# 10

# Solid Problem Nine

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

1. Sketch of Solid Problem Nine
2. Starting a 3D Part Drawing
3. Modifying How the UCS Icon is Displayed
4. Drawing Two 3D Spheres
5. Subtracting a 3D Solid from Another Solid
6. Creating a Hemisphere Using the Slice Command
7. Adding a Cylinder
8. Subtracting a 3D Solid from Another Solid
9. Hide the Solid
10. Moving the Solid to the Origin Point
11. Saving the Solid Problem

# Sketch of Solid Problem Nine

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You will start the ninth solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across one line, which you can call the Y-axis. In this exercise, a strategy you will continue to use is to draw closed Polylines that contains the most complex shape of the component and then use simple geometric solids to subtract detail. In problem nine, the master solid has half of a spherical shape. You will create two spheres, removing the mass of the second from the first. After adding a cylinder, subtract it to create a hole through the core of the ball. You will finish by cutting the master solid in half to end with the product as shown in Figure 10.1.
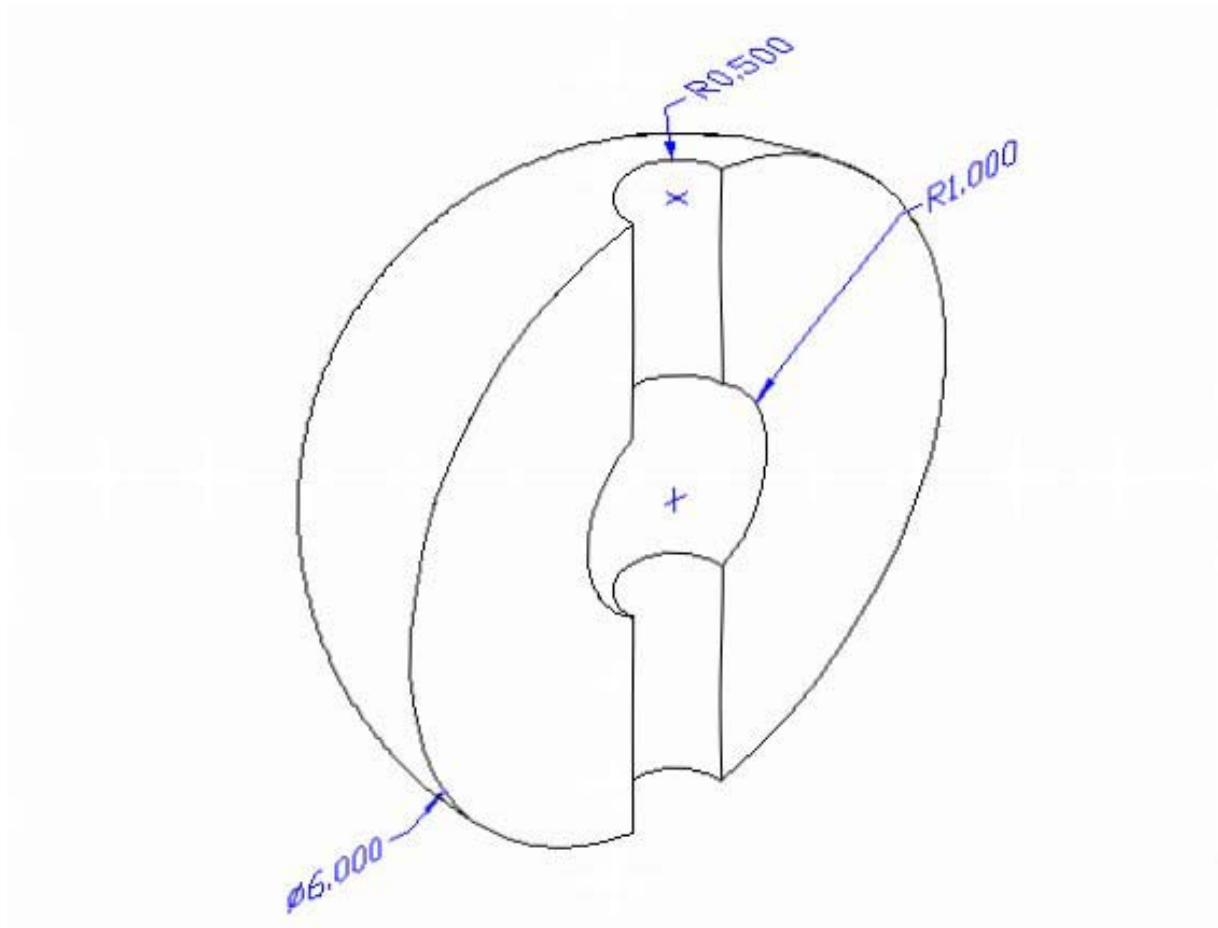
**Figure 10.1 – Problem None Sketch**

