polyline

Repeat the Fillet command by hitting **ENTER**. The next arc is just to the right of the first. 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 by 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

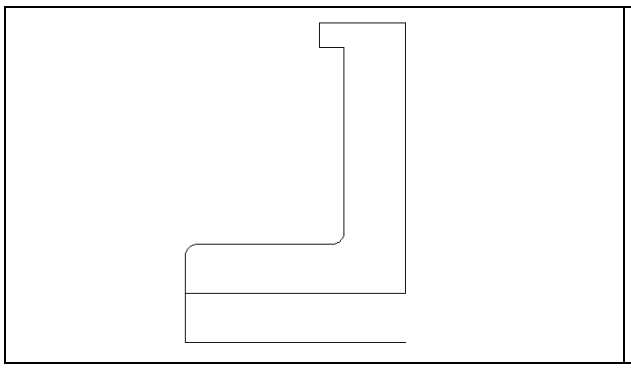


Figure 12.11 – The Second Fillet

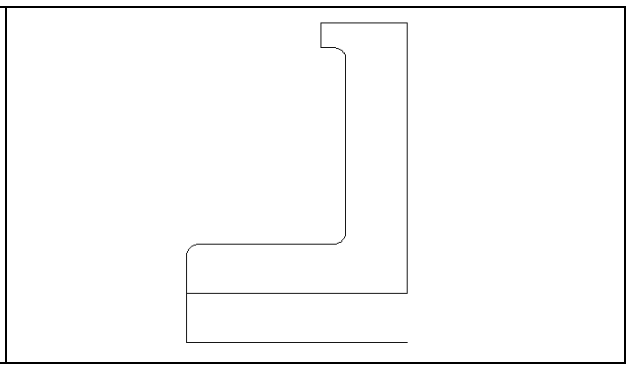


Figure 12.12 – The Third Fillet

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 3D, you only have to select an edge, which is represented by a single tessellation line. Either method will result in the same fillet. Because of the many commands in progeCAD, you can accomplish similar tasks using different approaches.

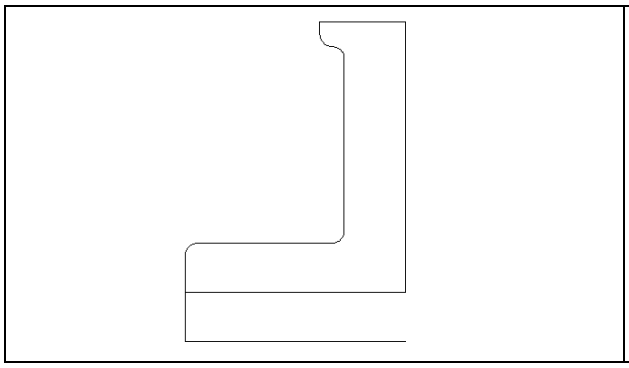


Figure 12.13 – The Fourth Fillet

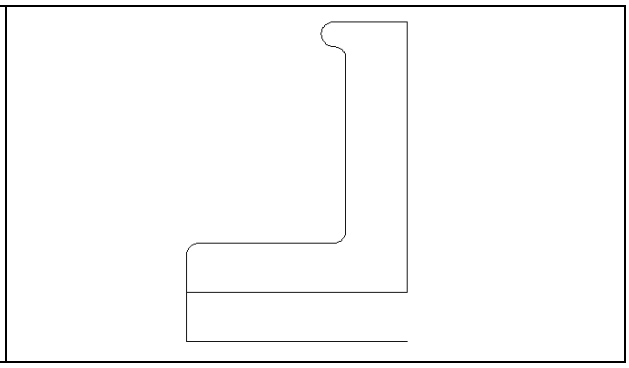


Figure 12.14 – The Fifth Fillet

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. Erase the vertical construction line before you continue.

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 12.15. Next, 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.

