



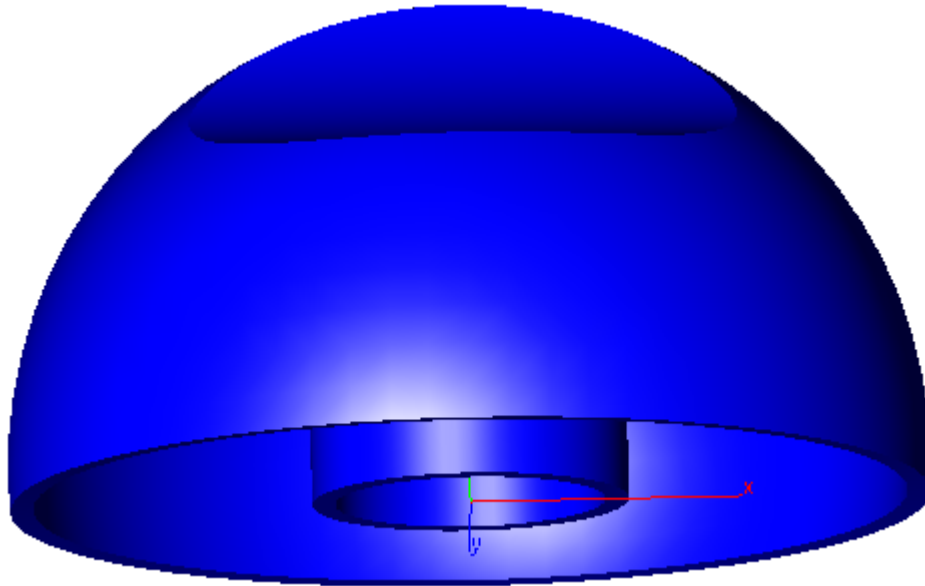
Curved Cap Tutorial

Using Cobalt[™], Xenon[™], Argon[™]

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3DCurvedCap1312

Curved Cap



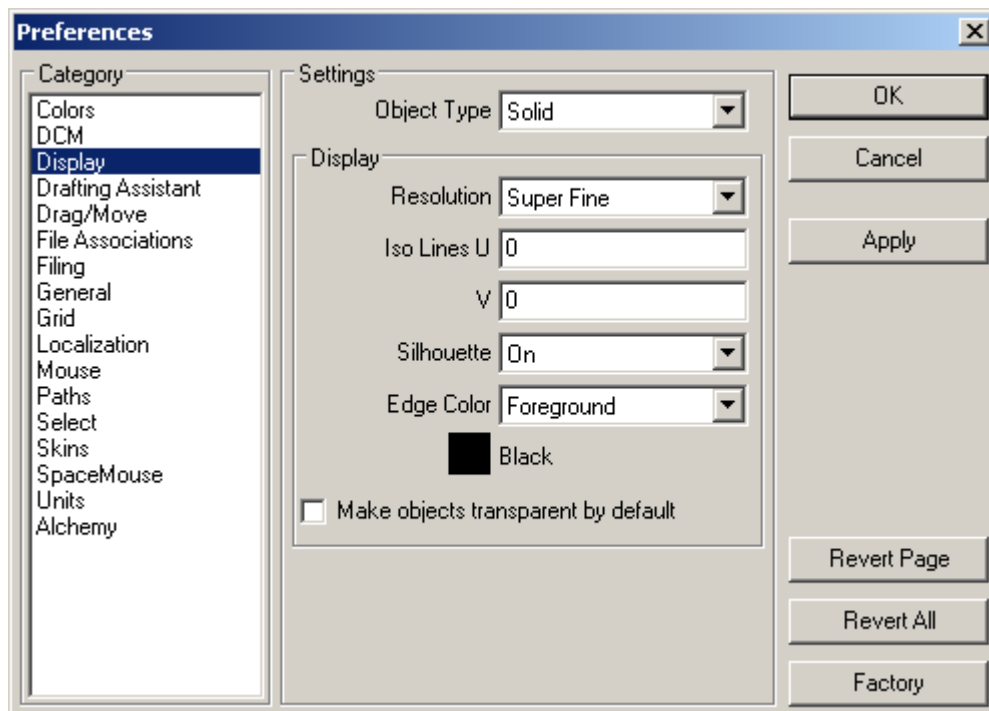
This tutorial describes how to use Ashlar-Vellum's 3D modeling products Argon, Xenon, or Cobalt to create a curved plastic cap. As with any modeling program, there are infinite ways in which to create the same thing. The intent of this tutorial is to give an overview of the types of tools these programs provide and a base knowledge to begin creating your own models. It is understood that the user is somewhat familiar with the interface and knows how to use certain aspects of the program, such as rotating a view, the data entry fields, and simple window commands. The following tutorial was created in Cobalt, but it can be done in Argon or Xenon.