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. You can see rather quickly that problem has different contours so separate each extrusion or solid that is made and subtract them from the master. This problem will help you address solid parts where there are different complexities and sizes. All ten 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

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In problem nine, you will begin the 3D drawing by selecting the Mechanical template. In Figure 10.2, the window "Create New Drawing" is loaded by selecting the New tool on the Standard Toolbar. Select the "Use the Template" (third button) at the top of the Create New Drawing window. A list of templates is loaded in the Select a Template list box. 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.
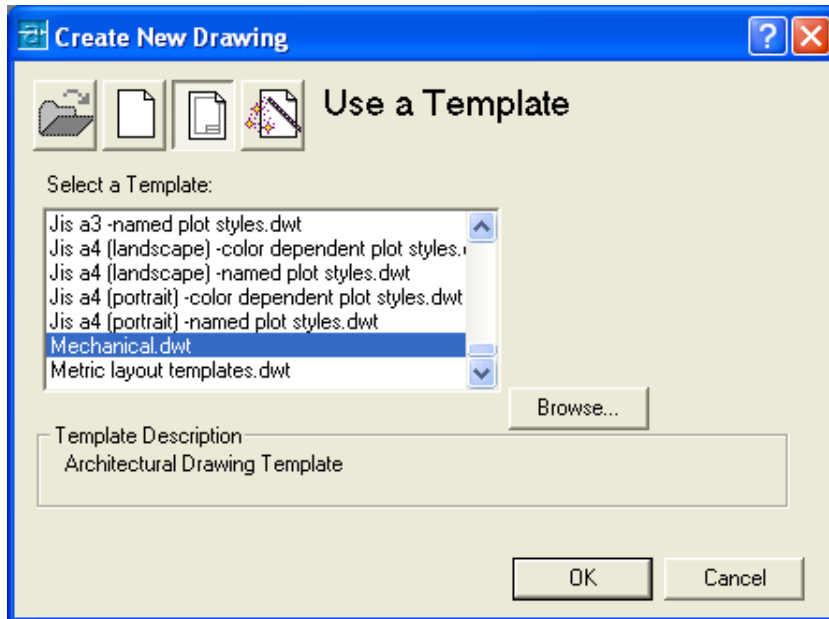


**Figure 10.2 – Starting the Drawing Using the Mechanical Template**

## Modifying How the UCS Icon is Displayed
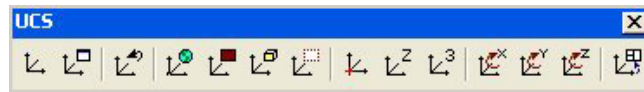
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**Figure 10.3 - The UCS Toolbar**