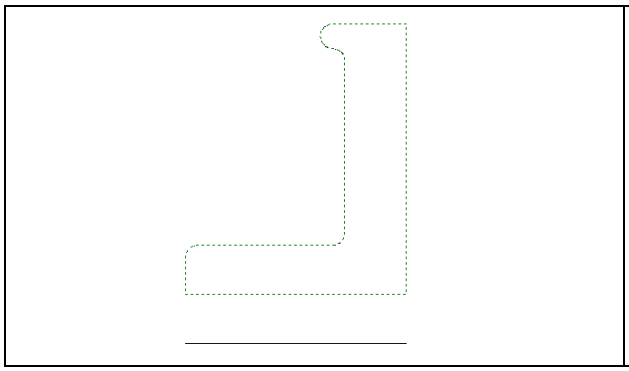


Figure 12.15 – Select the Polyline

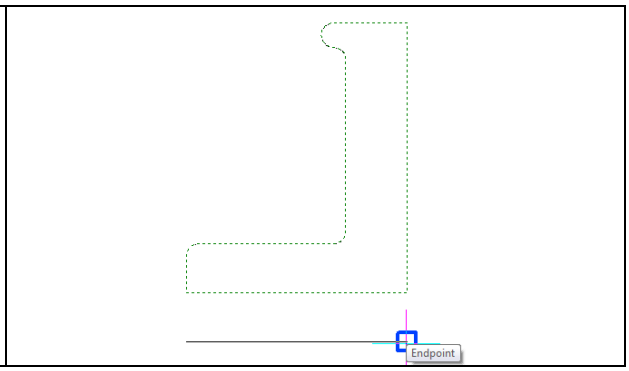


Figure 12.16 – The First Point of the Axis

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 12.19.

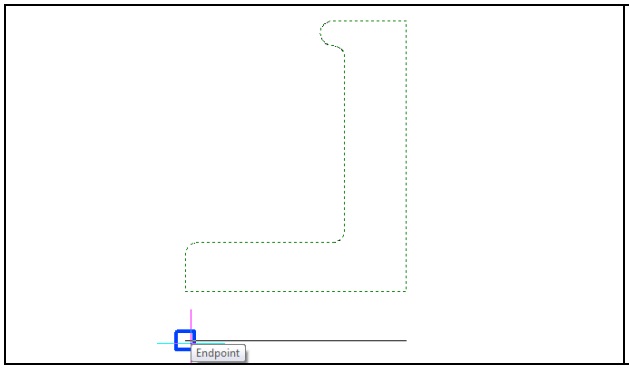


Figure 12.17 – Second Point of the Axis

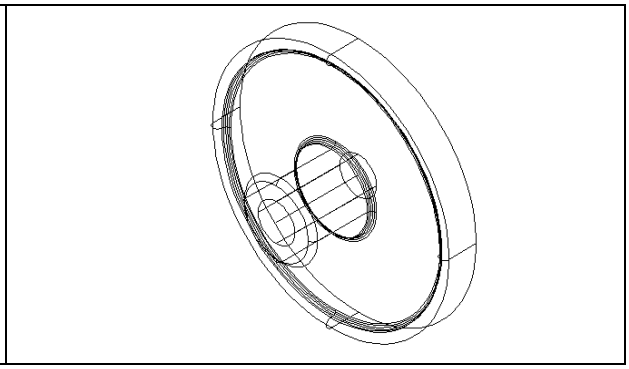


Figure 12.18 – The Revolved Solid

Rotating a 3D Solid Using the Rotate3D Command

Select 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 12.19. Type **90** and **ENTER** to rotate the solid as shown in Figure 12.20. You can also erase the Centerline that did not get rotated.

