C h a p t e r

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 Spheres
5. Subtracting a Solid from Another Solid
6. Creating a Hemisphere Using the Slice Command
7. Adding a Cylinder
8. Subtracting a Solid from Another Solid
9. Hiding 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 use is to draw full solids and then subtract detail as necessary. 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, 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.
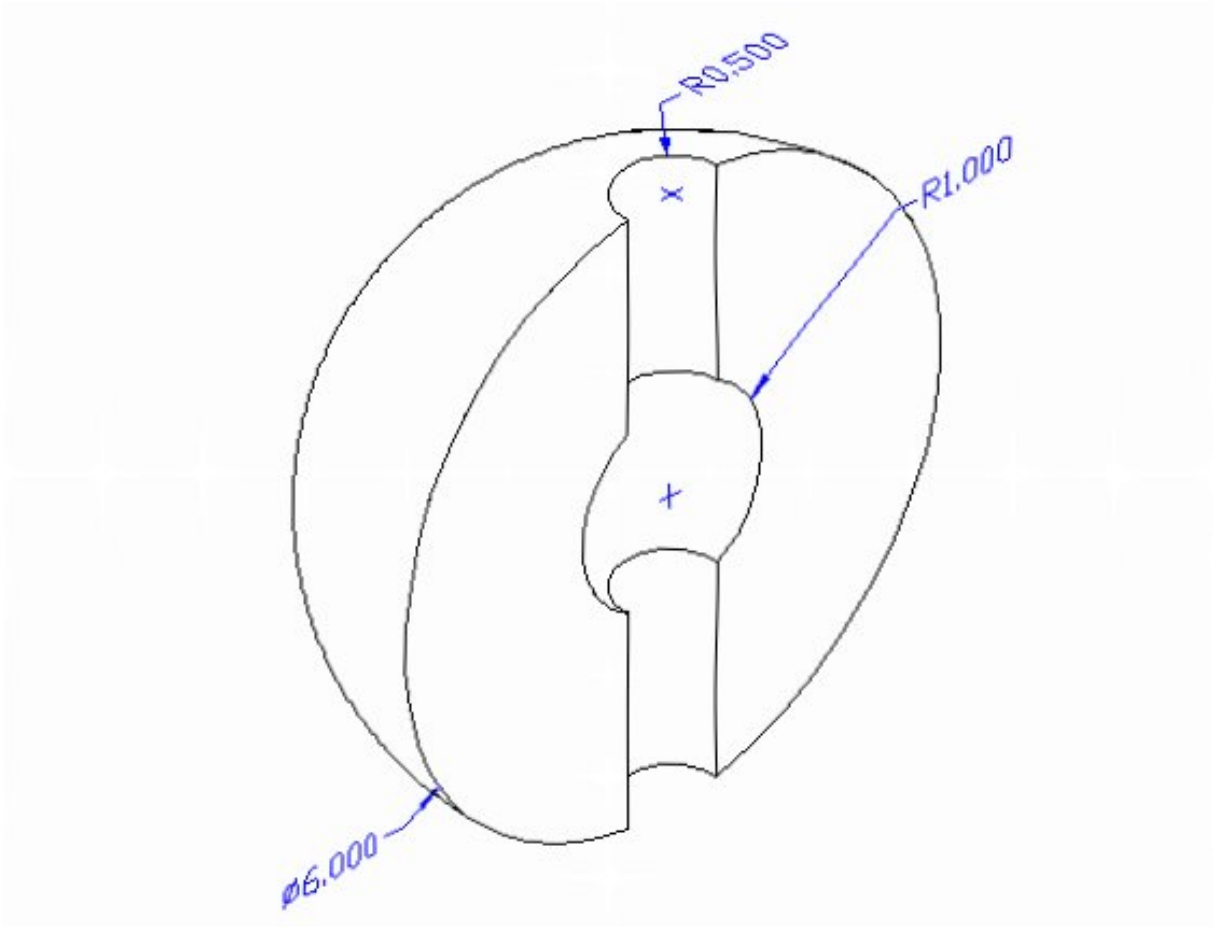

