C h a p t e r

# 12

# Revolving Problem One

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

- **Sketch of Revolving Problem One**
- **Starting a 3D part drawing**
- **Modifying UCS settings**
- **Drawing a centerline using the Line command**
- **Creating a perimeter using the Polyline command**
- **Adding curved edges using the Fillet Command**
- **Revolving a solid from a closed polyline**
- **Rotating a 3D solid using the 3D Rotate command**
- **Remove the center line using the Erase command**
- **Drawing a solid cylinder**
- **Moving a solid into position on the master solid**
- **Creating an array of a solid using the Array command**
- **Subtracting 3D solids from the master solid**
- **Hiding the solid**
- **Moving the solid to the origin point**
- **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 XZ plane. In this exercise, you will draw and revolve a closed Polyline that contains the most complex shape of the component and then use simple geometric solids to subtract each additional detail. Revolving problem one's master solid contains radius edges in the main shape that you will place using the Fillet tool. After revolving the polyline perimeter, you will place a cylinder onto the master solid, and then create an array of the cylinder to subtract. Finally, you will utilize the Chamfer command and put a beveled entry on the three mounting holes and the hole in the hub.
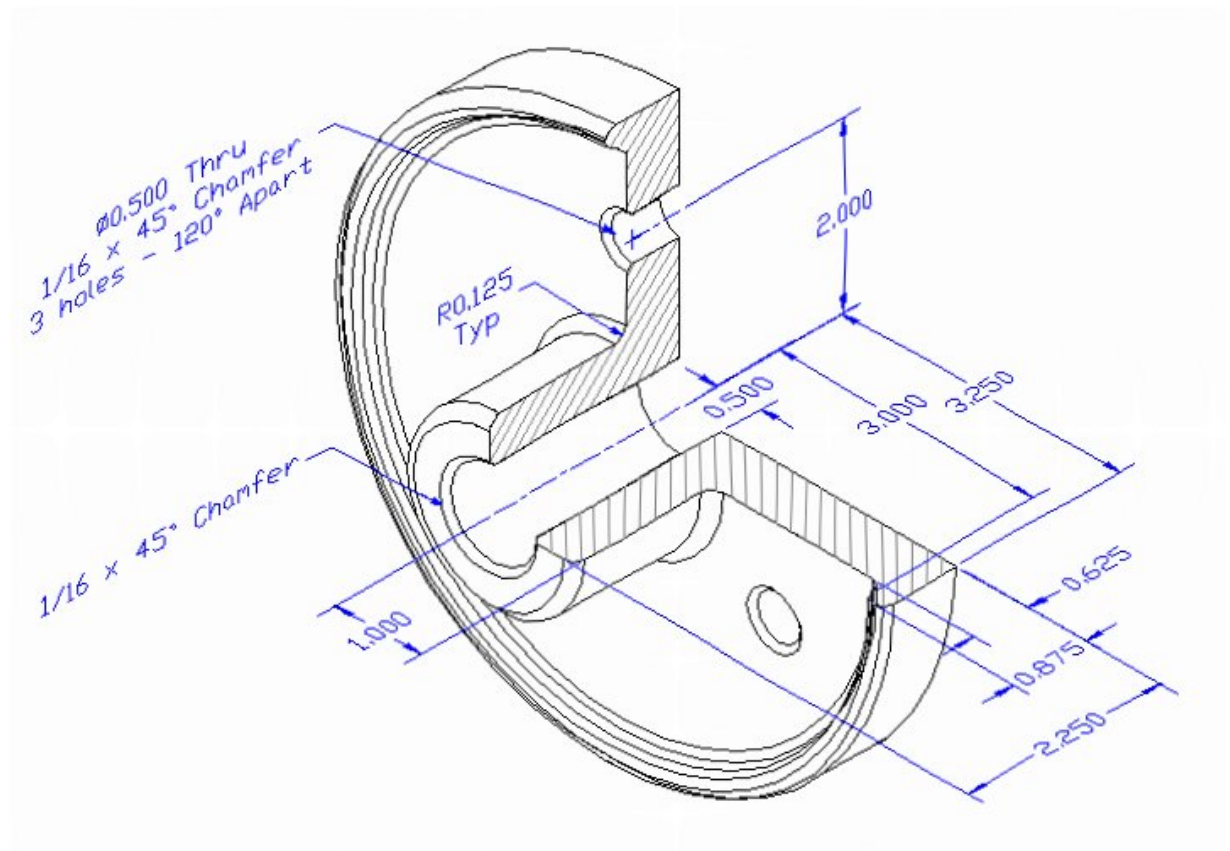


**Figure 12.1 – Revolving Problem One sketch**

Remember, when you receive a sketch from a professional, quickly identify the different shapes and deciding how to begin the drawing. You will find that using this strategy of revolving a closed perimeter is very effective for creating circular parts. Place as much detail in the perimeter and interior detail as possible, so that you will have less detail to add to the 3D solid. All of the drills in the Fundamentals of 3D Drawing textbook will walk you through such common repetitive maneuvers that you will use often in 3D modeling.