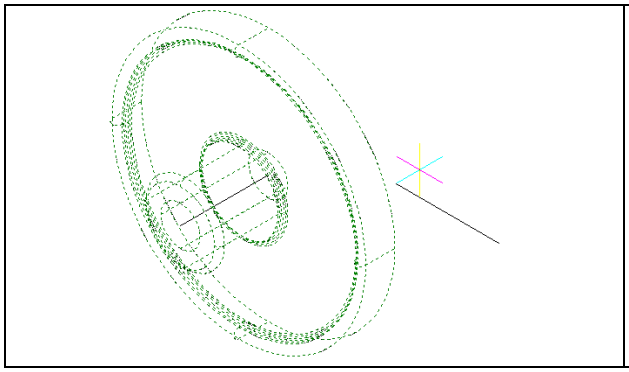


Figure 12.19 – The Rotation Axis

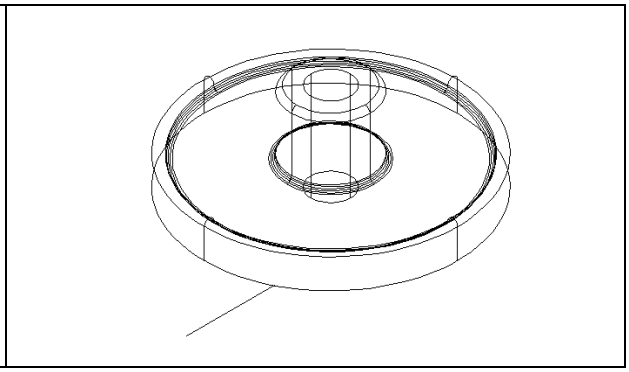


Figure 12.20 – The Rotated the Solid

Drawing a Solid Cylinder

You need to add three holes to the master solid. You will draw the first cylinder in the middle of the part. After moving it to a new location, you will use the Array command to place the other two holes.

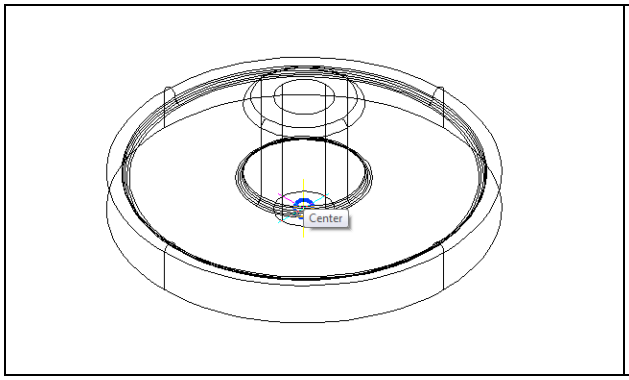


Figure 12.22 – Center of a Solid Cylinder

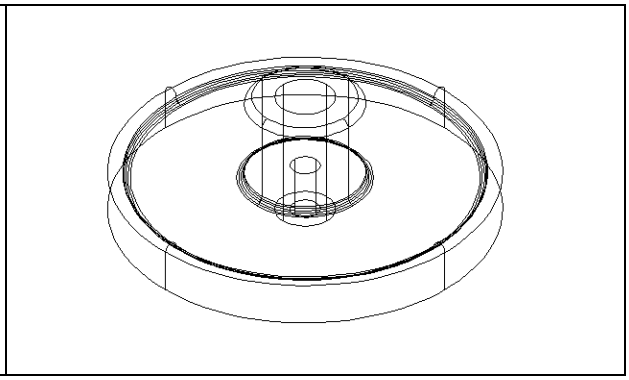


Figure 12.23 – The Solid Cylinder

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. Type "**D"** for diameter and **ENTER** to input the sketch's measurement of **0.5**. Hit **ENTER** to advance to the next prompt, which is "Center of second end/<Height of cylinder>:". Key in **0.875** for the height of the cylinder and hit **ENTER,** and the cylinder will in the graphical display 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 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.