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 addition 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 toolbar and the UCS window will appear the graphical display. Pick the Settings tab to make the UCS Icon settings change. Remove the check from "Display at UCS origin point" as shown in Figure 10.4 and press the OK button to have the system variable change. Now you can start to create the solid.
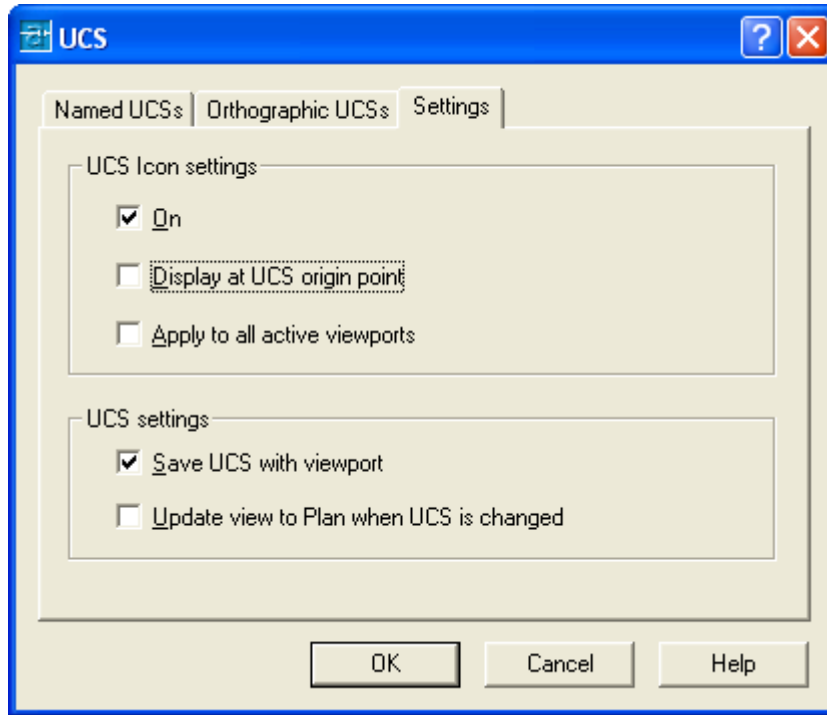
**Figure 10.4 - The UCS Window, Settings Tab**

# Drawing Two 3D Spheres

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Occasionally you will need to design a spherical component, so you will want to select the Sphere tool on the Solids toolbar. The command line information for the Sphere tools is very basic as shown in Figure 10.5.

```
Command: _sphere
Current wire frame density:  ISOLINES=4
Specify center of sphere <0,0,0>:
Specify radius of sphere or [Diameter]: 3
```

**Figure 10.5 – The First Sphere on the Command Line**
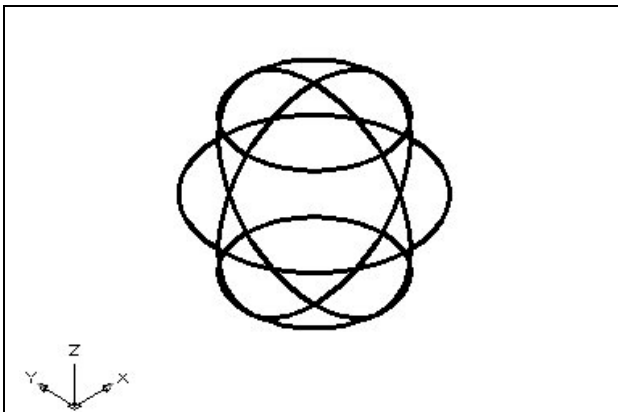


Pick anywhere in the middle of the graphical display to specify the center of the sphere. Like the Cylinder command, the next prompt will inquire for the radius of the sphere by default or type "D" for the diameter option. For this sphere, you will type **3** and **Enter. A** round ball will appear in the center of the display as shown in Figure 10.6. If you wish, you can try another sphere where you can type "D" for diameter and then input **6.** Both spheres will be the same size.

