Chapter 10

Solid Problem Nine

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

- **Sketch of Solid Problem Nine**
- **Starting a 3D part drawing**
- **Modifying UCS settings**
- **Drawing two 3D spheres**
- **Subtracting a 3D solid from another solid**
- **Creating a hemisphere using the Slice command**
- **Adding a cylinder**
- **Subtracting a 3D solid from another solid**
- **Changing the visual style**
- **Moving the solid to the origin point**
- **Saving the solid problem**

Sketch of Solid Problem Nine

You will start the ninth solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across the Y axis. In this exercise, you will create two spheres, removing the mass of the second from the first. After adding a cylinder, you will 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.

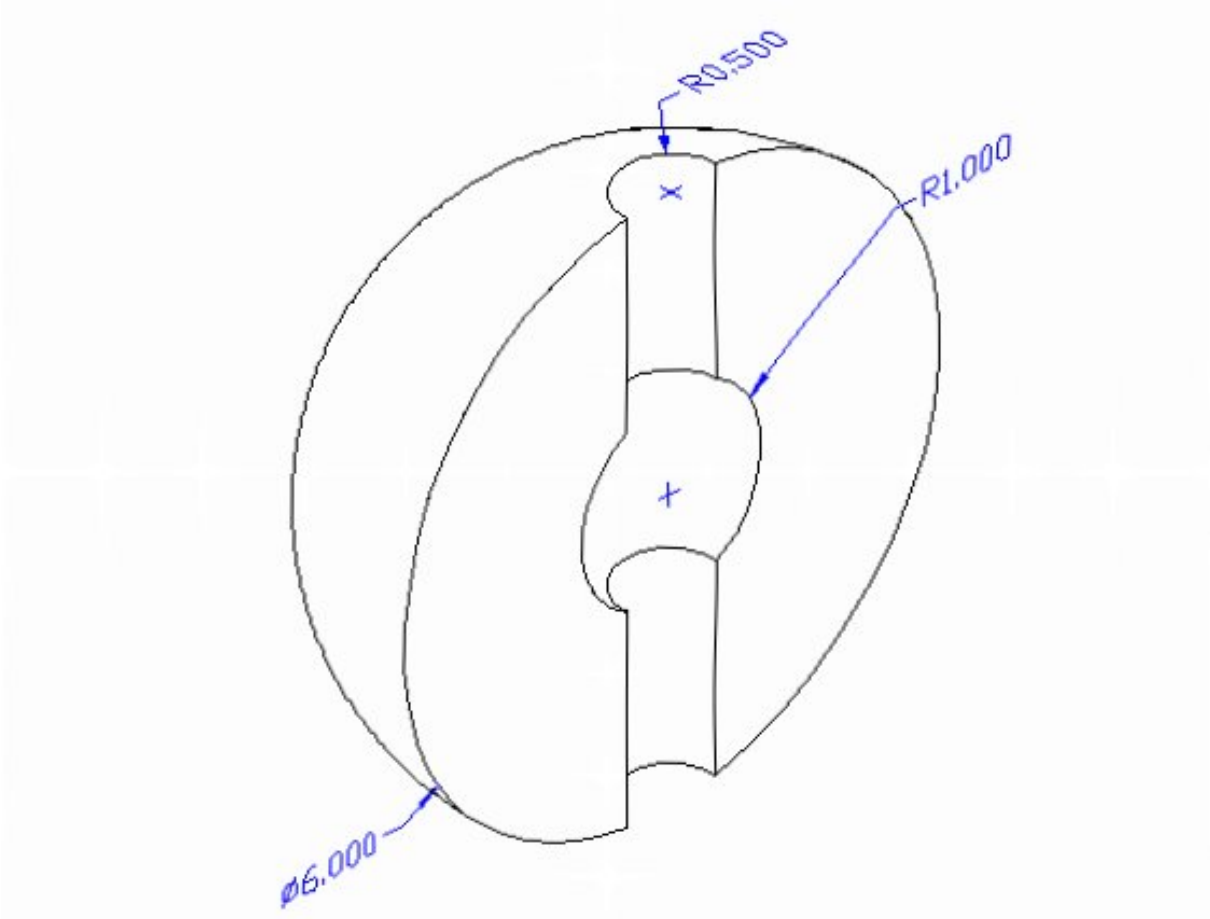


Figure 10.1 – Problem Nine Sketch

Remember, when you receive a sketch from a professional, quickly identify the different shapes and decide begin the drawing. Problem Nine has two shapes that you have not created in a problem yet, so pay close attention when placing the spheres. 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

You will draw Problem Nine 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.