# Starting a 3D Part Drawing

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You will begin Revolving Problem One using the Mechanical template.  First select the New tool on the Quick Access Toolbar. A list of templates is loaded in the "Select template" window.  Scroll down the list and select 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.
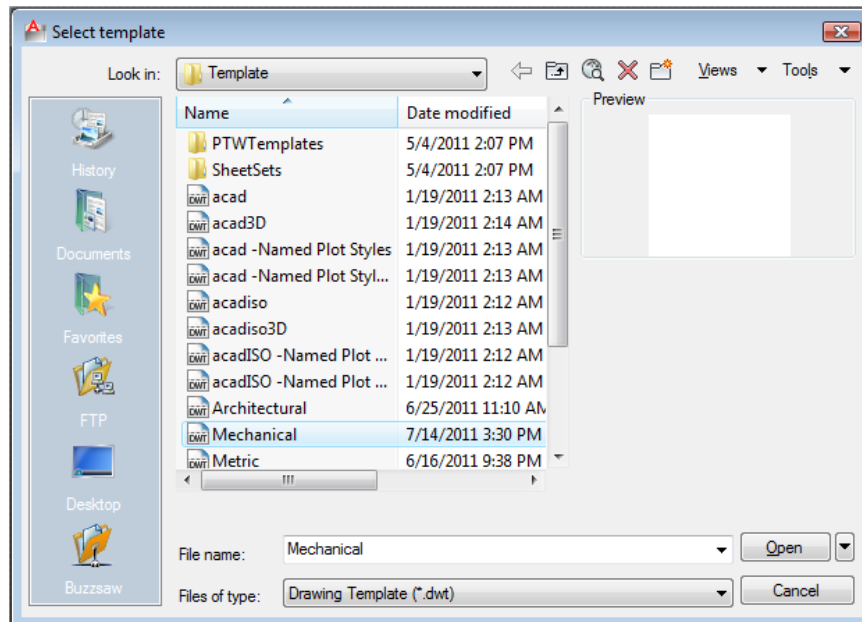


**Figure 12.2 – Starting the drawing using the Mechanical template**

# Modifying UCS Settings

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You may notice the UCS icon moving on the graphical display as you zoom or pan your view. If so, the UCS is currently representing the origin (0,0,0) of the drawing area.  This can sometimes be distracting, especially if the origin happens to lie in the vicinity of your drawing.  You can change the UCS settings so that the icon is always displayed in the bottom left-hand corner of the drawing area. Select the Show UCS Icon tool from the Coordinates panel under the Home tab (Figure 12.3) and select the "Show UCS Icon" option (Figure 12.4). You also have the choice of keeping the UCS displayed at the origin or turning it off completely.
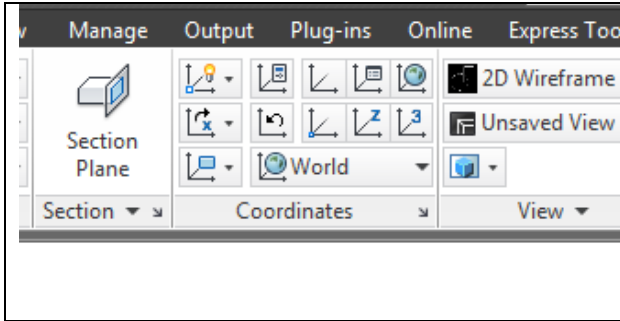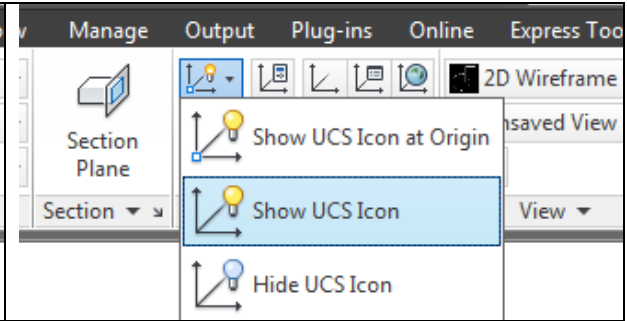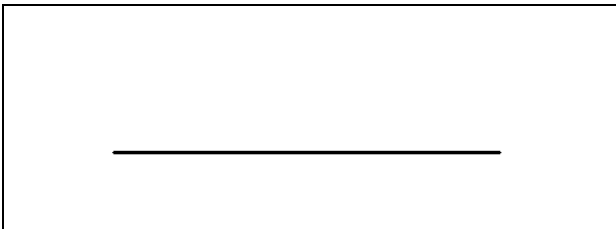
**Figure 12.3 - The Coordinates panel**

**Figure 12.4 - Changing the UCS display settings**

# Creating a Center Line Using the Line Command

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First, you need to draw a centerline with the Line tool from the Draw panel. Select a start point of the line in the bottom left-hand corner of the screen.



With the Ortho mode "on", drag the cursor to the left and type "**2.25**". Hit **ENTER** to complete the function. A line will appear on your graphical display as shown in Figure 12.5.

