Chapter 14

Revolving Problem Three

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

1. **Sketch of Revolving Problem Three**
2. **Starting a 3D Part Drawing**
3. **Modifying How the UCS Icon is Displayed**
4. **Creating Center and Perimeter Lines Using the Line Command**
5. **Creating a Joined Perimeter Using the Edit Polyline Command**
6. **Revolving a Solid from a Closed Polyline**
7. **Shading a 3D Solid**
8. **Rotating a 3D Solid Using the Rotate3D Command**
9. **Removing a Centerline Using the Erase Command**
10. **Drawing Two Cylinders**
11. **Moving a Solid into Position on the Master Solid**
12. **Copying and Arranging Solids Using the Array Command**
13. **Subtracting 3D Solids from the Master Solid**
14. **Moving the Solid to the Origin Point**
15. **Saving the Solid Problem**

Sketch of Revolving Problem Three

You will start the third revolving problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across two lines. 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 exercise in revolving problem three involves drawing just a portion of the component, and revolving the section into a solid. After revolving the Polyline perimeter into a solid, you will place two cylindrical solids onto that main solid. Using array, you will copy and arrange the detail and subtract the cylinders.

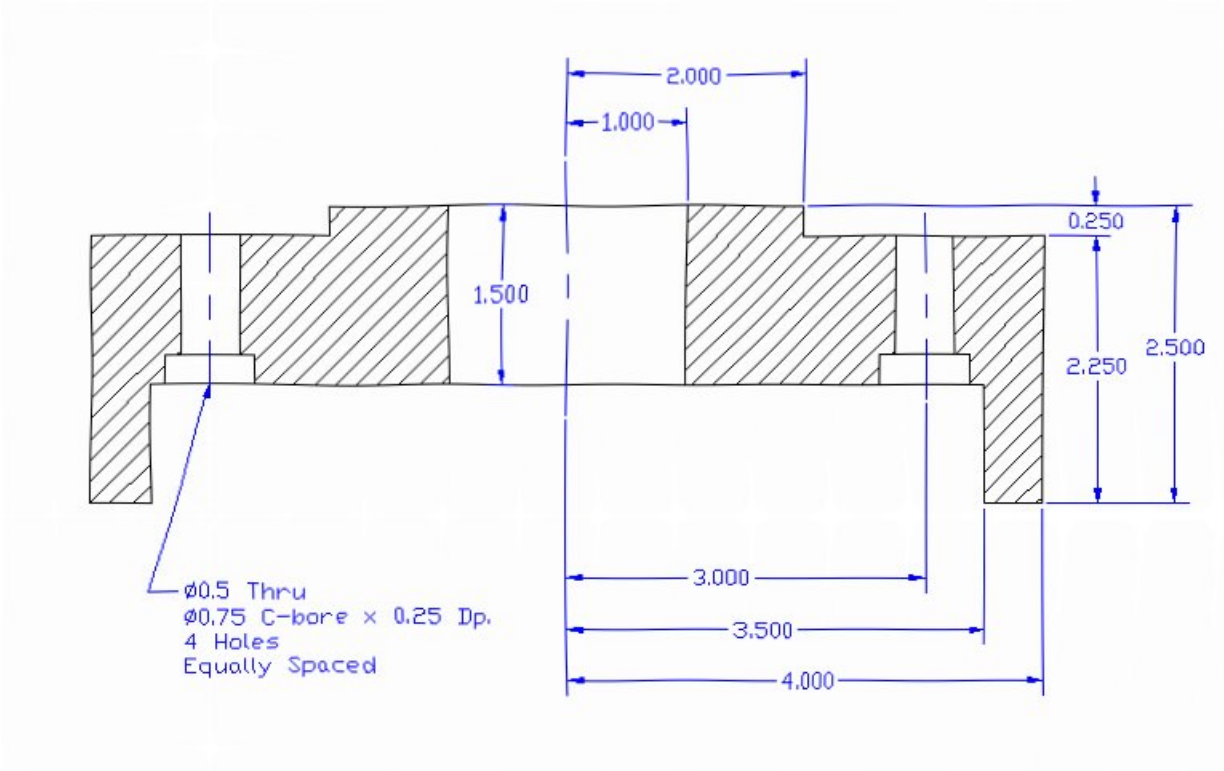


Figure 14.1 – Revolving Problem Three Sketch

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, revolving a closed Polyline is your most probable action. Place as much detail in the perimeter and interior detail to finish the master solid quickly, only leaving a few features 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 a successful in the 3D modeling.

Starting a 3D Part Drawing

In Revolving Problem 3, 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 14.2). 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.

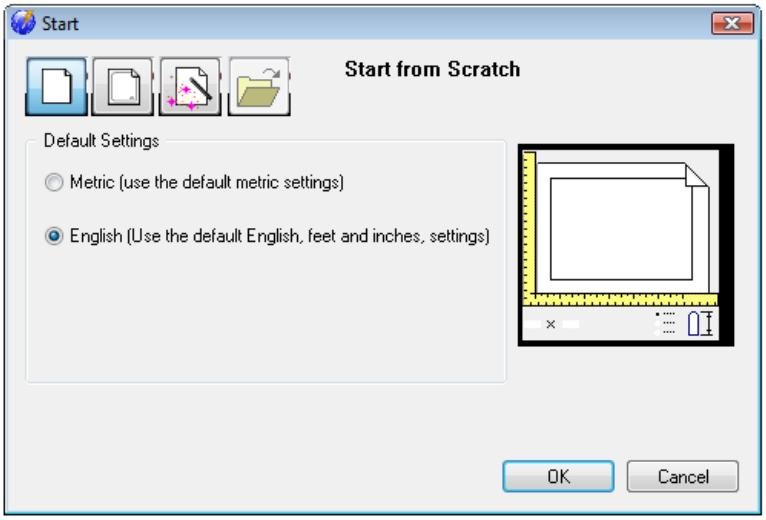
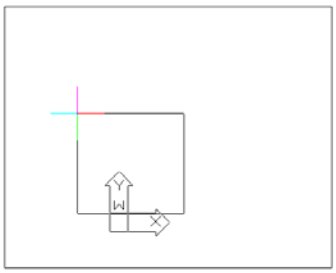


Figure 14.2 – Starting the Drawing

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 14.3 - 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.