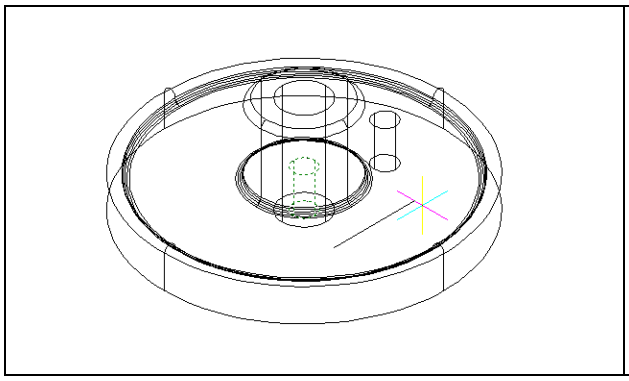


Figure 12.24 – Moving the Cylinder

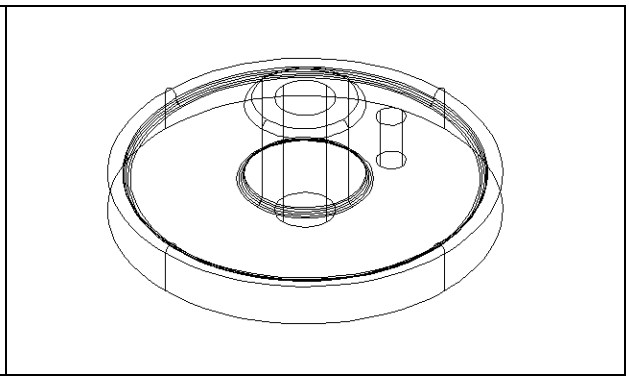


Figure 12.25 –The Cylinder in Position

Once you choose the Move command, you will be prompted to "Select entities to move:". Pick the cylinder 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 to the right and type **2** and **ENTER** at the command line (Figure 12.25). The cylinder will move 2 units in the direction of your cursor.

Using the Array Command to Create Circular Patterns

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

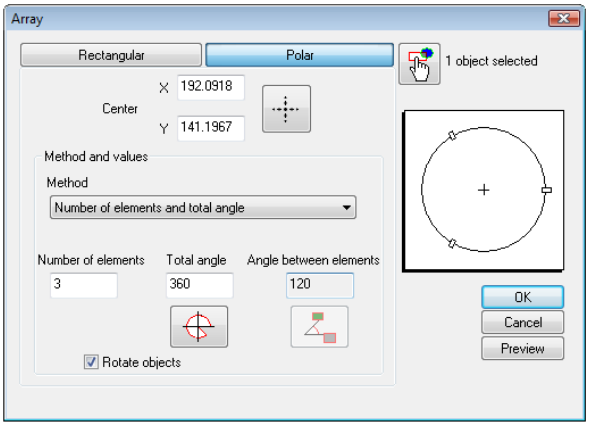


Figure 12.26 – The Array Window

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

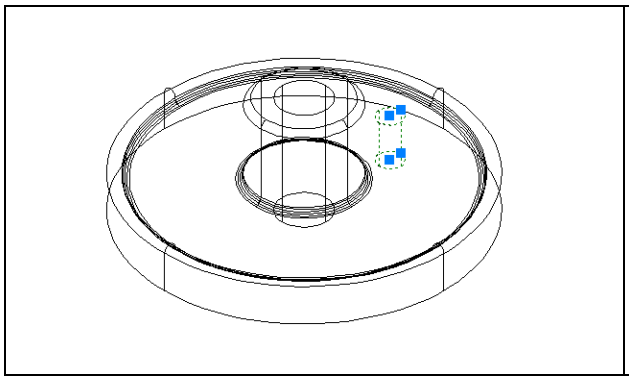


Figure 12.27 –Select the Cylinder

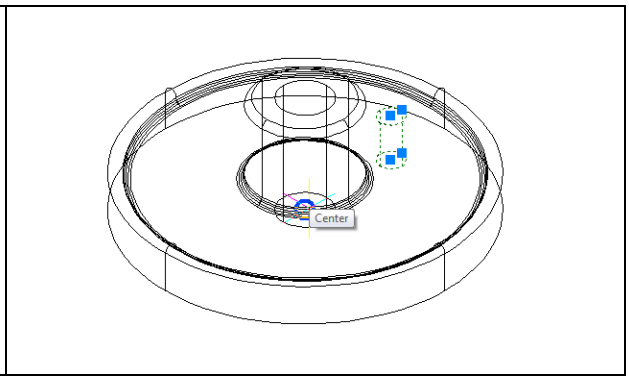


Figure 12.28 – Center of the Polar Array

Change the “Number of Elements” textbox to “3”. 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 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.