**Figure 12.5 – The centerline**

# Creating a Perimeter Using the Polyline Command

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Now you will create the closed polyline that you will then rotate around the centerline to create the master solid. You will create the polyline above the centerline and then use the Move command to place the outline in position.

```
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 command line prompts for the Polyline command**

**Figure 12.7 – Polyline moved above the centerline**
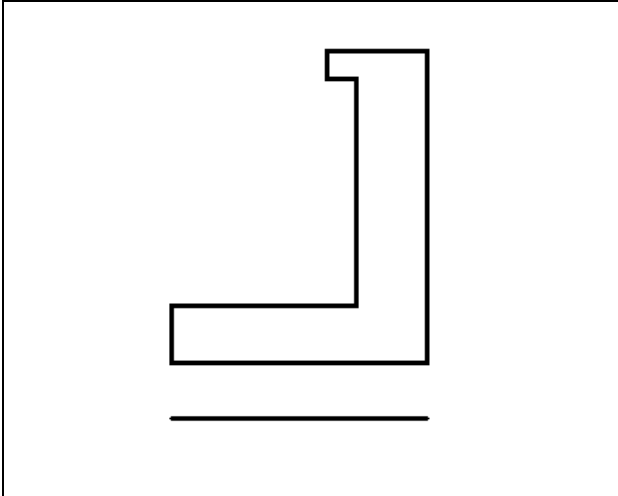
With the Ortho mode "on", use the Polyline command to start a line on the left endpoint of the centerline. Draw a **2.25** unit line to the right, a **2.75** line up, a **0.875** line to the left, a **0.25** line down, a **0.25** line to the right, a **2.0** line down, a **1.625** line to the left, and then type "**C**" and **ENTER** to close the Polyline. A closed outline will appear in the graphical display. Now use the Move tool to move the outline 0.5 units above the centerline as shown in Figure 12.7.

## Adding a Curved Edge Using the Fillet Command

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You placed a curve on the edge of a 3D solid in a previous problem, but on this exercise you will change the Polyline entity by placing the arc in the 2D perimeter before revolving it around the centerline.

Select the Fillet tool on the Modify panel and then type **R** and **ENTER**. Type **0.125** and **ENTER** to set the system for a 1/8 radius arc. Also type **M** and **ENTER** so you can place multiple 0.125 arcs on the perimeter. 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 corner of the polyline as shown in Figure 12.10.
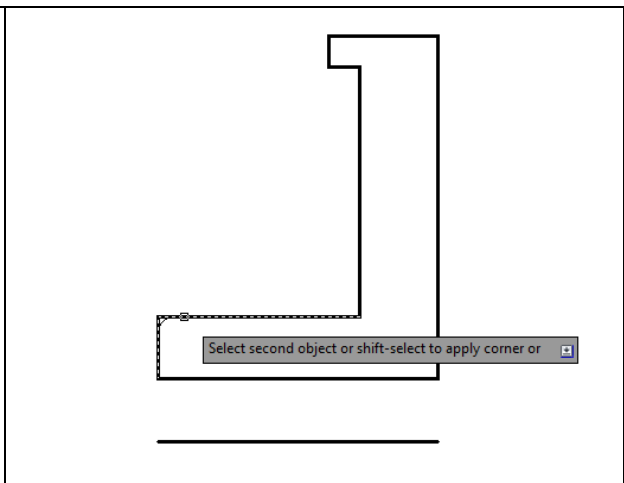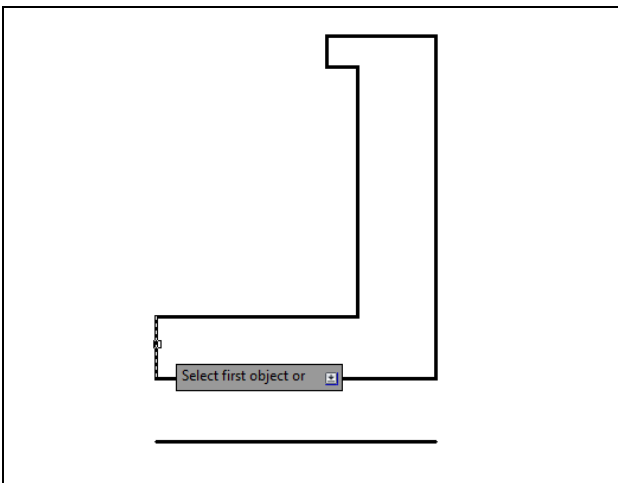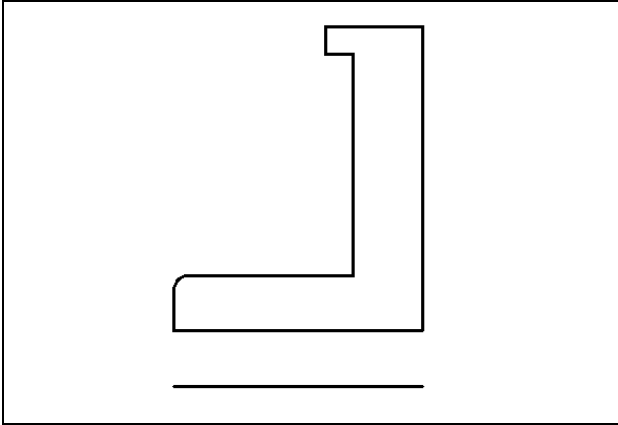


**Figure 12.8 – Selecting the first edge**



**Figure 12.9 – Selecting the second edge**

**Figure 12.10 – 0.125 arc on the polyline**

Continue to place arcs using the same technique. The next arc is just to the right of the first and the radius is still set, so select the horizontal and vertical line to place another arc as shown in Figure 12.11. The third curved segment as shown in Figure 12.12 is above the second. 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