Figure 10.1 – Problem Nine Sketch

Remember, when you receive a sketch from a professional, you need to practice to quickly identify the different shapes and decide how to proceed with the solid exercise. 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 Eight, 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 8.2). 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.
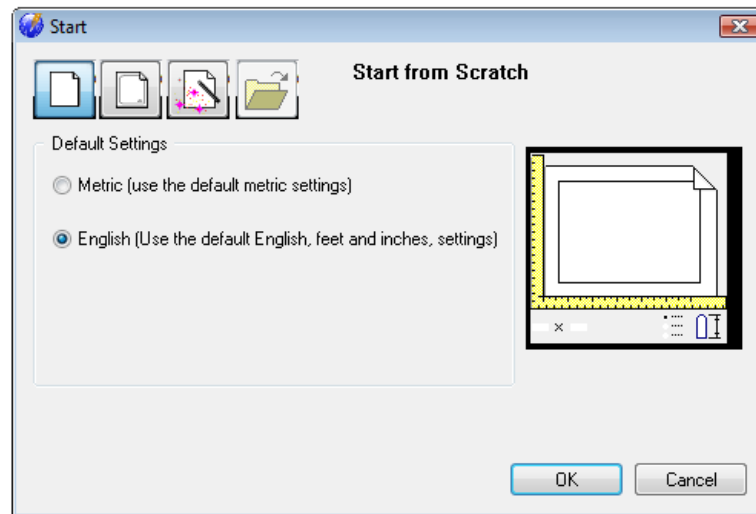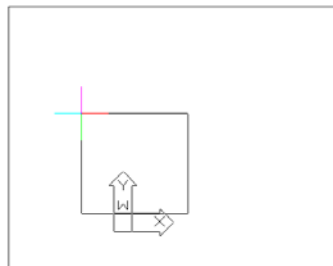

Figure 10.2 – Starting the Drawing

Modifying How the UCS Icon is Displayed

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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 10.3 - The UCS in the way

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.
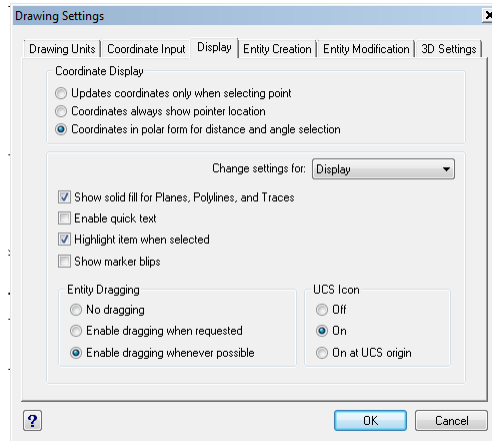
Figure 10.4 - Drawing Settings window, UCS Icon

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.