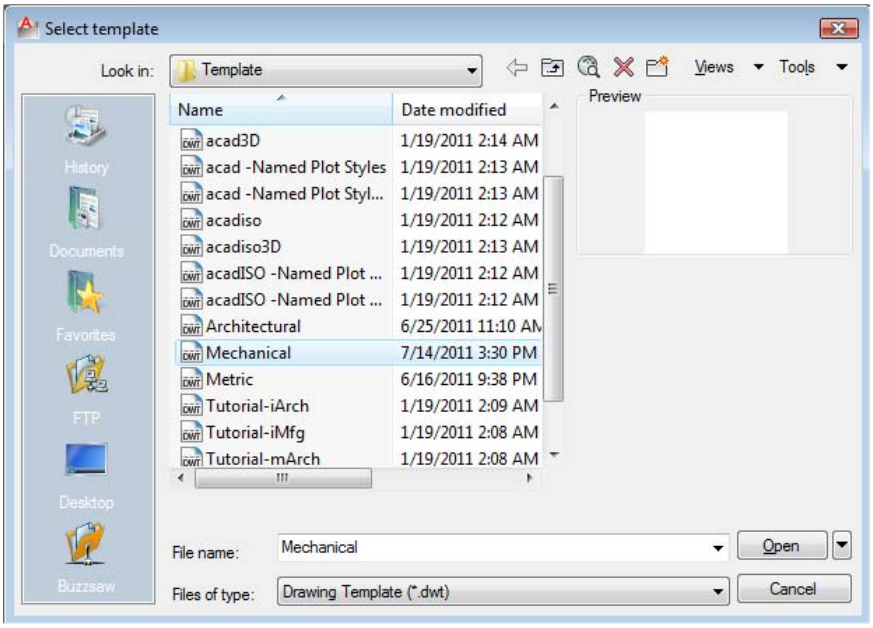


Figure 10.2 – Starting the drawing using the Mechanical template

Modifying UCS Settings

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 10.3) and select the "Show UCS Icon" option (Figure 10.4). You also have the choice of keeping the UCS displayed at the origin or turning it off completely.