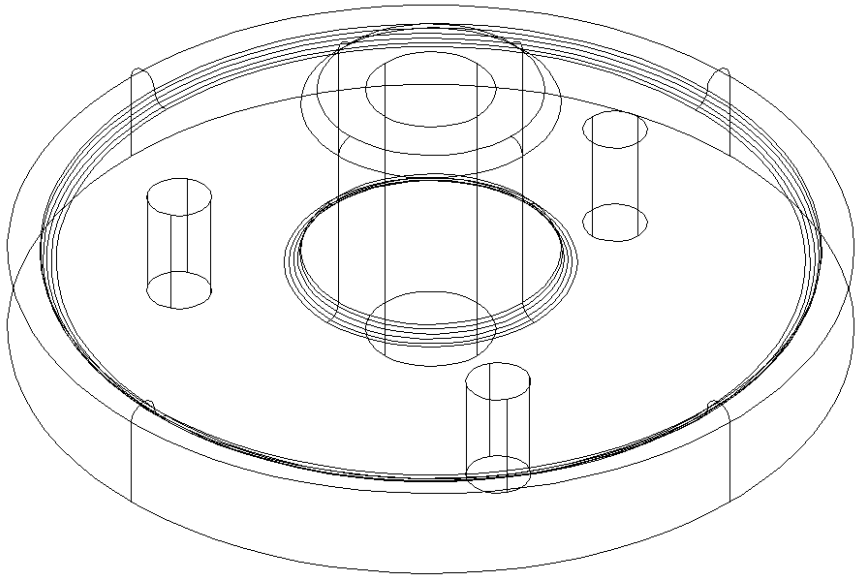


Figure 12.29 – The Finished Array

Subtracting 3D Solids from the Master Solids

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

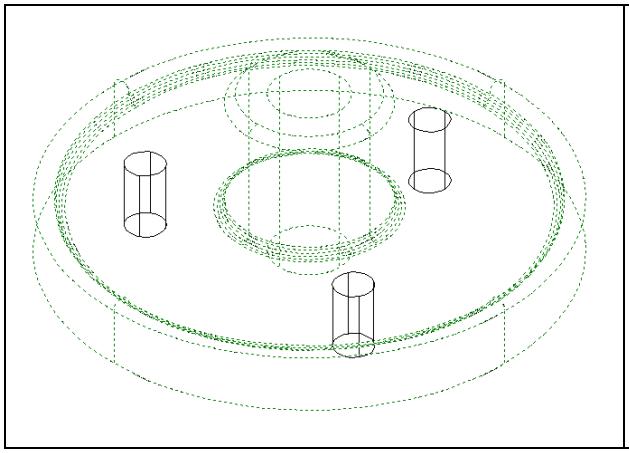


Figure 12.30 – Pick the Master Solid

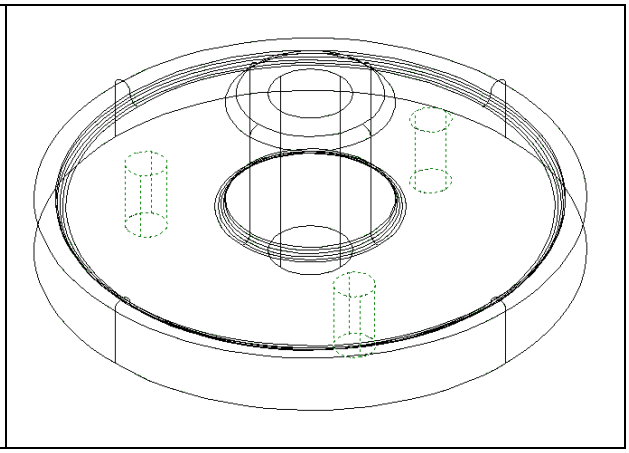


Figure 12.31 – Pick the Three Cylinders

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 12.30). Hit **ENTER** to proceed to the next step, which is to “Select ACIS objects to subtract:” (Figure 12.31). Pick the cylinders inside of the master solid and the command line will reply with “Entities in set: 3”. Hit **ENTER** and a region exactly the shape of the cylinders will be removed from the main solid.

Add a Bevel Edge Using the Chamfer Command