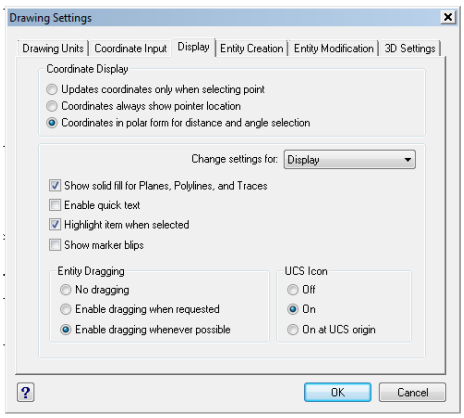
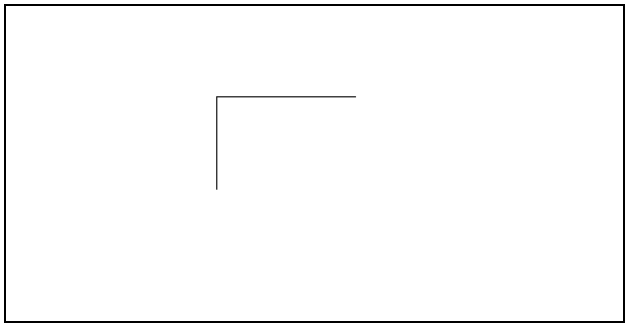


Figure 14.4 - Drawing Settings window, UCS Icon

Creating a Center and Perimeter Lines Using the Line Command

First, draw a 1.5 center line from right to left using the Line command. Pick the Line tool on the Draw toolbar to start drawing the detail perimeter of the problem.



With your mouse, select any point at the upper right side of the graphical display. Keep the Ortho mode on, pull the line to the right, type **1.5** and hit **ENTER**. Draw another line straight down **1** unit as shown in Figure 14.5.

Figure 14.5 – Draw a Center and Construction Line

Hit ENTER to repeat the Line command. Starting at the end of the vertical line you just drew, draw a **1.50** unit line to the right, a **1.0** line downward, a **0.25** line to the left, a **2.0** line down, a **2.25** line to the right, a **0.5** line upward, a **1.0** line to the right and then type "**C**" to close the line. The command line prompts are listed below in Figure 14.6.

Command: LINE Specify first point: **@0,-1.0**
Specify next point or [Undo]: **1.5**
Specify next point or [Undo]: **1.0**
Specify next point or [Close/Undo]: **0.25**
Specify next point or [Close/Undo]: **2.0**
Specify next point or [Close/Undo]: **2.25**
Specify next point or [Close/Undo]: **0.5**
Specify next point or [Close/Undo]: **1.0**
Specify next point or [Close/Undo]: **c**

Figure 14.6 – Drawing a Series of Lines

Every step in the process of drawing the detailed perimeter is shown in Figures 14.6 through 14.12. A closed entity of the final shape will appear in the graphical display as shown in Figure 14.13. Zoom Extents to have the box fill the monitor, so you can proceed to the next step in the drawing process. Erase the 1 unit vertical line you used to start the perimeter before continuing.