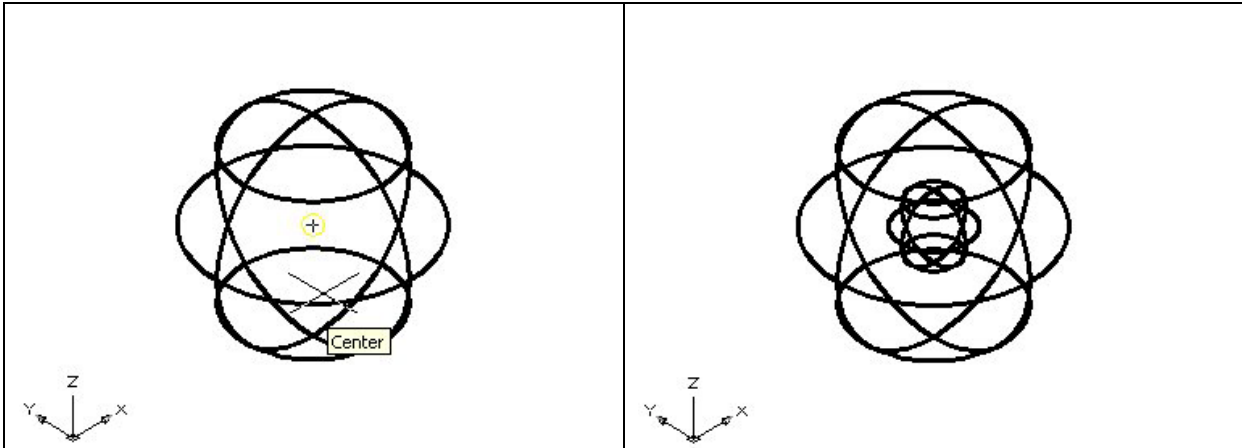**Figure 10.6 – Drawing the First Sphere**

**Figure 10.7 – The Next Sphere's Center    Figure 10.8 – The Second Sphere**

Press **Enter** to repeat the Sphere command. Pick the Center Osnap tool on the Object Snap toolbar and then place your cursor near a tessellation line as shown in Figure 10.7. The Center object snap will appear in the middle of the sphere, so pick that point. The next prompt will request the radius of the sphere by default, so you will type **0.5** and **Enter.** A round ball will appear in the middle of the first sphere as shown in Figure 10.8.

## Subtracting a 3D Solid from Another Solid

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To remove the small spherical solid from the master solid, select the Subtract tool on the Solids Editing toolbar.   The program will prompt you to "**Select solids and regions to subtract from**".
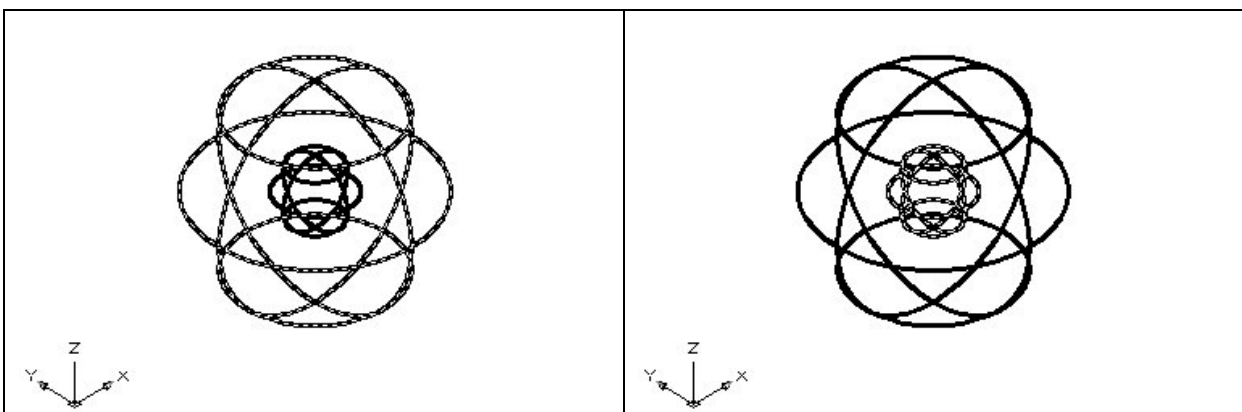


**Figure 10.9 – Selecting the Outside Sphere    Figure 10.10 – Selecting the Inside Sphere**

Select the larger master solid and the system will respond with "**1 found**" as shown in Figure 10.9. You need to Enter to proceed to the next step, which is "**Select solids and regions to subtract**". Pick the smaller sphere and the software application will counter with "**1 found**". Enter and a region exactly the shape of the small sphere is missing from the larger one as shown in Figure 10.10. Notice that you cannot see any difference in the solid when you subtract the cylinders.