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Command : _SPHERE
Center of sphere:
Diameter/<Radius of sphere>: D
Diameter of sphere: 3
```

Figure 10.5 – The First Sphere on the Command Line
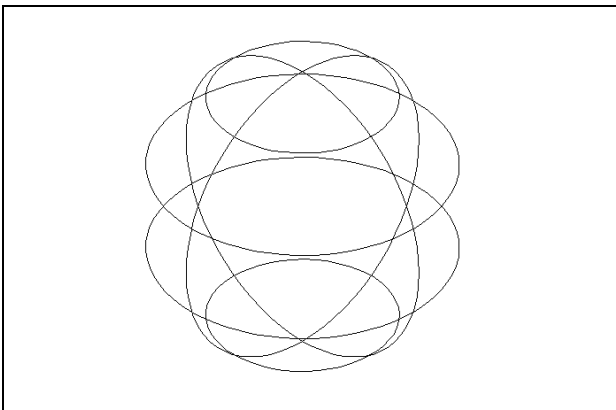
Pick anywhere in the middle of the graphical
display to specify the center of the sphere.
Like the Cylinder command, the next prompt
will inquire for the radius of the sphere by
default or type "D" for the diameter option.
For this sphere, type **6** and hit **ENTER.** A
round ball will appear in the center of the
display as shown in Figure 10.6. If you wish,
you can try another sphere where you can
type "R" for radius and then input **1.5.** Both
spheres will be the same size.

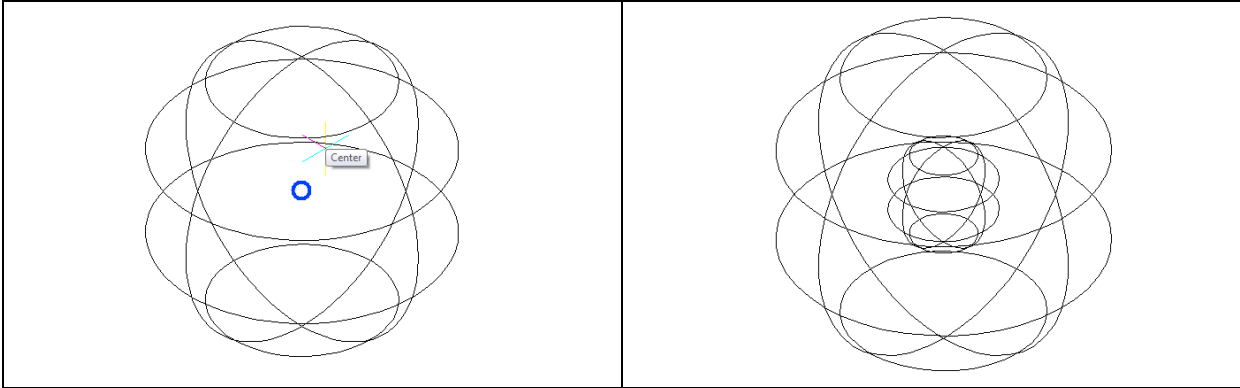**Figure 10.6 – Drawing the First Sphere**

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

Hit **ENTER** to repeat the Sphere command. Pick the Center Esnap tool on the Entity Snap toolbar and then place your cursor near a tessellation line as shown in Figure 10.7. The Center entity 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 type **0.5** and hit **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 ACIS object to subtract from:**".
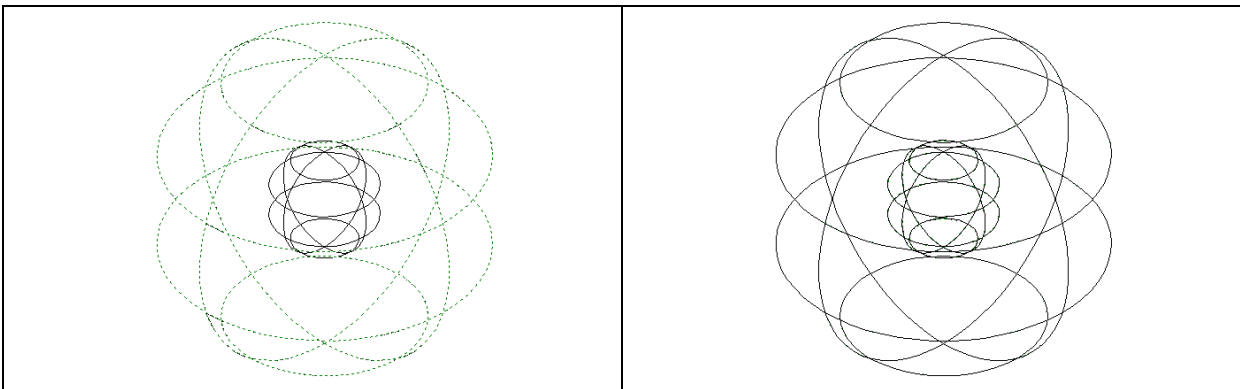

Figure 10.9 – Selecting the Outside Sphere   Figure 10.10 – Selecting the Inside Sphere

Select the larger master solid and the system will respond with "**Entities in set: 1**" as shown in Figure 10.9. Hit ENTER to proceed to the next step, which is to "**Select ACIS objects to subtract:**". Pick the smaller sphere and the software application will counter with "**Entities in set: 1**". Hit ENTER and a region exactly the shape of the small sphere will disappear 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.