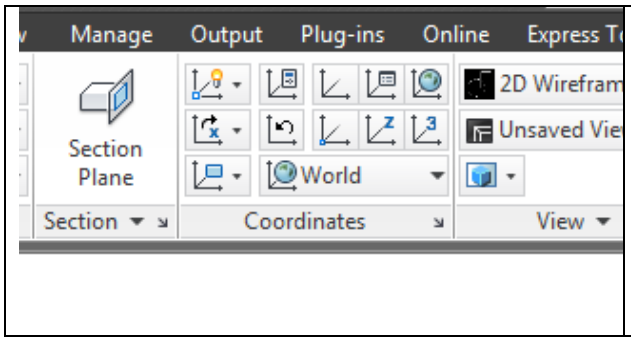


Figure 10.3 - The Coordinates panel

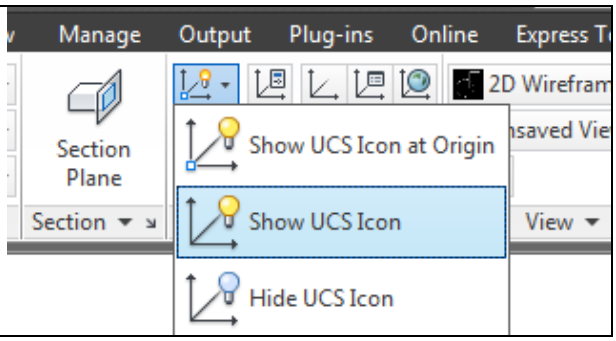


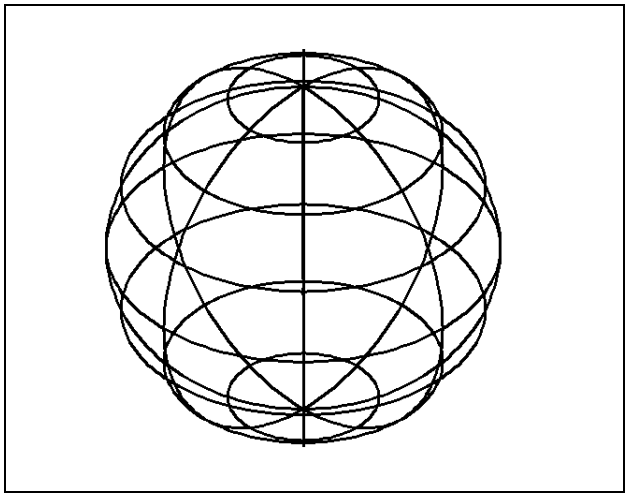
Figure 10.4 - Changing the UCS display settings

Drawing Two 3D Spheres

Occasionally you will need to design a spherical component. Luckily there is a Sphere command found by expanding the Box tool on the Modeling panel. Select the Isometric View on the View panel and then start the Sphere command. The command line information for the Sphere tool is very basic (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; you can also type "D" for the diameter option. For this sphere, you will just 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 place the sphere by typing "D" for diameter and then inputting **6.** Both spheres will be the same size.

Figure 10.6 – The first sphere

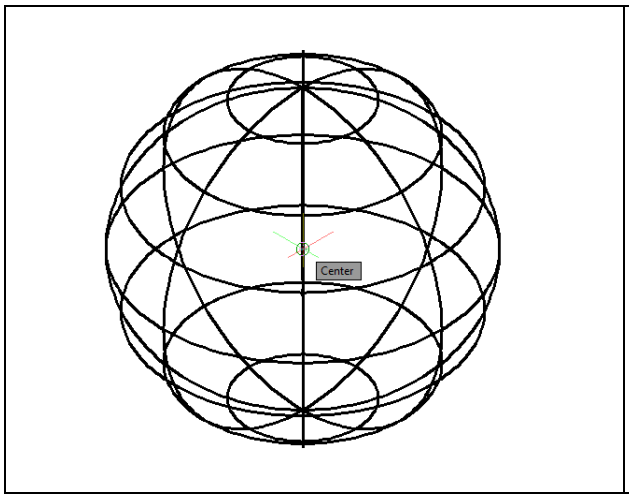


Figure 10.7 – The center of the second sphere

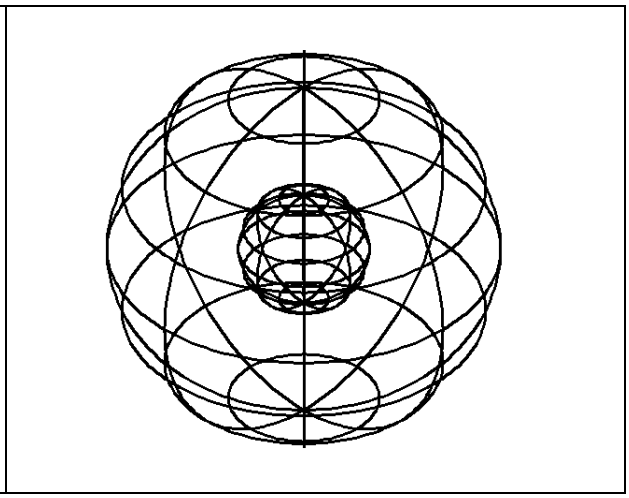


Figure 10.8 – The second sphere

Hit **ENTER** to repeat the Sphere command. Select the center Osnap of the first sphere as the center of the second sphere as shown in Figure 10.7. The next prompt will request the radius of the sphere by default, so type **1** 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

To remove the small spherical solid 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 larger sphere 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**".