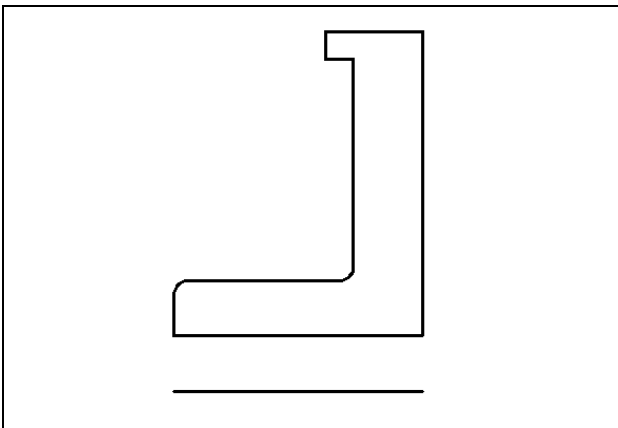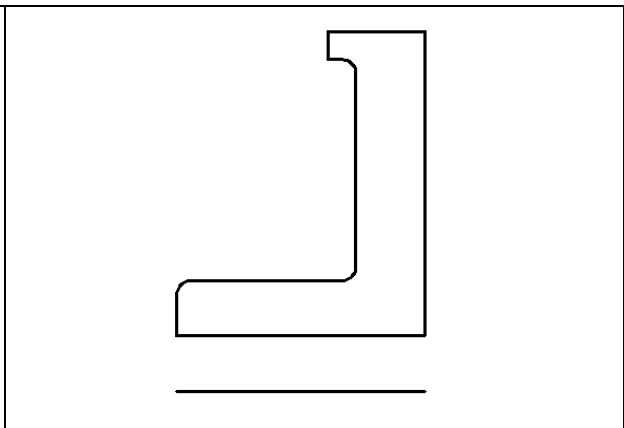


**Figure 12.11 – The second fillet**



**Figure 12.12 – The third fillet**

Remember that you can use the Fillet command in 2D or 3D. When you use the tool in 2D, you must select two lines to describe the corner on which you want to add the curve. When you are utilizing the tool in 3D, you only select the edge because it represents the intersection of two lines. Both techniques are effective.
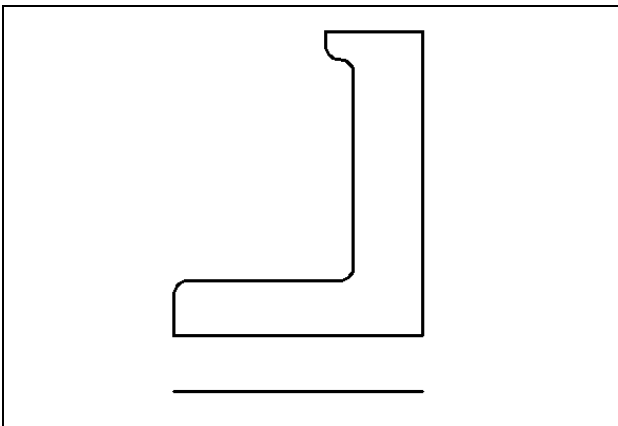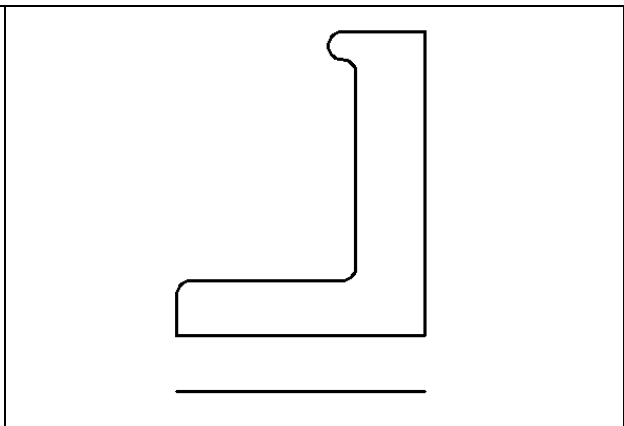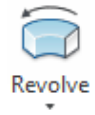


**Figure 12.13 – The fourth fillet**



**Figure 12.14 – The fifth fillet**

# Revolving a Solid from a Closed Polyline

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After you add the last fillet, you are ready to revolve the closed Polyline into a very complex 3D Solid. You will use the centerline for the axis of rotation when turning the perimeter into the solid.

Select the Revolve tool by clicking on the drop-down arrow below the Extrude icon on the Modeling panel and selecting the Revolve tool. The command line will prompt you to "select objects". Pick the perimeter 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 left endpoint Osnap of the centerline as shown in Figure 12.16 for the first point. Pick the right endpoint Osnap of the centerline as shown in Figure 12.17 for the second point.
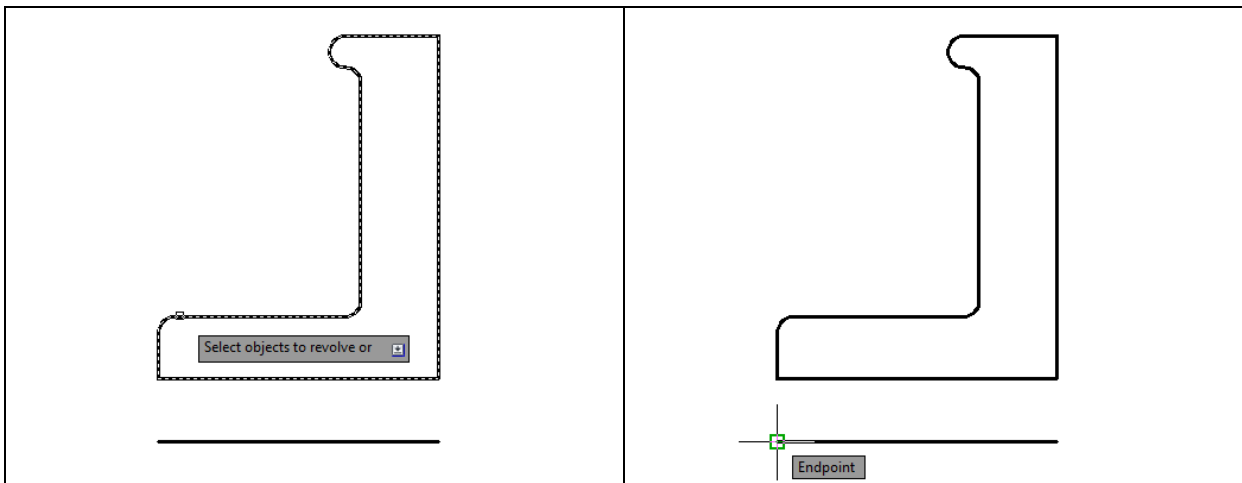
**Figure 12.15 – Select the polyline**        **Figure 12.16 – The first point of the axis**

You will then be prompted to "Specify angle of revolution or [STart angle/Reverse/EXpression] <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. Select the Southwest Isometric View to allow you to see the part in three dimensions more clearly as shown in Figure 12.18.
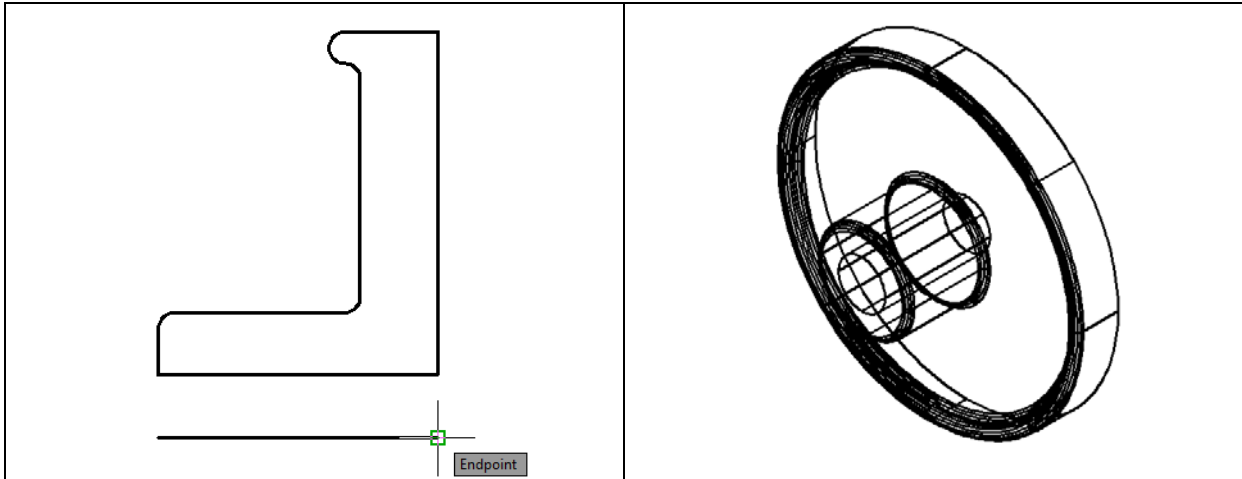