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.

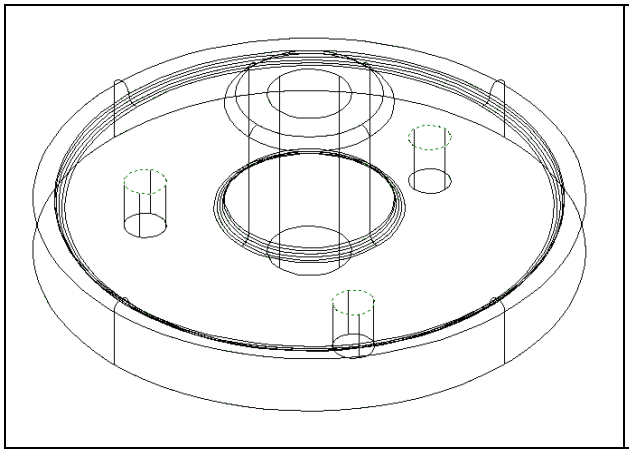


Figure 12.32 – Pick the Top of the Holes

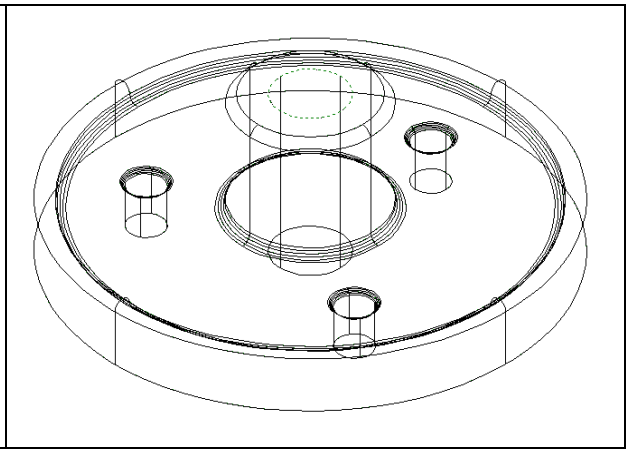


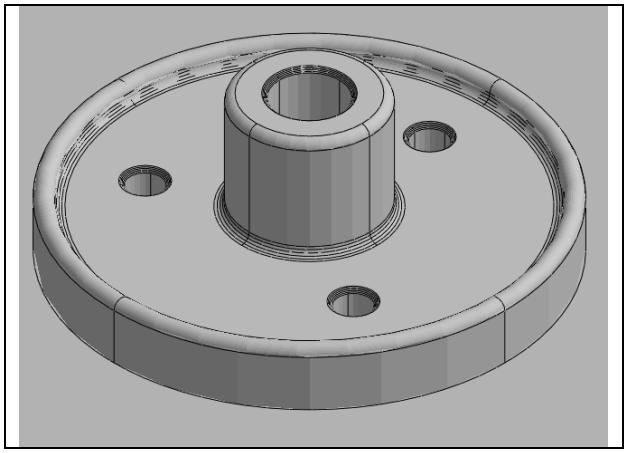
Figure 12.33 – Chamfers on Three Holes

Pick the Chamfer tool on the Modify toolbar and select the top tessellation line on the edge of a 0.5 cylinder. The command line will prompt you with “Enter surface selection option: Next/<OK (current)>:”, so just hit ENTER to continue. Type 0.0625 as the Chamfer distance of the First and Second entity. When prompted to select an edge or loop, select the top edge of all three holes. Hit ENTER and the edges of each hole will change to the chamfered edge as shown in Figure 12.33.