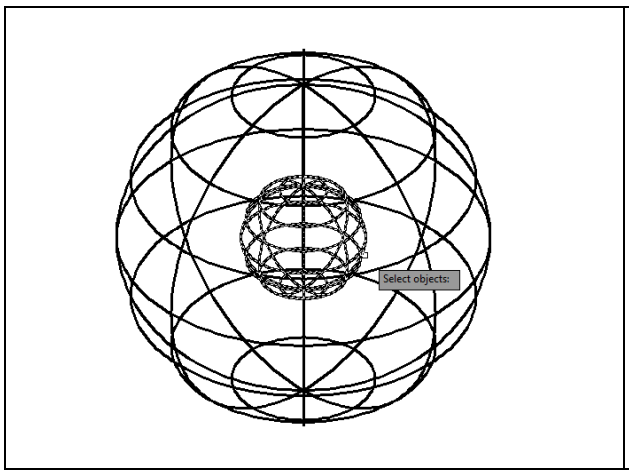


Figure 10.9 – Selecting the sphere to subtract

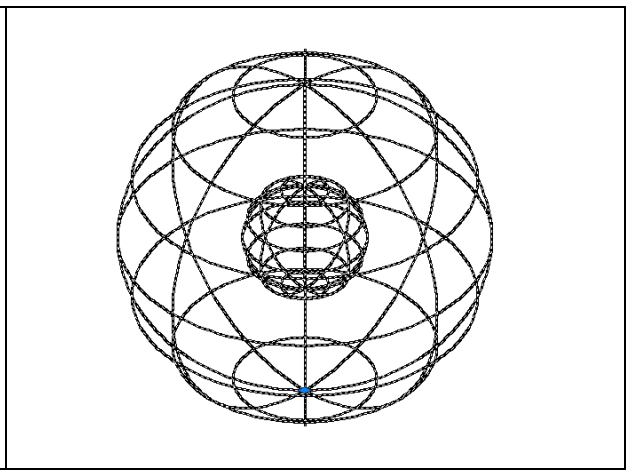


Figure 10.10 – The master solid, selected

Select the smaller sphere and the command line will respond with "**1 found**" (Figure 10.9). Hit **ENTER** and the shape of the smaller sphere will disappear from the master solid as shown in Figure 10.10.

Creating a Hemisphere with the Slice Command

The Slice function allows you to remove a part of a solid by selecting a line of the X or Y axis to use as a cutting plane. 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.