# Creating a Hemisphere with the Slice Command

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Whenever you need only a portion of any solid, the Slice function will enable you to remove a fraction of a solid form the whole. Picking the location of the cutting plane can be challenging on complex solids, but for the spherical solid you are creating, the task is relatively simple.
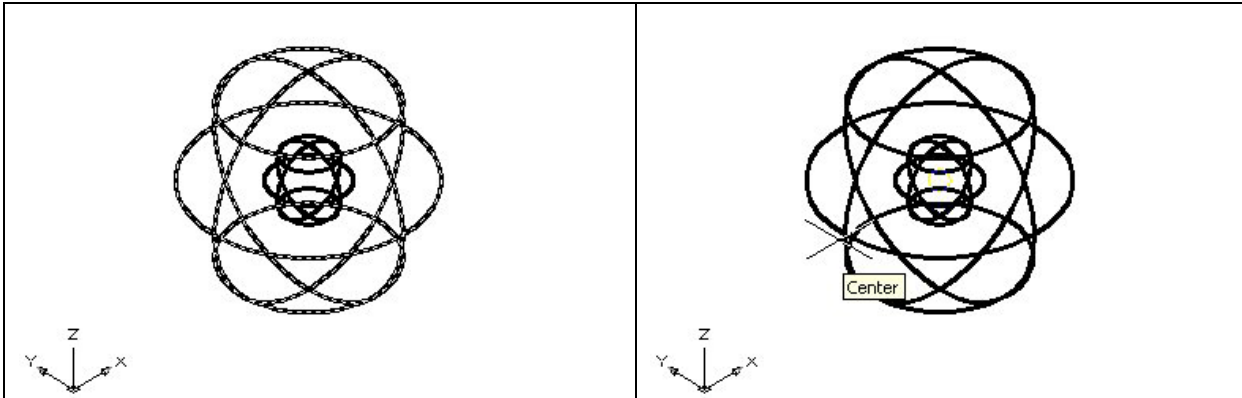


**Figure 10.11 – Selecting the Master Solid**      **Figure 10.12 – Selecting the Center**

Pick the Slice tool on the Solids toolbar, and the first step in the procedure is to pick the solid, which requires cutting. In Figure 10.11, you will select a tessellation line of the master sphere. The second action you will learn in this exercise is to pick the cutting plane. Type "**ZX**" to describe the path the virtual blade will cut the sphere. Now the third step is to pick the point on the plane where the invisible knife will cut. In Figure 10.12, you will again use the Center Osnap to choose the center of the solid ball. Making the selection divides the Sphere.

# Erasing the Front Hemisphere

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The World Class CAD does not overly rely on the erase function since removing entities once drawn is losing efficiency. Eventually, you will use the Slice function to remove a side of the cut solid.
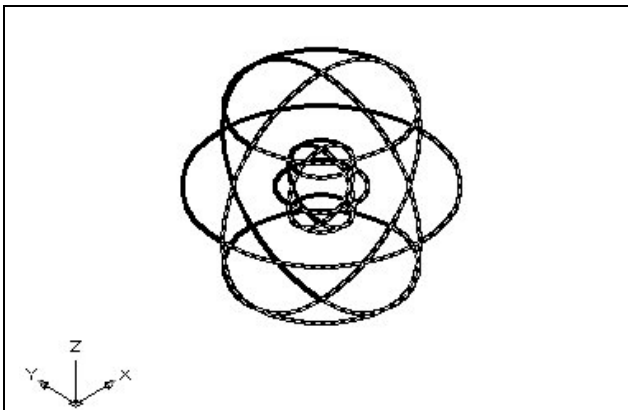


Although you can pick what side of the solid you wish to retain in the Slice function, you will find the new command easier to add to your toolbox if you just slice the solid and leave both sides present. Use the Erase tool on the Modify toolbar to remove the portion of the solid you no longer need. Select the Erase tool on the Modify toolbar, and select the entity on the right side as shown in figure 10.13. **Enter** and this will remove the hemisphere on the front side.