**Figure 12.17 – The second point of the axis**     **Figure 12.18 – The revolved solid**


# Rotating a 3D Solid Using the 3D Rotate Command

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Select the 3D Rotate command on the Modify panel. At the prompt "select objects", pick the solid on the graphical display and hit **ENTER**.  Now select the bottom quadrant Osnap as the base point of rotation.  Specify the Y axis as the axis of rotation by selecting the circle that orbits the Y axis (Figure 12.19).  With the Ortho mode "on", specify an angle start point and then use the cursor to move the solid 90 degrees clockwise (Figure 12.19).  Finish the rotation by clicking with the left mouse button and the solid will appear rotated on its base as shown in Figure 11.12.
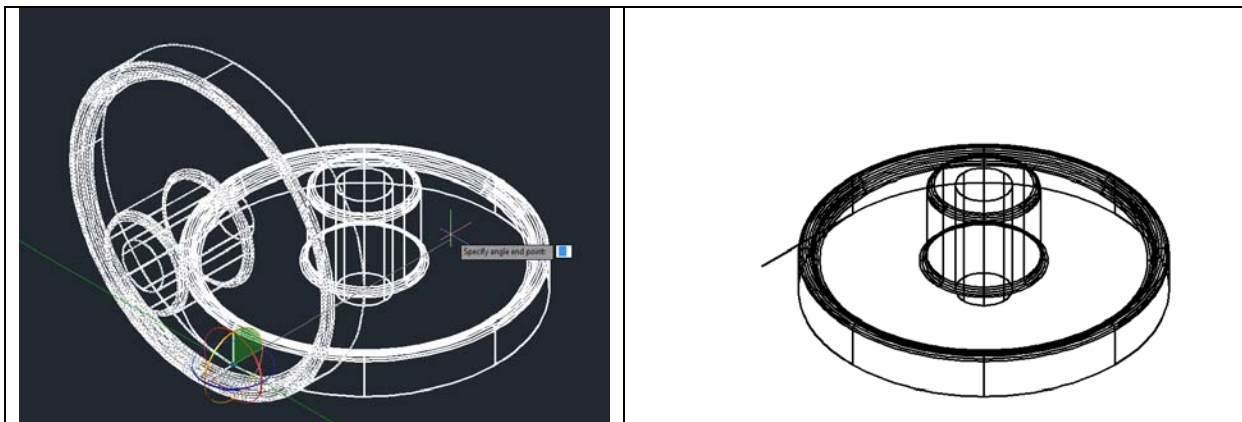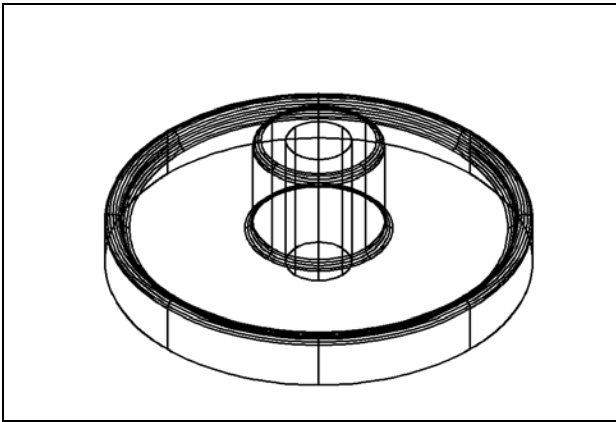



**Figure 12.19 – Specifying angle end point**     **Figure 12.20 – The rotated solid**

# Removing the Center Line Using the Erase Command
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Don't forget to erase the centerline that you used as the axis of revolution after you rotate the solid on its base. Select the line and hit DELETE on the keyboard or use the Erase command from the Modify panel. It is always important to remove construction entities as you go along so that they do not interfere with the details you will continue to add to a drawing.

**Figure 12.21 – The centerline erased**


# Drawing a Solid Cylinder
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You need to add three holes to the master solid: one 0.5 diameter hole going completely through the master solid, and two more identical 0.5 diameter holes that will you will create with the Polar Array command.
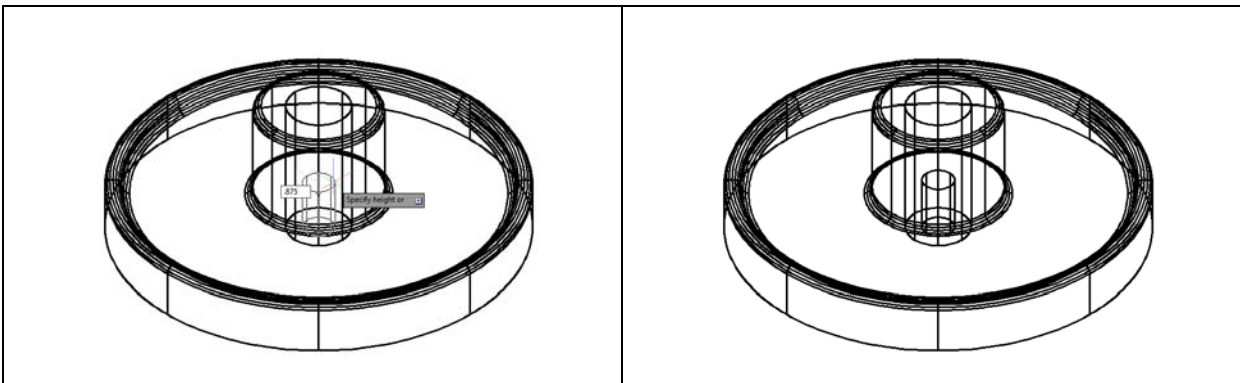


**Figure 12.22 – Placing the cylinder**          **Figure 12.23 – The cylinder**

Select the Cylinder tool on the Modeling panel under the drop-down arrow beneath the Box command. Specify the center point for the base of the cylinder as the center Osnap of the middle of the part. Type "**D"** for diameter and hit **ENTER** and input the sketch's measurement of **0.5**. Hit **ENTER** to go to the