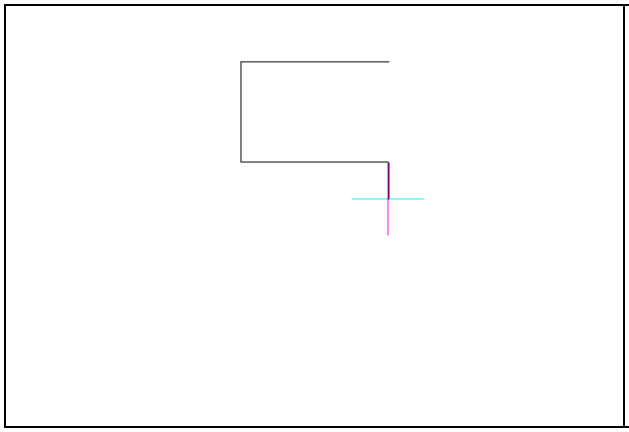


Figure 14.7 – Draw a Line 1.5 to the Right

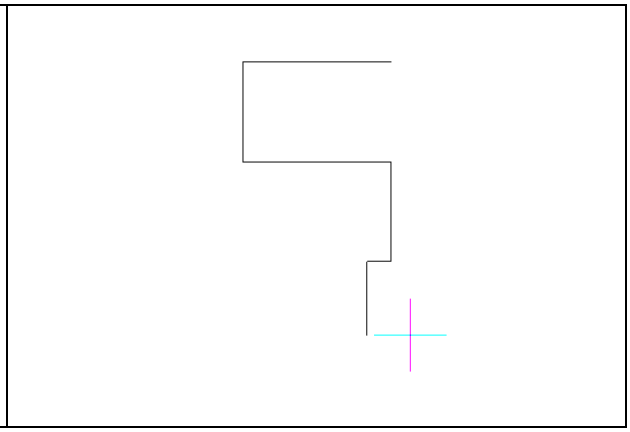


Figure 14.8 – Lines 1.0 Down and 0.25 Left

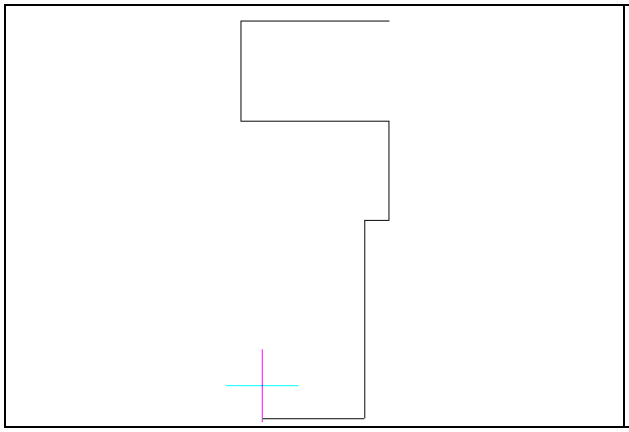


Figure 14.9 – Draw a Line 2.0 Downward

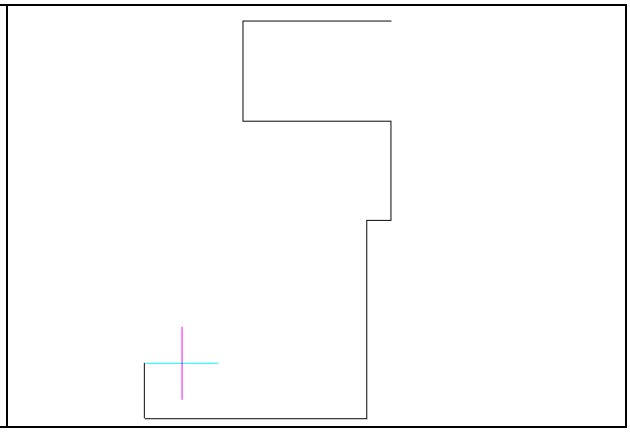


Figure 14.10 – Draw a Line 2.25 Left

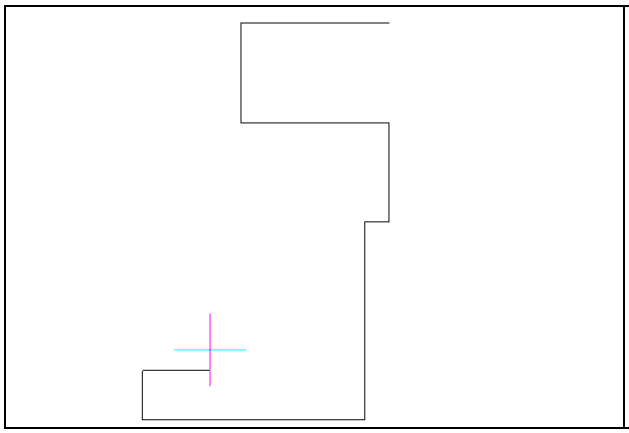


Figure 14.11 – Draw a Line 0.50 Upward

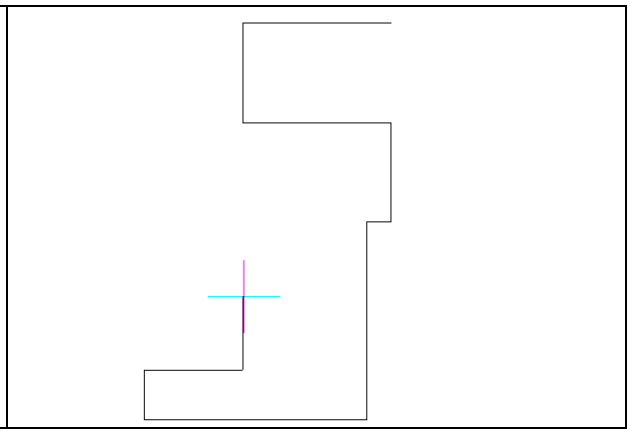
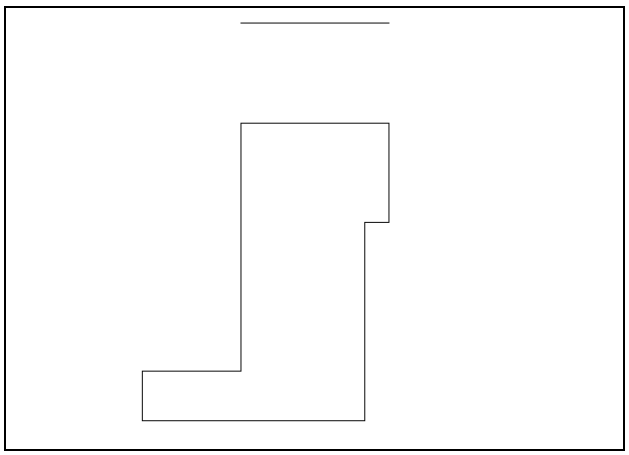


Figure 14.12 – Draw a Line 1.0 Right

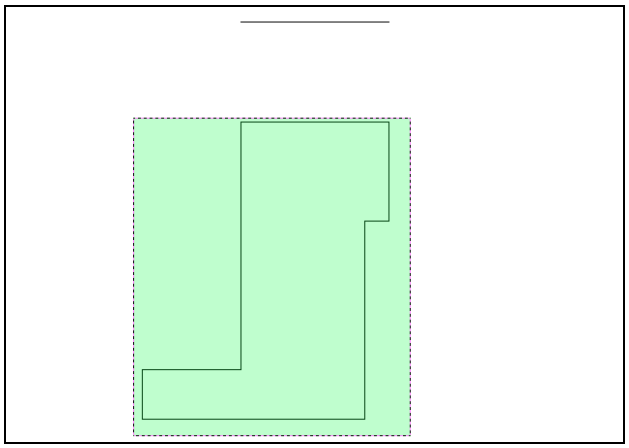


You could use the Polyline tool to draw this entire simple perimeter and then go straight to revolving the perimeter around the center line. But often you may find the task easier to do if you are using the Line tool like you did in this exercise. The Polyline function is very rigid; many computer aided designers will use the Line function so that they can use the Offset, Trim or Fillet tools to modify the perimeter. As you complete the 3D problems in this textbook, you will develop preferences for some tools over others.

Figure 14.13 – Close the Series of Lines

Creating a Joined Perimeter Using the Edit Polyline Tool

To merge the lines and arc segments together you need to use the Edit Polyline command. Select the Edit Polyline tool on the Modify II toolbar and the command line will prompt you to "**Select polyline to edit:**". Select one of the lines in the part and hit ENTER.



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. 6 vertices added to polyline." Hit Escape to exit the Edit Polyline command.

Figure 14.14 – 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 14.15. Next, choice two points to describe the axis of rotation. Pick the Endpoint of the centerline as shown in Figure 14.16 for the first point. Pick the Endpoint of the centerline as shown in Figure 14.17 for the second point. 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 14.18.