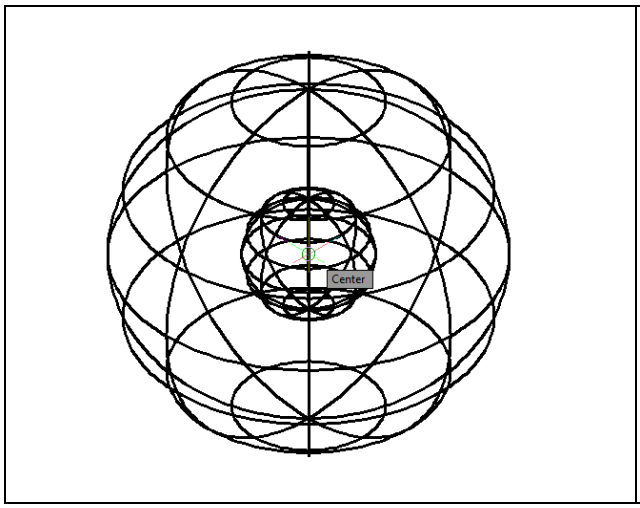


Figure 10.11 – Selecting the center Osnap

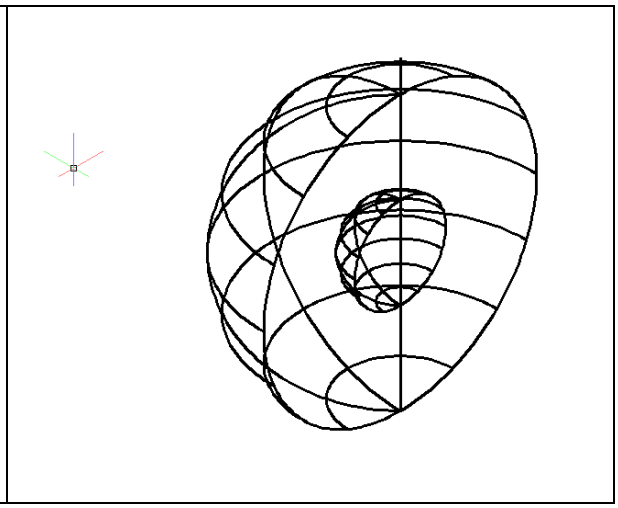


Figure 10.12 – Choosing the desired side

Select the Slice tool on the Solid Editing panel. First you must select the object you wish to slice, in this case the master solid. After selecting the sphere you will be prompted to specify the start point of the slicing plane, which is the center Osnap of the sphere (Figure 10.11). With the Ortho mode on, pull your cursor towards the bottom left-hand corner of the screen to choose the Y axis. Finally, you must specify a point on the desired side of the slicing plane to show which side of the part you wish to keep. In this case select a point in the upper left-hand corner of the screen to keep the left half of the sphere (Figure 10.12). You could also choose to keep both sides by just hitting ENTER, but for this problem you would have to then erase the half on the right that remained.

Drawing a Solid Cylinder

You need to add a 0.5 radius hole through the master solid. You will use the Cylinder command to create the feature.

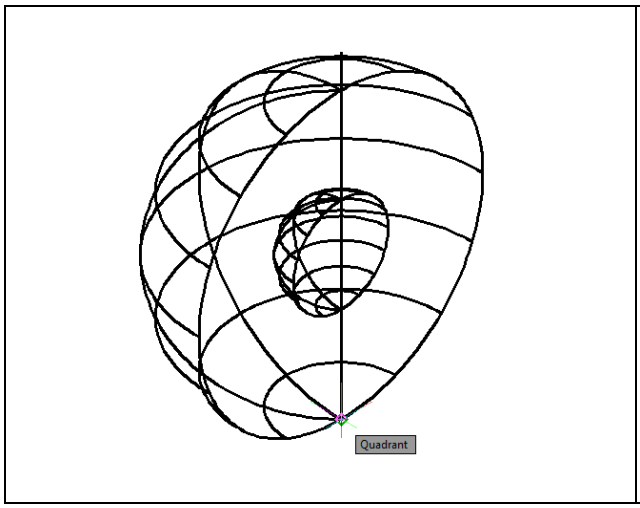


Figure 10.14 –The quadrant Osnap

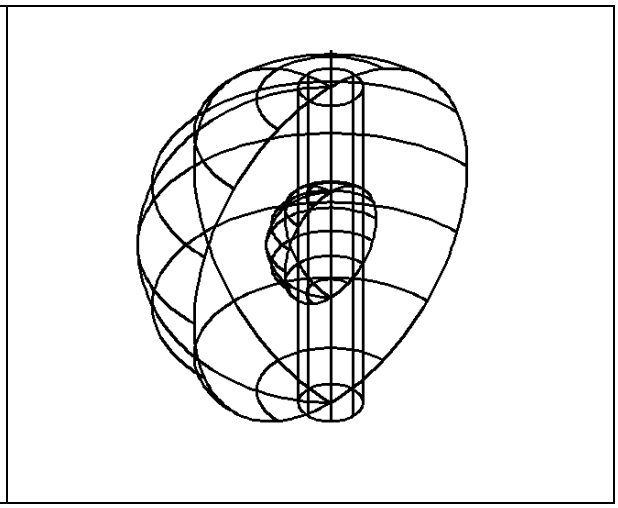
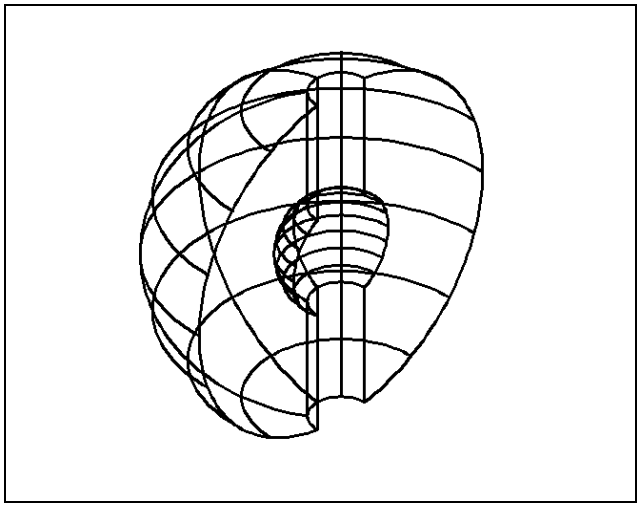