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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 part of it from 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.
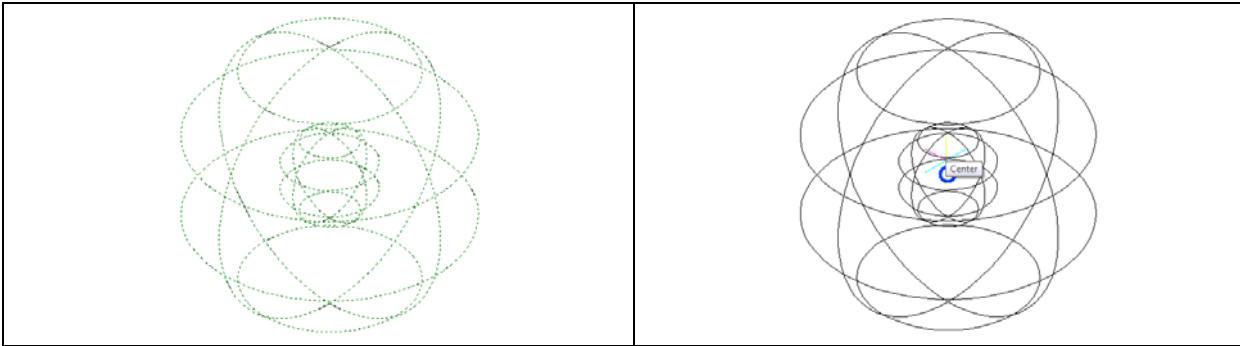

Figure 10.11 – Selecting the Master Solid   Figure 10.12 – Selecting the Center

Pick the Slice tool on the Solids toolbar The first step in the procedure is to pick the solid. 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 Esnap to choose the center of the solid ball. Finally select a point behind the solid to specify that you want to keep the back half. Making the selection divides the Sphere.

Drawing a Solid Cylinder

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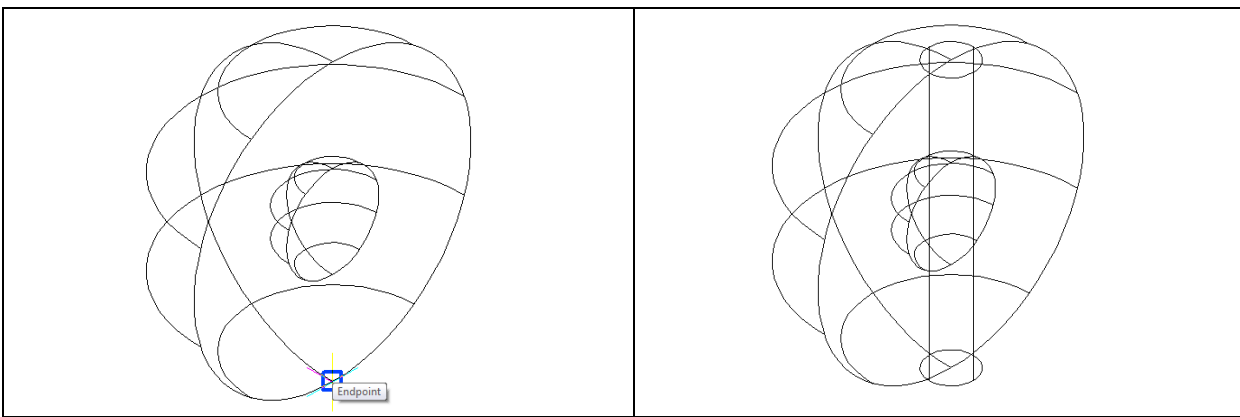

Figure 10.14 – A Cylinder's Starting Point   Figure 10.15 – The Cylinder

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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**. Hit **ENTER** to advance to the next prompt, which is to specify the "Center of second end/<Height of cylinder>:". Key in **6.0** for the elevation of the can shaped solid and hit **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
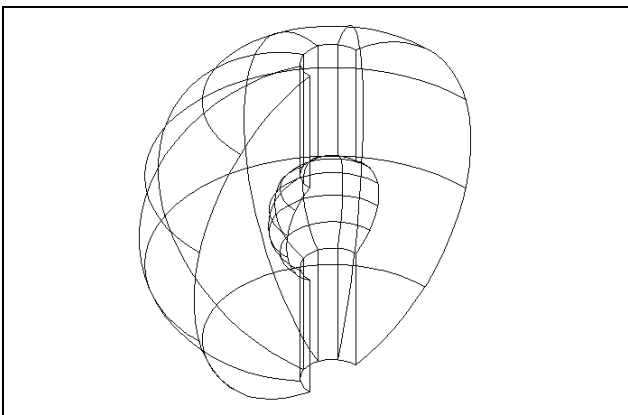
Select the larger master solid and the system will respond with "**Entities in set: 1**" as shown in Figure 10.9. Hit ENTER to proceed to the next step, which is to "**Select ACIS objects to subtract:**". Pick the cylinder and the software application will counter with "**Entities in set: 1**". Hit ENTER and a region exactly the shape of the cylinder will disappear from the larger one as shown in Figure 10.10.