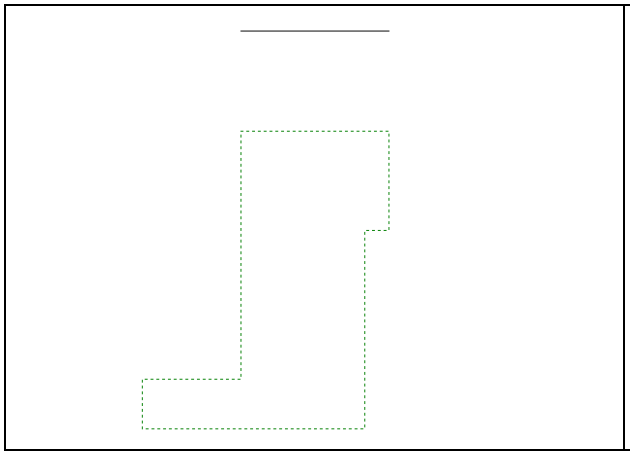


Figure 14.15 – Select the Polyline

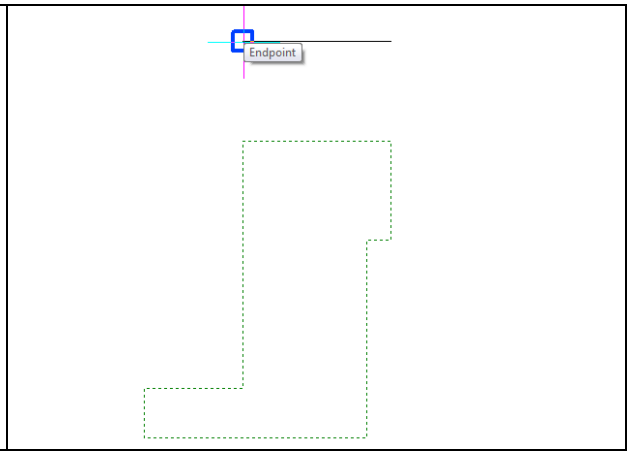


Figure 14.16 – The First Point of the Axis

Again the tessellation lines on the screen can be confusing, so you may wish to proceed to the next step of this exercise where you will use a Shade function to color the entire surface of the master solid.

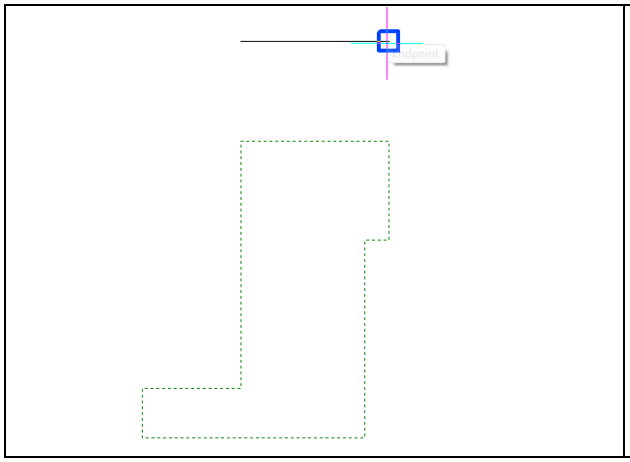


Figure 14.17 – Second Point of the Axis

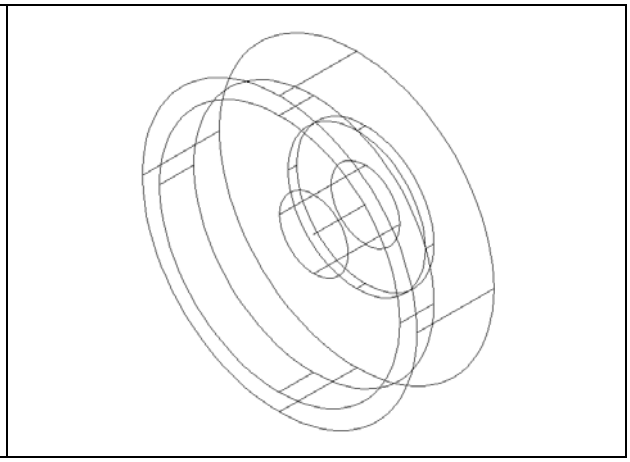
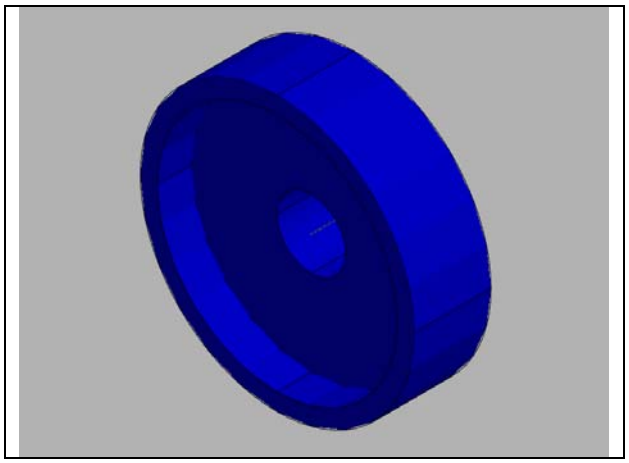


Figure 14.18 – The Revolved Solid

Shading the 3D Solid

To view the 3D solid with holes with more clarity, 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.



Select the Shade Highlighting Outline tool on the Shade toolbar. The part is now a solid blue color as shown in Figure 14.19. The tessellation lines associated with the Hide command are no longer interfering with your ability to read the drawing. Your customers will most likely prefer this mode to the Hide command since they can visualize the product clearly. If you ever desire to return to standard viewing, choose the 2D Wireframe tool on the Shade toolbar to return the drawing to the previous settings.

Figure 14.19 – Shade 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 14.20. Type **90** and **ENTER** to rotate the solid as shown in Figure 14.21. You can also erase the Centerline that did not get rotated.