**Figure 10.13 – Pick the Front Hemisphere**

# Drawing a Solid Cylinder

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You need to add a 0.5 radius hole in the master solid, so you need to use the Cylinder command to create the feature.
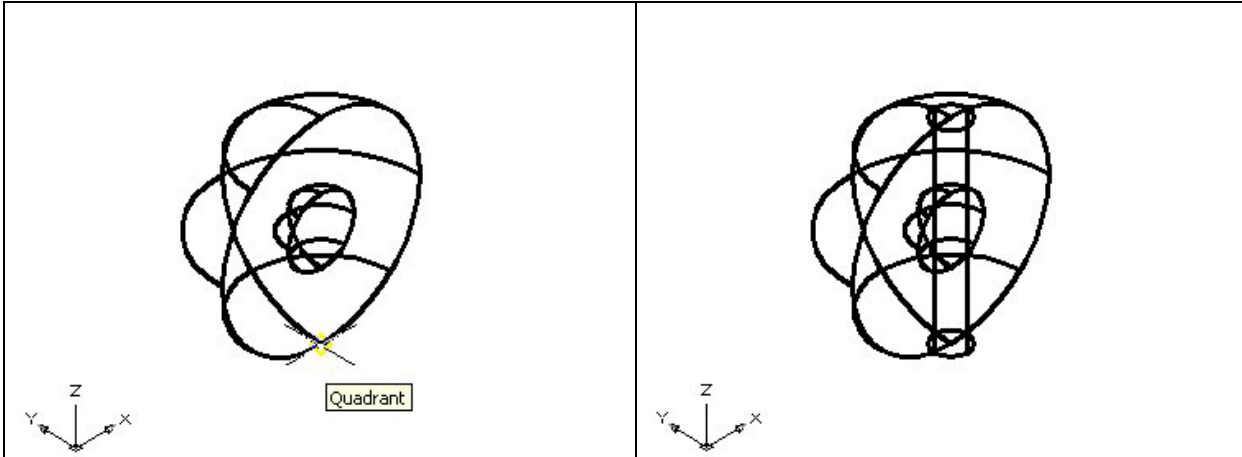


**Figure 10.14 – A Cylinder's Starting Point**    **Figure 10.15 – The Cylinder**

On the Solids toolbar pick the Cylinder tool. Specify the center point for the base of the cylinder as the lower Quadrant on the solid hemisphere as shown in Figure 10.14. Input the sketch's measurement of **0.5**. **Enter** to the next prompt, which is "specify the height of the cylinder". Key in **6.0** for the elevation of the can shaped solid and **Enter**. The cylinder will appears in the graphical display as shown in Figure 10.15.

# Subtracting a 3D Solid from Another Solid

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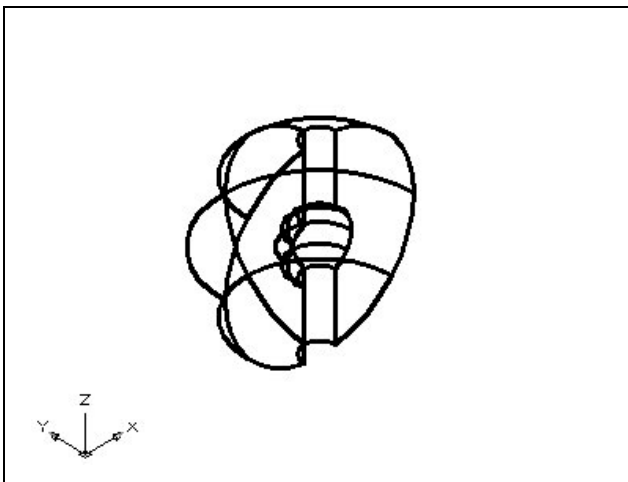


Then the AutoCAD program will prompt you with "**Select solids and regions to subtract from**". Select the larger master solid and the system will respond with "**1 found**". You need to Enter to proceed to the next step, which is "**Select solids and regions to subtract**". Pick the cylinder and the software application will counter with "**1 found**". Enter and a region exactly the shape of the cylinder is missing from the larger one as shown in Figure 10.16. Notice that in this view you can see the difference in the solid when you subtract the cylinder.