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 Hide tool on the Shade toolbar. This is a very simple command because after picking the tool the tessellation lines representing the back of the part are removed from view as shown in Figure 10.17.
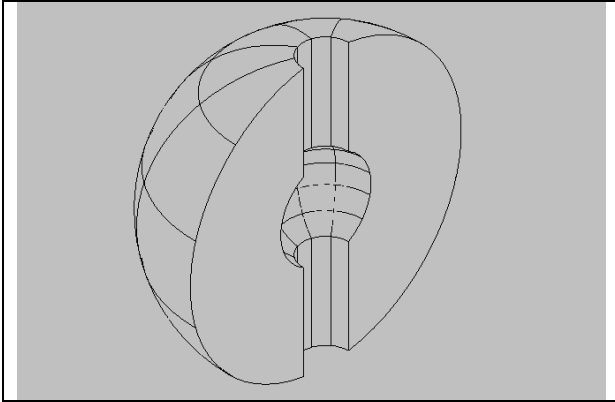
If you wish to return the graphical display to the previous setup in which all the tessellation lines can be seen, choose the first tool on the Shade toolbar called 2D Wireframe. By choosing the 2D Wireframe tool, the solid part and the UCS icon will return to the previous appearance. This command does 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 10.17 – Hide the Solid

Moving the Solid to the Origin Point

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To move Problem Seven 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 10.18 at the lower left-hand side of the Problem Seven, 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 Seven will move to the new origin point. Try using Zoom Extents if the part completely escapes your viewing area.

* World Class CAD Challenge 101-09 * - 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.
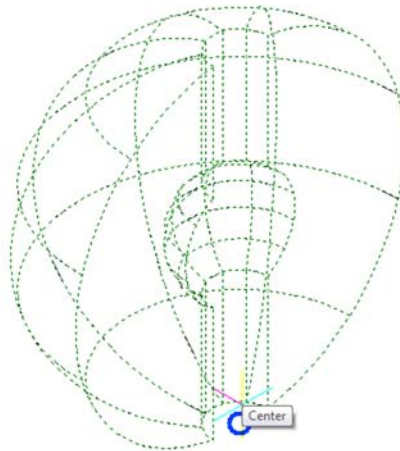

Figure 10.18 – Moving the Finished Solid to the Origin

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Saving the Solid Problem

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To save Problem Nine, 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 (Figure 10.18).
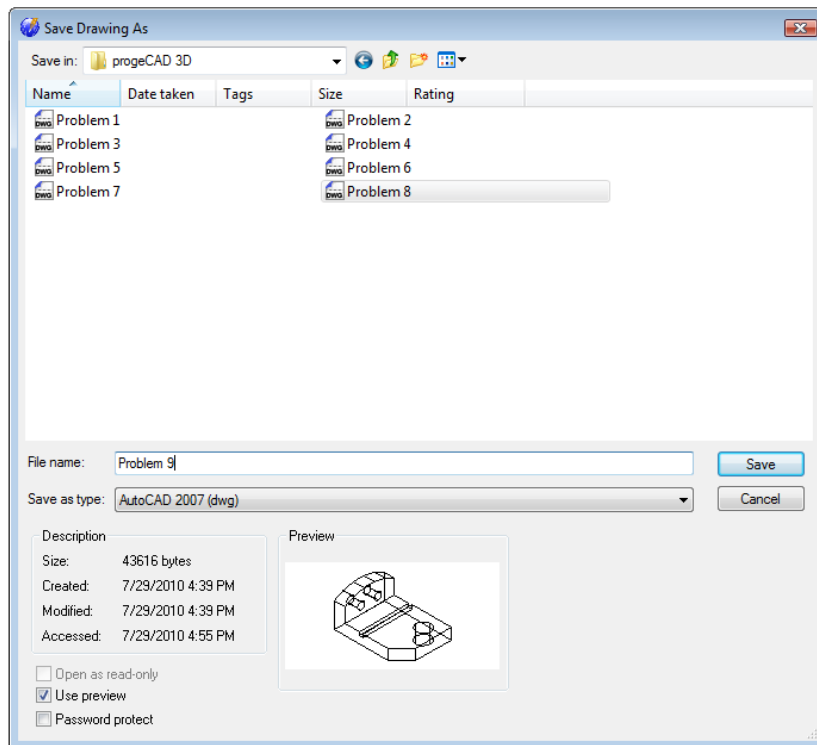

Figure 10.19 – Saving Problem None

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