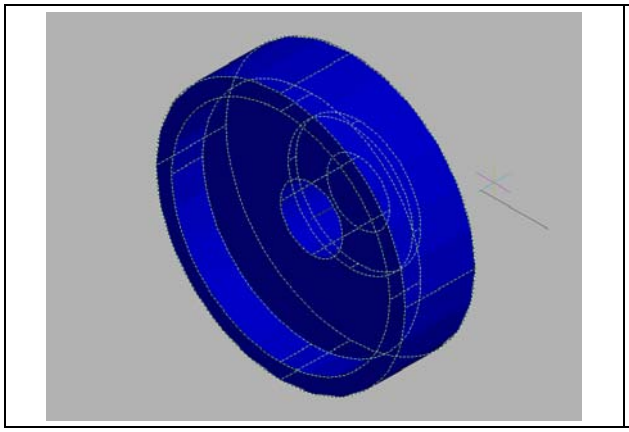


Figure 14.20 – The Rotation Axis

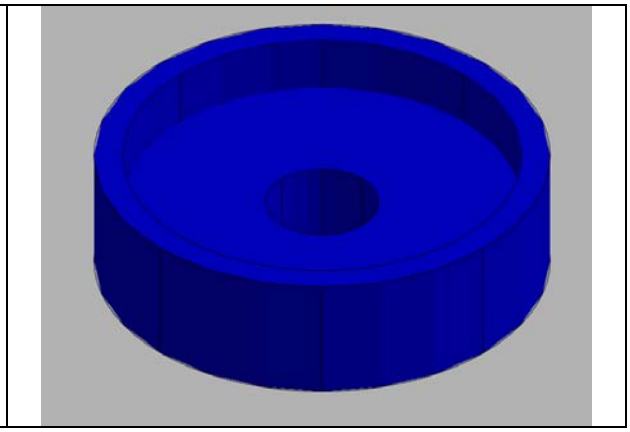


Figure 14.21 – The Rotated the Solid

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 Two Cylinders

You need to add four holes to the master solid. You will draw one hole going completely through the master solid, and the other three holes will be a copied from the first after you perform a polar array.

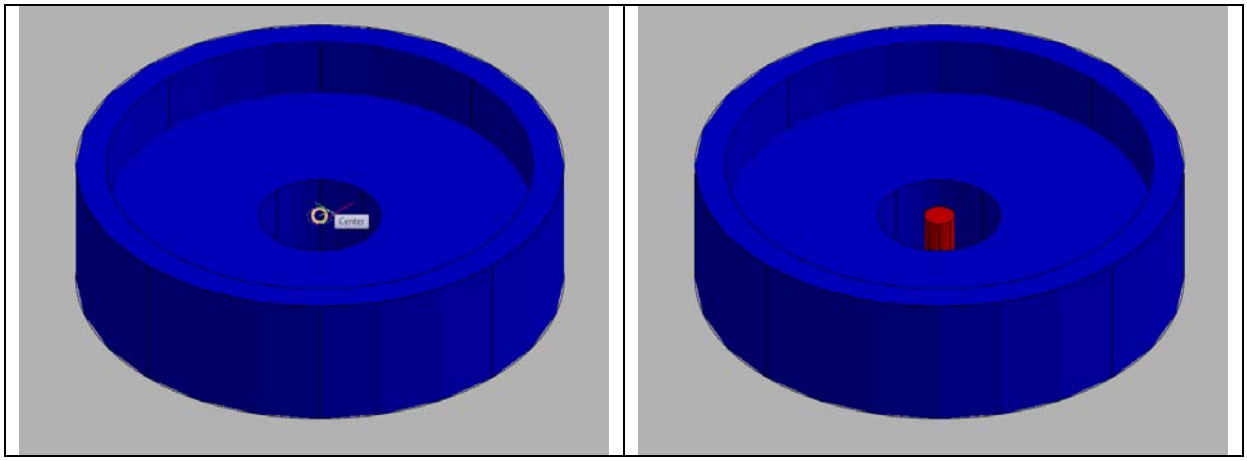


Figure 14.23 – Center of the Cylinder **Figure 14.24 – The Cylinder**

Pick the Cylinder tool on the Solids toolbar. Specify the center point for the top of the cylinder as the Center of the master solid as shown in Figure 14.23. 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 **-1.25** for the height of the cylinder and hit **ENTER,** and the cylinder will in the graphical display as shown in Figure 14.24.

Hit ENTER to repeat the Cylinder command. Specify the center point for the top of the cylinder as the Center of the master solid as shown in Figure 14.25. Type “**D”** for diameter and **ENTER** to input the sketch’s measurement of **0.75**. Hit **ENTER** to advance to the next prompt, which is “Center of second end/<Height of cylinder>:”. Key in **-0.25** for the height of the cylinder and hit **ENTER,** and another cylinder will in the graphical display as shown in Figure 14.26.

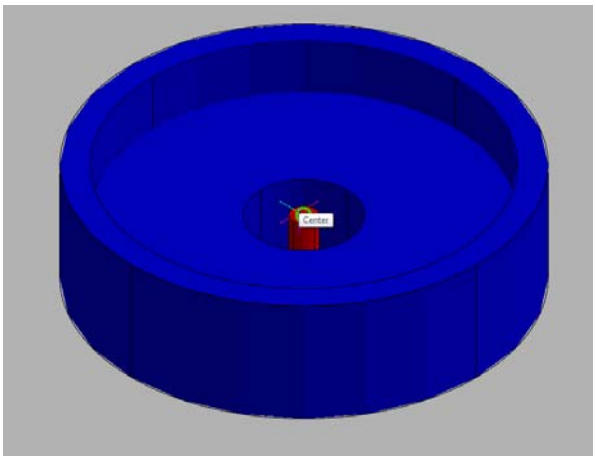


Figure 14.25 – Center of Second Cylinder

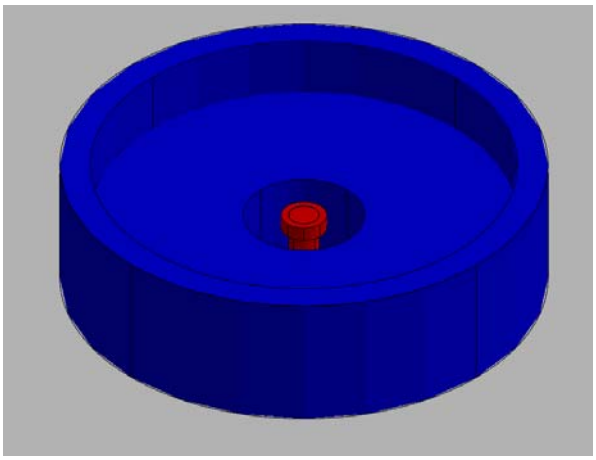


Figure 14.26 – The Second Cylinder

Union Two Solids

To combine both cylinders into a single 3D solid shape, use the Union command. 