Repeat the Chamfer command by pressing **ENTER** on the keyboard. Select the top tessellation line of the hole in the middle of the part. Hitting **ENTER** should advance you quickly through the prompts using the previous values for chamfer distances. Select the inside edge of the middle hole when prompted to select an edge or loop, and the edge will become chamfered as shown in Figure 12.34.

Shading the 3D Solid

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 Shade Highlighting Outline command to remove the hidden lines from your graphical display.



Select the Shade Highlighting Outline tool on the Shade toolbar. If you wish to return the graphical display to the previous setup in which all the tessellation lines can be seen, choose the first tool on the Shade toolbar called 2D Wireframe. By choosing the 2D Wireframe tool, the solid part and the UCS icon 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 – Shading 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, select the 3D solid, and hit ENTER. When prompted to select a "**Vector/<Base point>:**," select the endpoint shown in Figure 12.35 at the bottom center of the revolving problem one, 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.

*** World Class CAD Challenge 101-11 * - 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.**

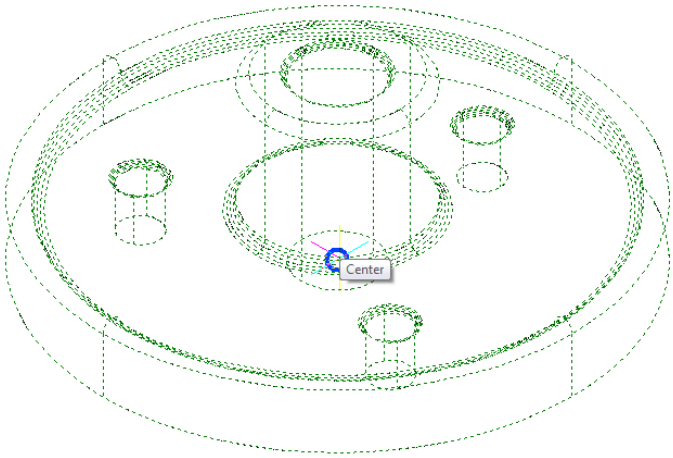


Figure 12.35 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Revolving Problem 1, 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 select the Save button to save the drawing (Figure 12.36).

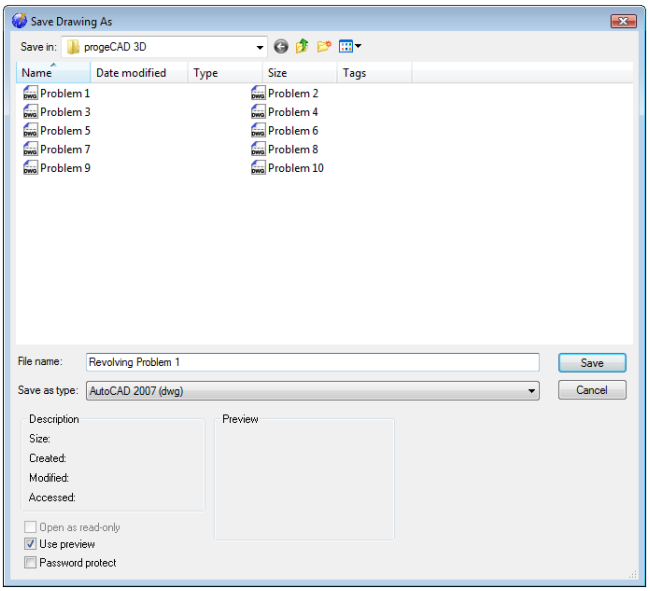


Figure 12.36 – Saving Revolving Problem One

*** World Class CAD Challenge * - Report your best times to World Class CAD at www.worldclasscad.com to obtain your world class ranking.**