next prompt, which is to "specify the height of the cylinder". With Dynamic Input toggled "on", pull the cylinder down into the master solid, type **0.875** for the height of the cylinder and hit **ENTER** (Figure 12.22). The cylinder will appear as shown in Figure 12.23. If you do not have Dynamic Input on type **-0.875** to pull the cylinder -0.875 units on the Z axis.

# Moving 3D Solids into Position on the Master Solid

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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.
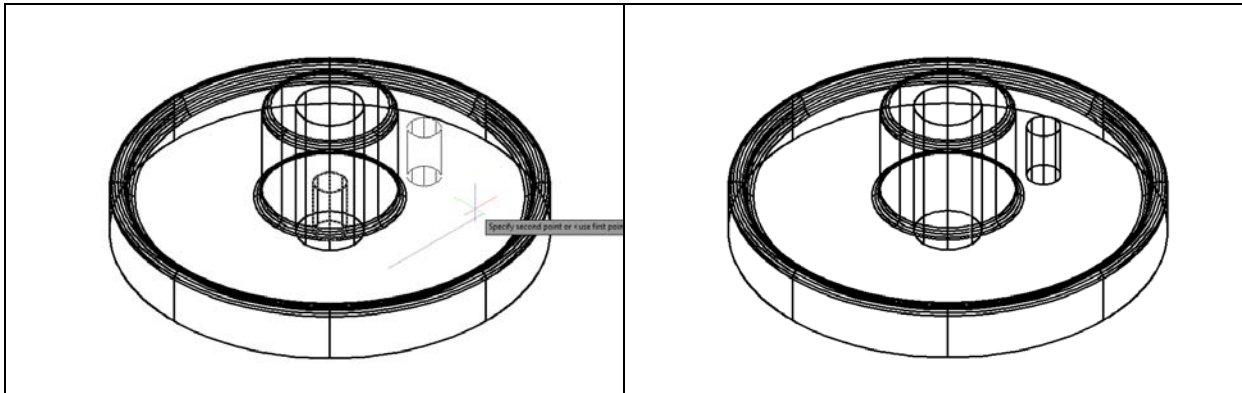


**Figure 12.24 – Moving the cylinder**          **Figure 12.25 –The cylinder in position**

To move the cylinder, select the Move tool on the Modify panel.  Once you choose the Move command, the prompt will be "Select Objects".  Pick the cylinder and the command line will respond with "1 found", so just press **ENTER** to go the second part of the command.  Next you need to "specify base point or displacement", which will be anywhere on the graphical display as shown in Figure 12.24.  For the "second point of displacement", 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

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

Select the drop-down arrow next to the Array tool on the Modify panel and select the Polar Array tool.  First you will be prompted to select objects, so select the cylinder and hit ENTER (Figure 12.26).  Next you need to specify the center of your polar array, so select any center Osnap point that corresponds with the middle of the master solid (Figure 12.27).  Enter **3** for the number of items, and when prompted to specify the angle to fill type **360** and **ENTER** (Figure 12.28).  Two more cylinders should appear so that each is 120 degrees from the other as shown in Figure 12.29.
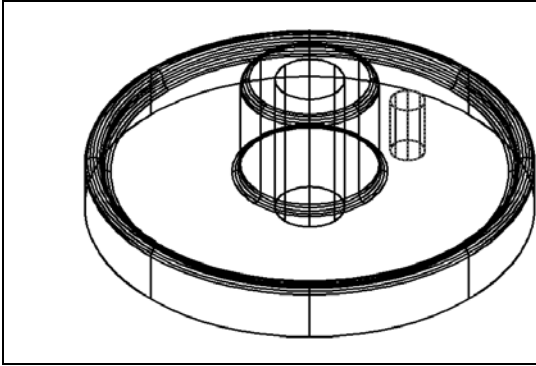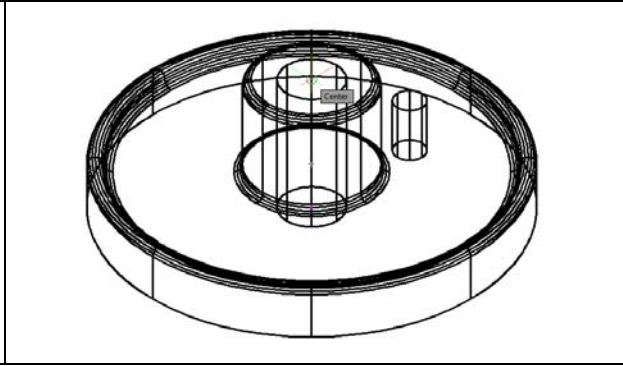
**Figure 12.26 –Selecting Objects**
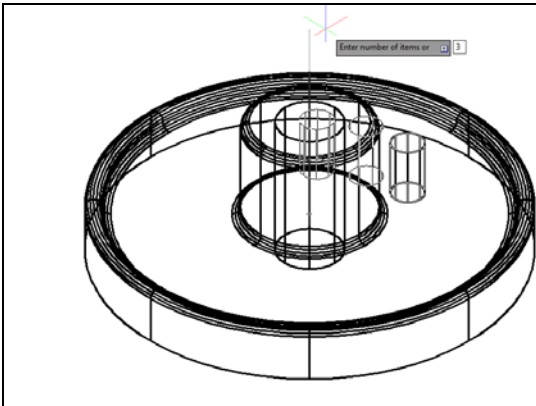


**Figure 12.27 – Center of the Polar Array**
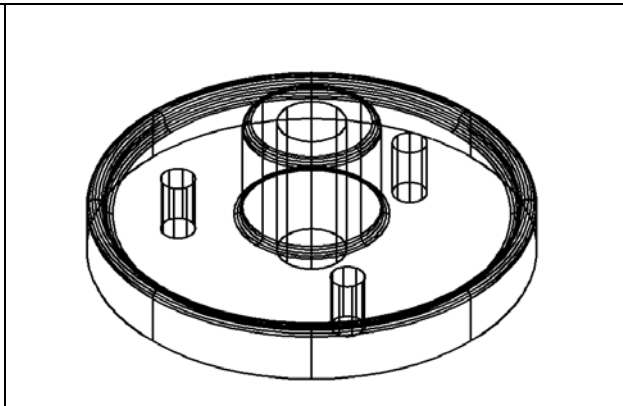