**Figure 10.16 – Subtract the Cylinder**

## Hide the 3D Solid

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In Figure 10.16, 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.
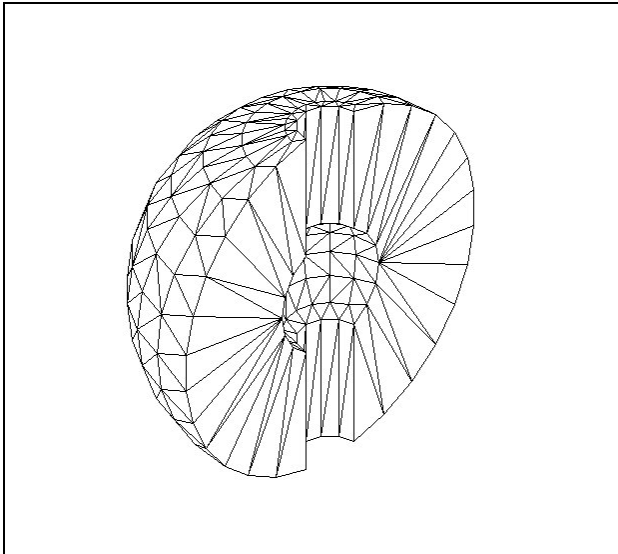


Select the Hidden tool on the Shade toolbar. This is a very simple command, because moments after picking the tool, the tessellation lines representing the back of the part are removed from view. If you wish to return the graphical display to the previous setup where all the tessellation lines are seen, then choose the 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 10.17 – Hide the Solid**

## Moving the Solid to the Origin Point

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To move problem nine 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 10.18 at the lower left hand side of the problem nine, which will become the problem nine insertion point into an assembly drawing. At the "second point of displacement", type **0,0,0** and **Enter**. Problem nine will move to the new origin point. There is more than one way to do many actions in AutoCAD, but the end 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 03-09 \* - 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.**
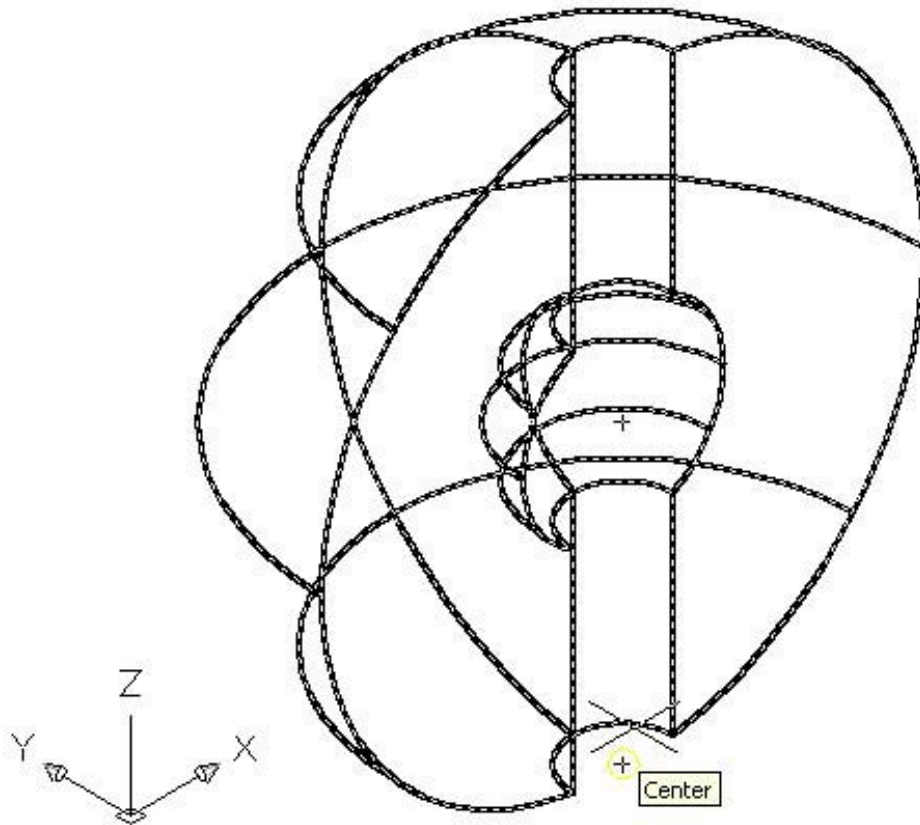
**Figure 10.18 – Moving the Finished Solid to the Origin**