Select the Union tool on the Solids Editing toolbar and you will be prompted to “Select ACIS Objects to Union”. Select both cylinders and the system will respond with “Entities in set: 2”. Hit ENTER and now all the solids will combine into one entity as shown in Figure 14.27. You can see in the graphical display the difference in that the tessellation lines outlining the two solids and now the overlapping tessellation lines are gone. To test your solid, select the entity with one pick of the mouse and the entire solid will become highlights.

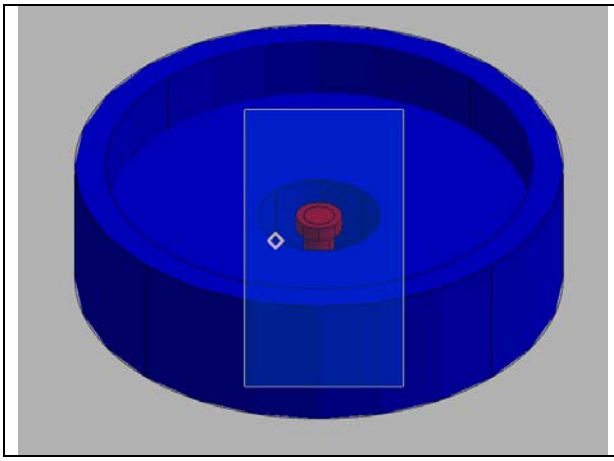


Figure 14.27 – Selecting Both Cylinders

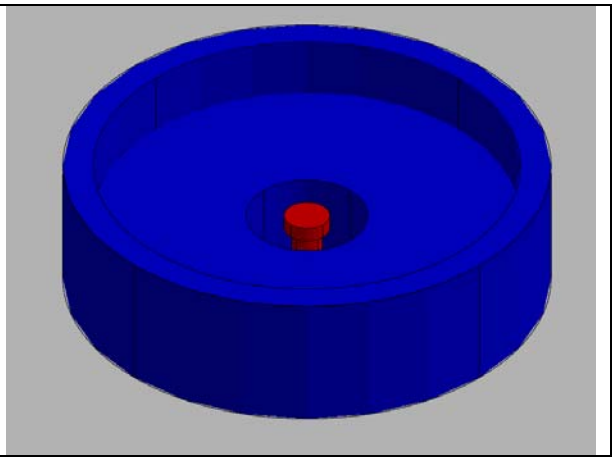


Figure 14.28 – The Combined Cylinders

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.

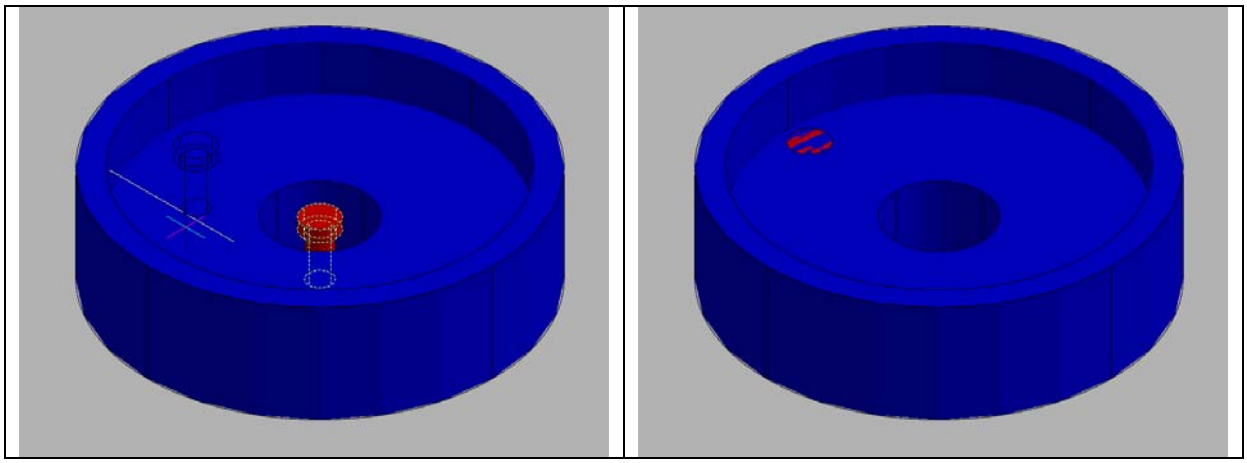


Figure 14.29 – Moving the Solid **Figure 14.30 –The Solid in Position**

To move the cylinder, 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 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 towards the left and type **3** and **ENTER** at the command line. The extruded circle will move 3 units in the direction of your cursor. The cylinder will be ready to polar array as shown in Figure 14.30.

Using the Array Command to Create Circular Patterns

You now need to copy and rotate the solid cylinder 3 times, spacing each one 90 degrees apart. Use the Array function with the polar option as the tool for this operation.

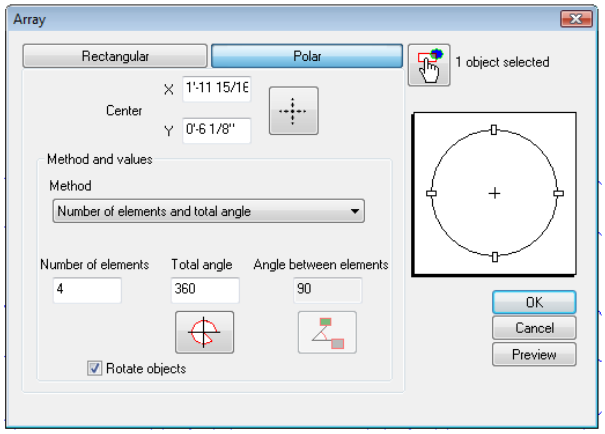


Figure 14.31 – The Array Window

Select the Array tool on the Modify toolbar, which will bring up the Array window as shown in Figure 14.31. Pick the Polar option, and then the Select Objects button that will allow you to select the cylinder as shown in Figure 14.32. 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 14.33.