**Figure 12.28 – Subtracting Solids**



**Figure 12.29 – Subtracting Solids**

## Subtracting 3D Solids from the Master Solids

Before subtracting the solids, you must explode the polar array by selecting the Explode too from the Modify panel and then selecting the array and hitting ENTER. Now you are ready to subtract the three cylinders from the master solid.
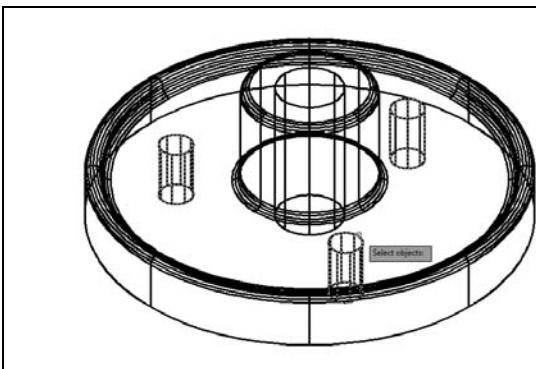


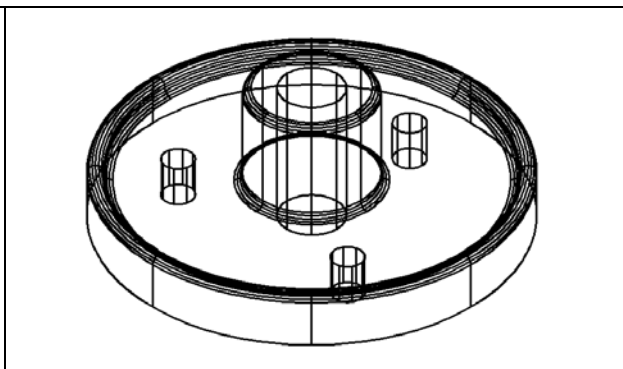**Figure 12.30 – Selecting the cylinders**



**Figure 12.31 – The subtracted solids**

To remove the cylinders from the master solid, select the Subtract tool on the Solid Editing panel. The command line will prompt you to "**Select solids and regions to subtract from**". Select the master solid and the system will respond with "**1 found**". Hit **ENTER** to proceed to the next step, which is to "**Select solids and regions to subtract**". Pick the three cylinders inside of the master solid and the command line will respond with "**3 found**" (Figure 12.30). Hit **ENTER** and the shape of the two cylinders will disappear from the master solid as shown in Figure 12.31.

## Adding a Bevel Edge Using the Chamfer Command
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You can use the Chamfer command to add a beveled edge to the three holes. In many parts, adding a broken edge will help the worker during assembly.
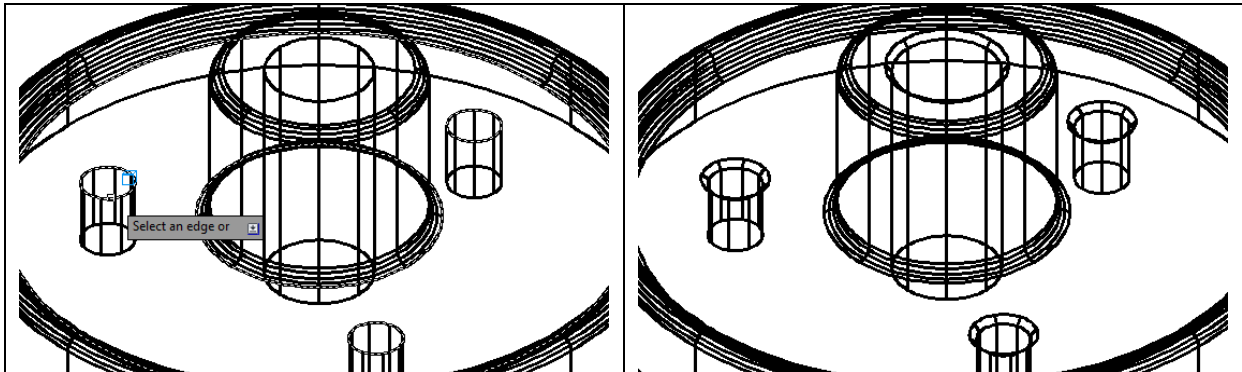


**Figure 12.32 – Selecting edges**          **Figure 12.33 – Four chamfered edges**

On the Modify panel, select the drop-down arrow next to the Fillet tool and then select the Chamfer tool. Use the pick box to select the top edge of one of the 0.5 holes (Figure 12.32). Hit ENTER, and then type "**0.0625**" for the offset distance. Hit ENTER and type "**0.0625**" for the second offset distance. Hit ENTER once more, and also type **M** and ENTER so you can then select the top edge of each 0.5 hole on the master solid. 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 hitting ENTER on the keyboard, and place another bevel on the inside edge of the middle hole of the master solid as shown in Figure 12.33