# Saving the Solid Problem

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To save Problem 9 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 "Problem 9" and press the Save button to save the drawing. (See Figure 10.19)
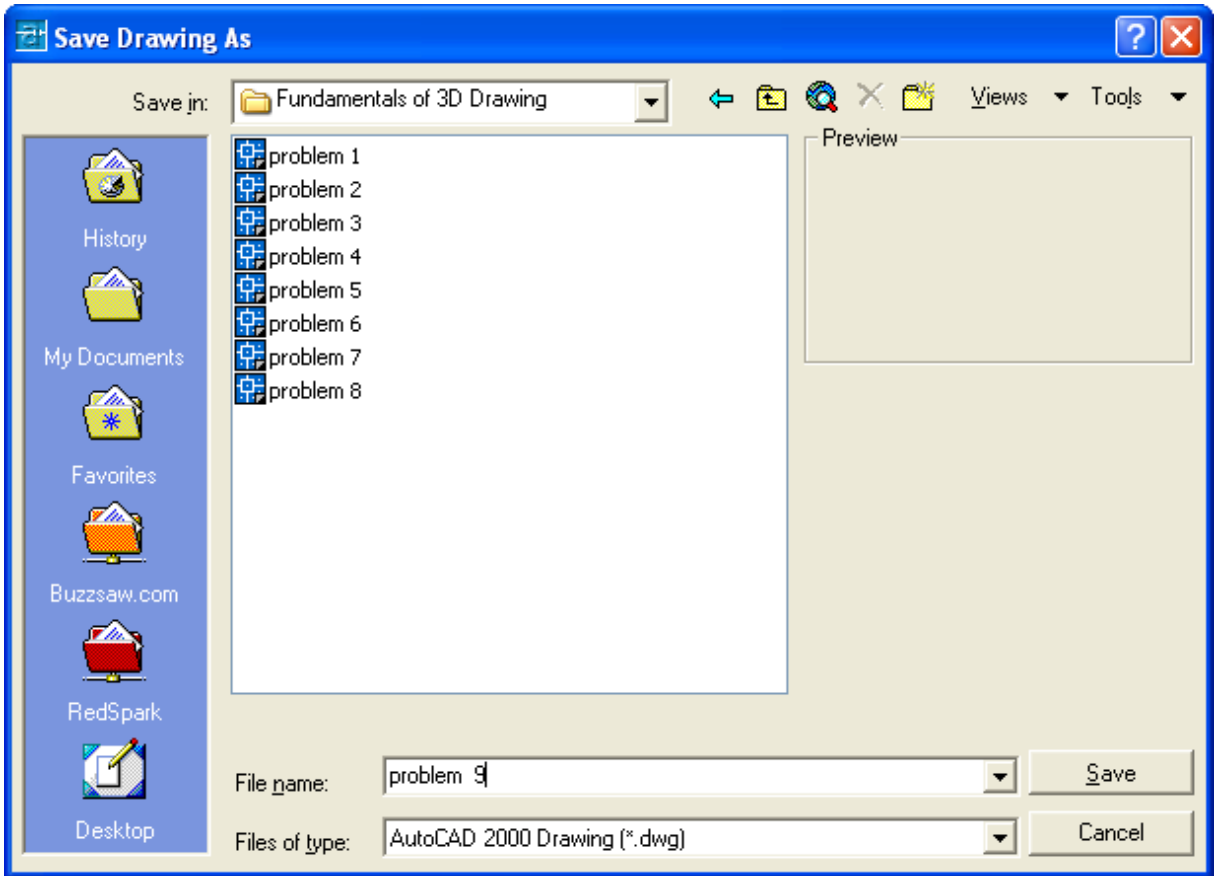
**Figure 10.19 – Saving Problem None**

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