Figure 10.15 – The cylinder

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 south quadrant Osnap of the outer perimeter of the master solid (Figure 10.14). Type "**D"** for diameter and hit **ENTER** to input the sketch's measurement of **1.0**. Hit **ENTER** to go to the next prompt, which is to "specify the height of the cylinder". Type **6** for the height of the cylinder and hit **ENTER**. The cylinder will appear as shown in Figure 10.15.

Subtracting a 3D Solid from Another Solid

To remove the cylinder from the master solid, select the Subtract tool on the Solid Editing panel.

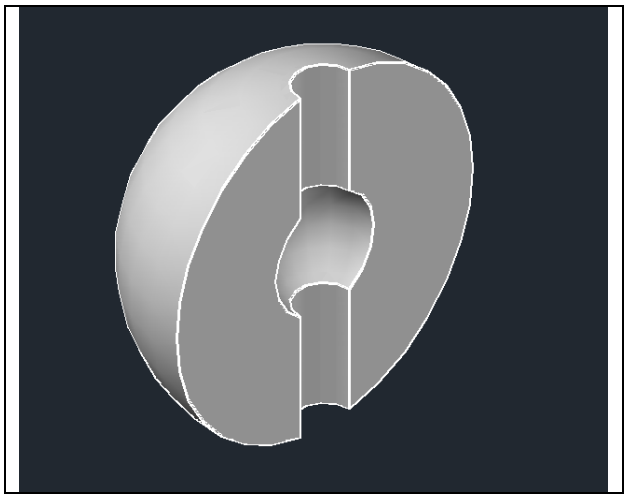


The command line will prompt you to "**Select solids and regions to subtract from**". Select the larger sphere and the system will respond with "**1 found**". Hit **ENTER** to proceed to the next step, which is to "**Select solids and regions to subtract**". Select the cylinder and the command line will respond with "**1 found**". Hit **ENTER** and the shape of the cylinder will disappear from the master solid as shown in Figure 10.16.

Figure 10.16 – Subtracting the cylinder

Changing the Visual Style

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 an actual solid part, you would not be able to see through the solid unless the material was transparent. You can use a different visual style to see a more realistic view of the part.



Select the Shaded with Edges visual style from the Visual Style panel under the View tab. The part will appear rendered with a metallic sheen and highlighted edges as shown in Figure 10.17. 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 10.17 – The Shaded with Edges visual style

Moving the Solid to the Origin Point

To move Problem Nine 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 at the bottom of the part as shown in Figure 10.18, 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. Problem Nine 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.