## 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 the solid unless its material was transparent. You can use a different visual style to remove the hidden lines from your graphical display.
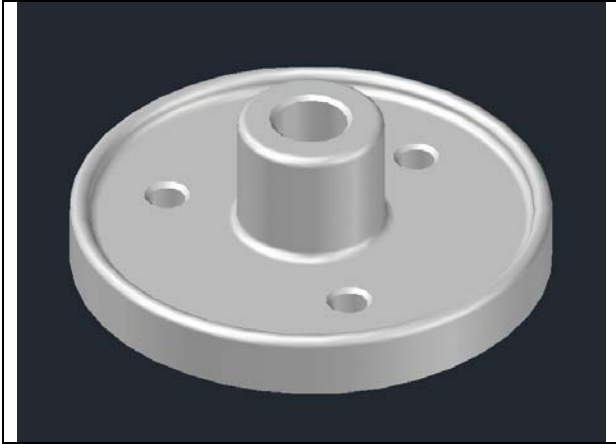
Realistic

Select the Realistic visual style from the Visual Style panel under the View tab. The part will appear rendered with a metallic sheen as shown in Figure 12.34. If you wish to return the graphical display to the previous visual style, choose the visual style called 2D Wireframe. Neither command alters the mass of the solid or any of the dimensions, but both are available for the computer aided designer as a viewing option.

**Figure 12.34 – The Realistic visual style**

## Moving the Solid to the Origin Point
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To move Revolving Problem One to the drawing origin of 0,0,0 on the x, y and z axis, select the Move tool from the Modify panel under the Home tab, select the 3D solid, and hit ENTER. At the prompt "specify base point or displacement," select the center Osnap of the bottom of the part as shown in Figure 12.35, which will become its insertion point into an assembly drawing. At the "second point of displacement," type 0,0,0 and ENTER, or #0,0,0 and ENTER if using Dynamic Input. Revolving Problem One will move to the origin point. Moving a part at the origin is important for when you wish to insert the solid into an External Referenced Assembly drawing.
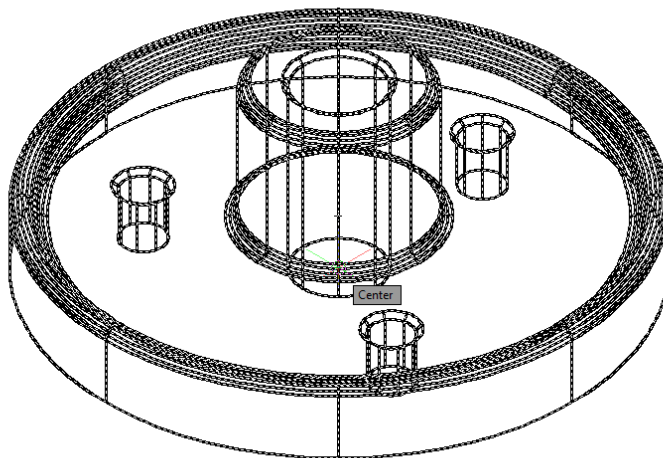


**Figure 12.35 – Moving the finished solid to the origin**

# Saving the Solid Problem

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To save Revolving Problem 1 in your Fundamentals of 3D Drawing Folder, select the Save As tool on the Quick Access toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. Type "Revolving Problem 1" in the File Name textbox and hit the Save button to save the drawing (Figure 12.36).
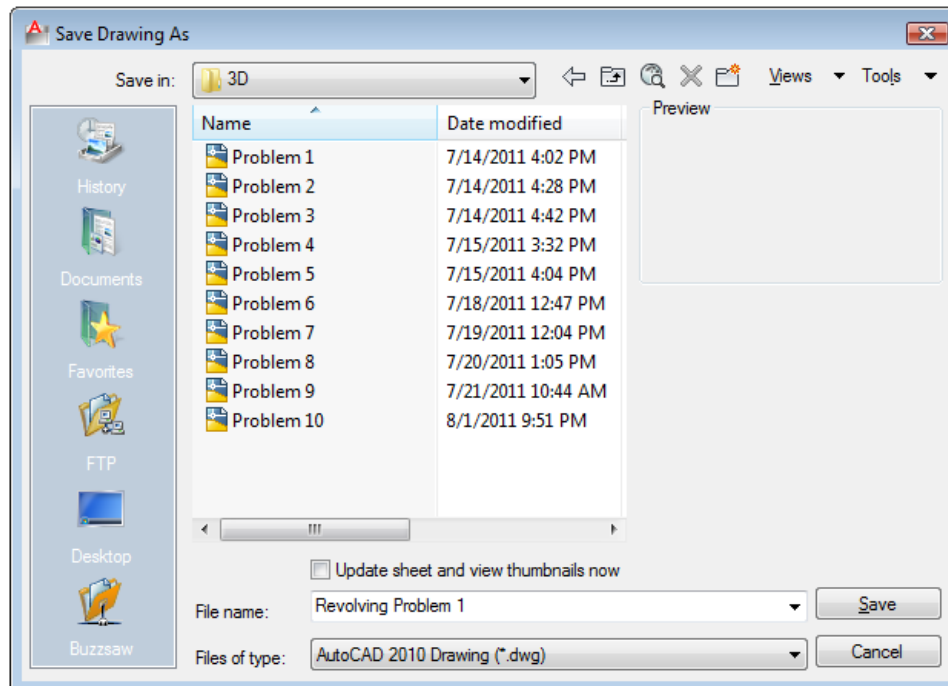


**Figure 12.36 – Saving Revolving Problem One**

**\* 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 in less than 5 minutes to maintain your World Class ranking.**

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