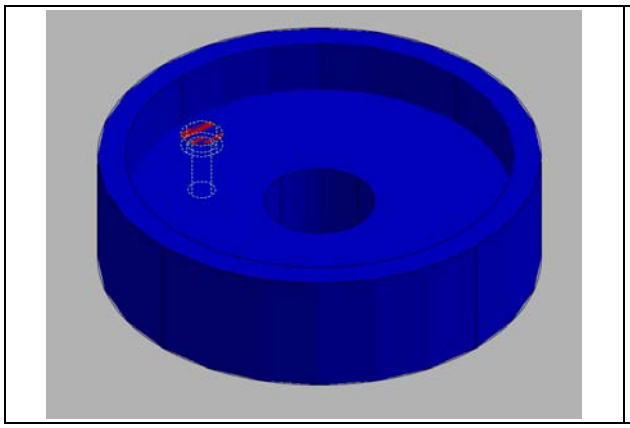


Figure 14.32 – Selecting The Cylinder

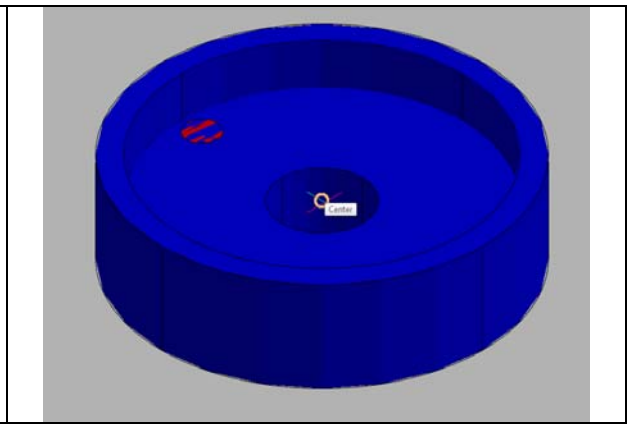


Figure 14.33 – Center of the Polar Array

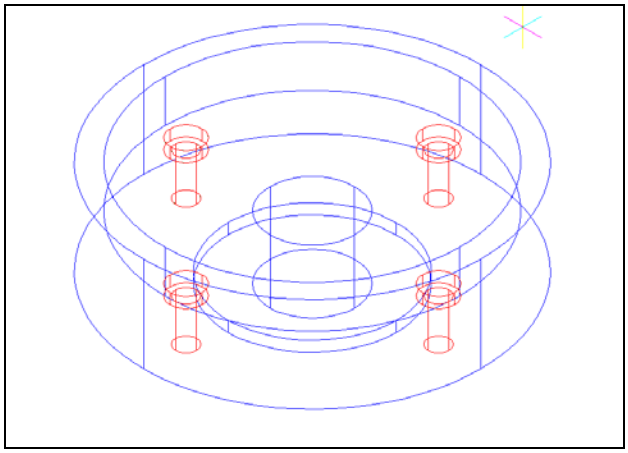


Figure 14.34 – 4 Solids to Subtract

Change the "Number of Elements" textbox to "4". 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 14.34, 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. You may need to rotate your view of the part to see all four cylinders. 