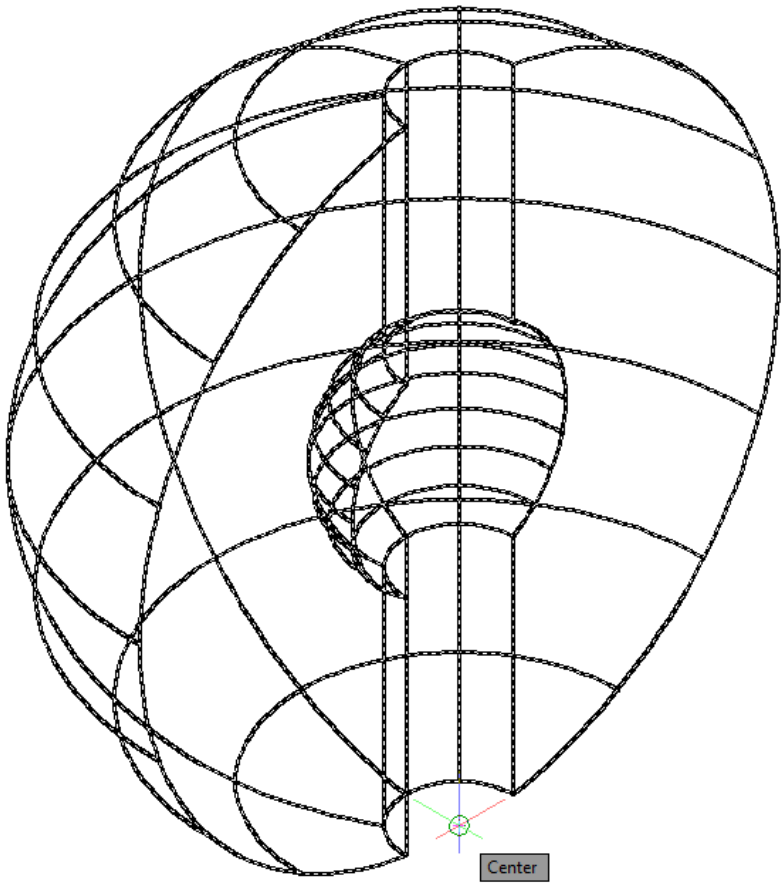


Figure 10.18 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Problem 9 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 "Problem 9" in the File Name textbox and hit the Save button to save the drawing (Figure 10.19).

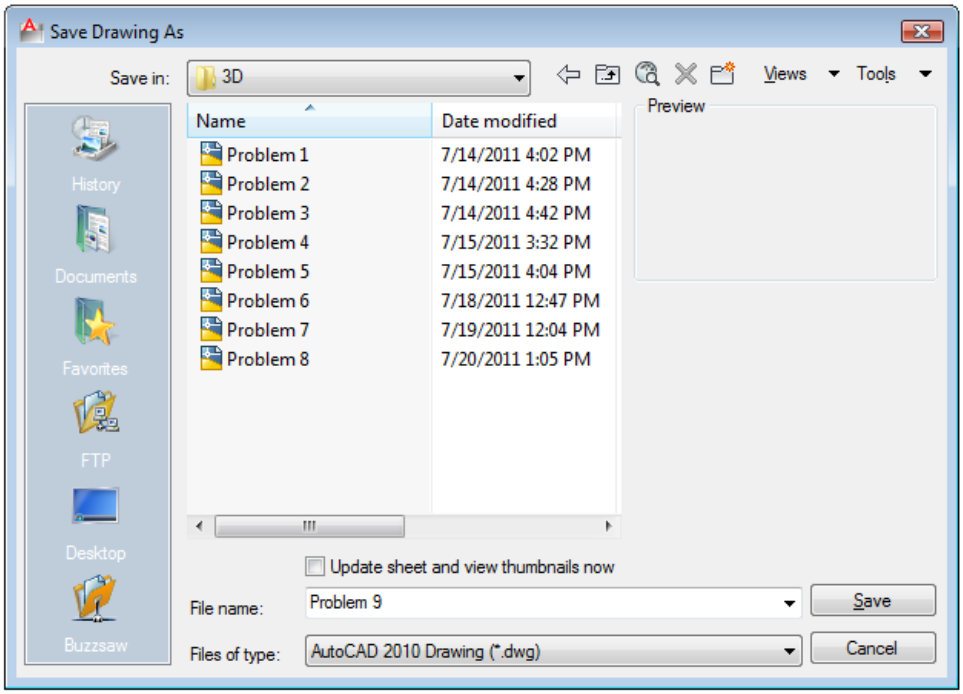


Figure 10.19 – Saving Problem Nine

*** 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 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.**