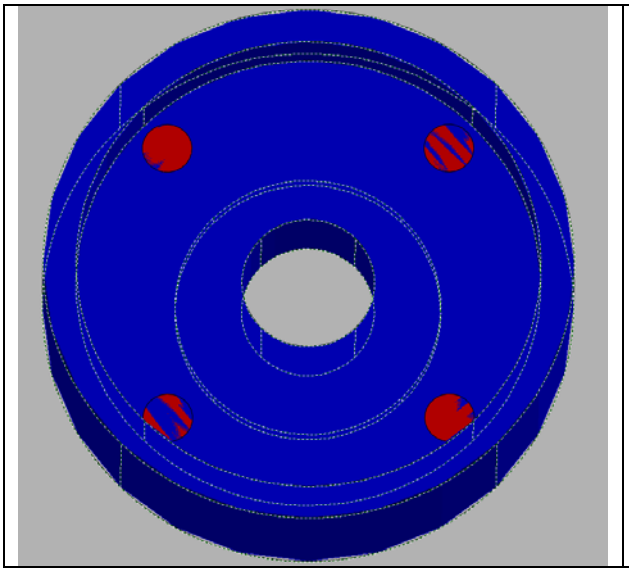


Figure 14.35 – Selecting the Master Solid

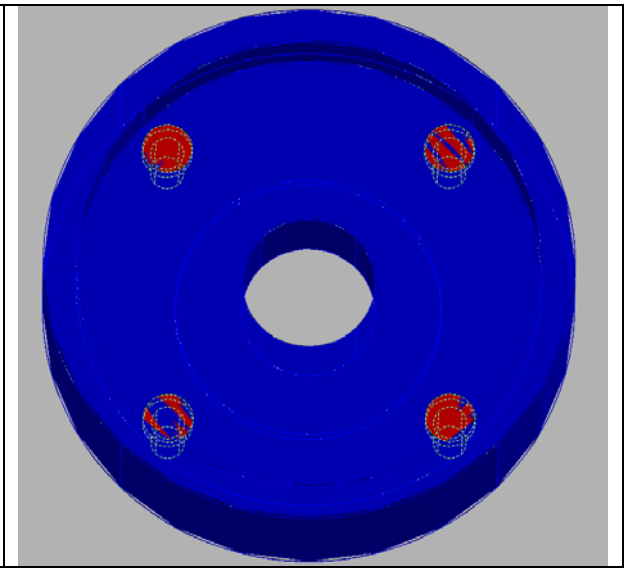


Figure 14.36 – Entities to Subtract

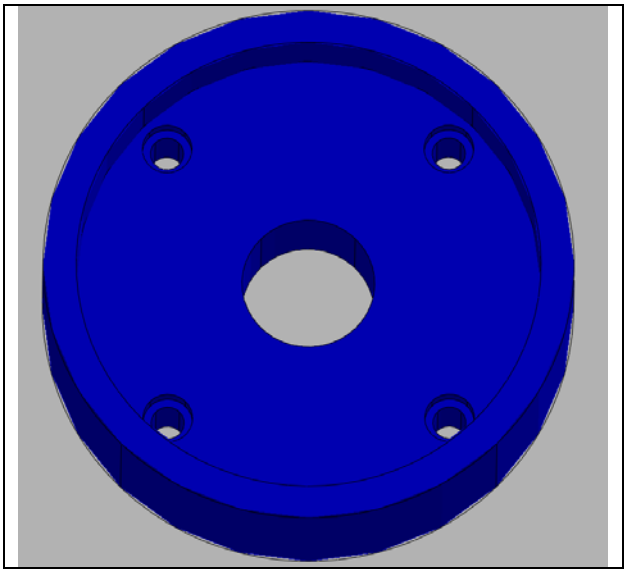
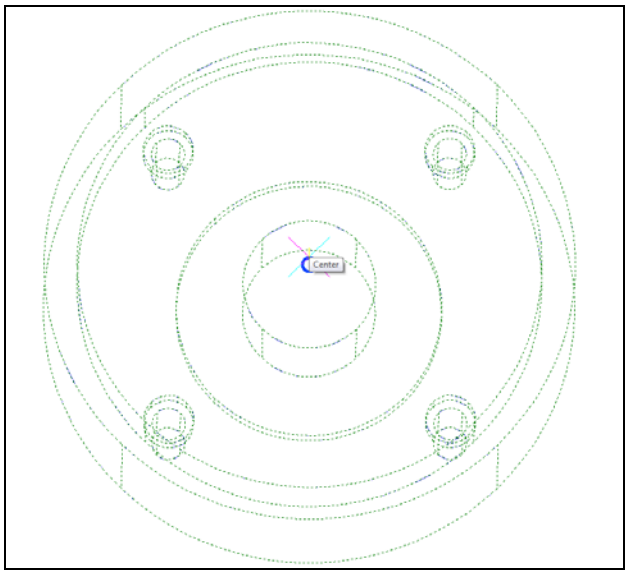


Figure 14.37 – Master Solid with Holes

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 14.35). Hit **ENTER** to proceed to the next step, which is to "Select ACIS objects to subtract:" (Figure 14.36). Pick the cylinders inside of the master solid and the command line will reply with "Entites in set: 4". Hit **ENTER** and a region exactly the shape of the cylinders will be removed from the main solid (Figure 14.37).

Moving the Solid to the Origin Point

To move revolving problem three 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 14.39 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 14.39 – Moving to the Origin

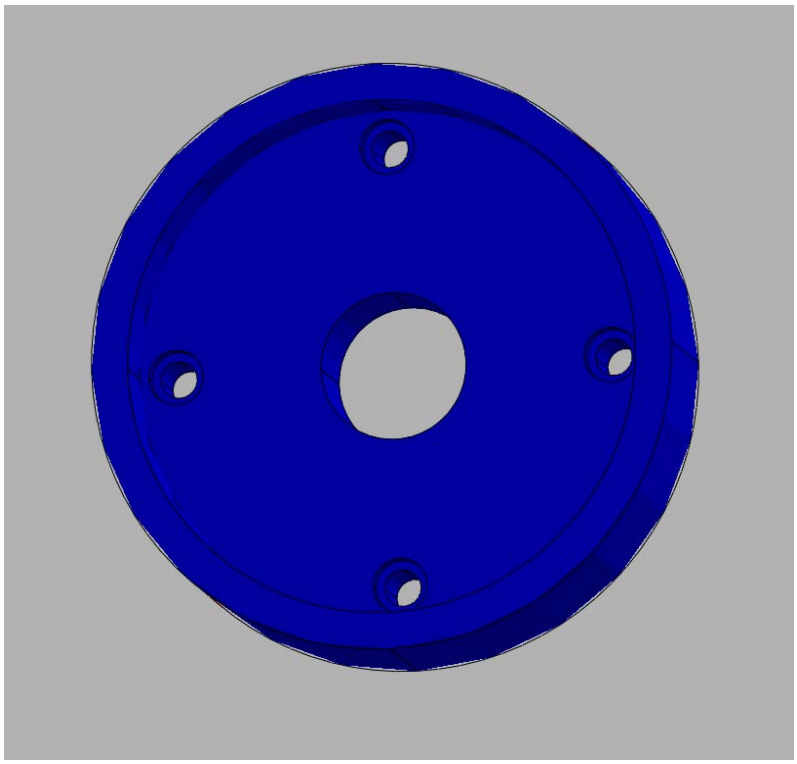


Figure 14.40 – The Finished Part

*** World Class CAD Challenge 101-13 * - 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 3, 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 3” and select the Save button to save the drawing (Figure 14.41).

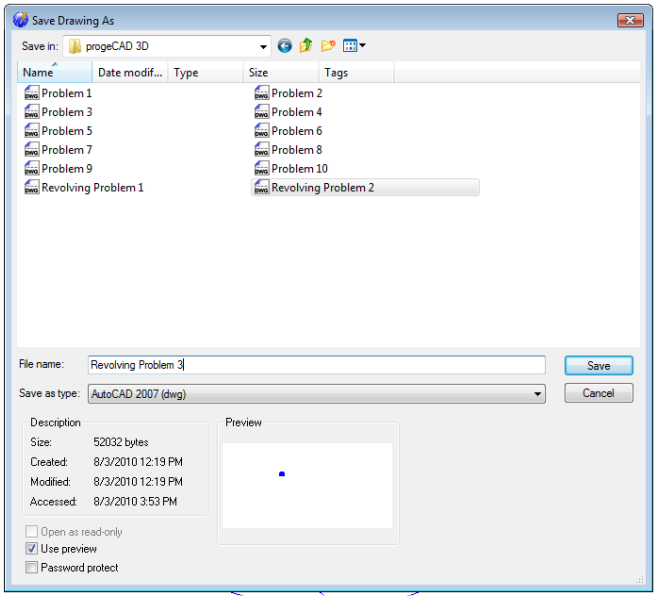


Figure 14.41 – Saving Revolving Problem Two

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