Setting Up the Work Environment

When opening the program for the first time, all things are set to the default configuration. For this project, modify some of these settings to better suit the tutorial.

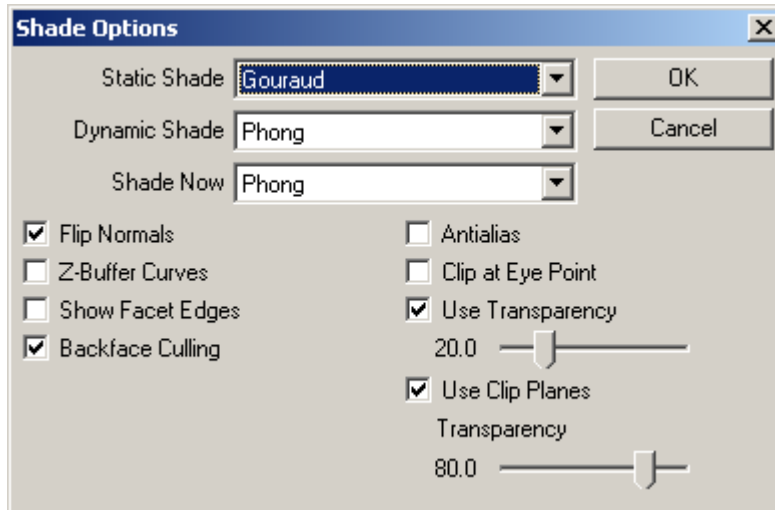
To change these settings go to **File>Preferences**.

The first thing to change is the Display resolution. Since the object being created is not very complex, set these all to the highest setting and don't worry about it slowing down the machine. If working on something with a higher detail, keep the settings at their defaults. Set the Curve, Surface and Solid resolution to be Super Fine.



The measures in this tutorial are given in inches, so make sure that the inches are chosen on the Units tab.

Also change the setting for the look of the model on the screen during the creation process. This is done by going to **View>Shade Options**. Change the Static Shade setting to be Gouraud. This shows all the surfaces of the model. Choose Phong for the Dynamic Shade and Shade Now.

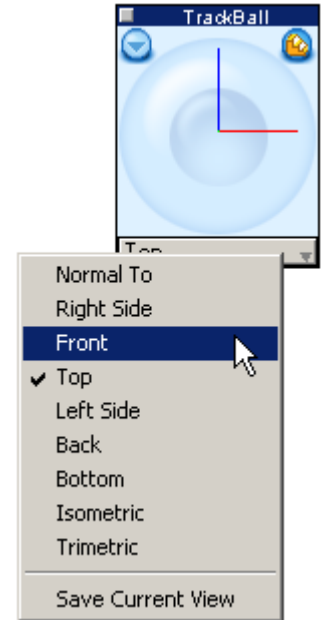


It is usually best to organize the palettes on the screen to maximize the work area. Close the Design Explorer (it will not be used for this tutorial) and move the Trackball and Hide-Show palette to the upper right corner to give more room for the model. Before beginning to create the curved cap, save the file as Curved Cap on the computer.

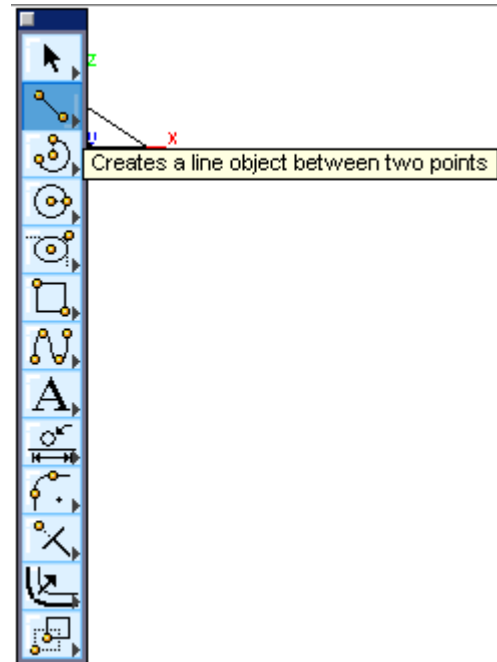
Creating the Curved Cap

The main body of the curved cap for a plastic spray container is simply a surface created by revolving and then thickened.

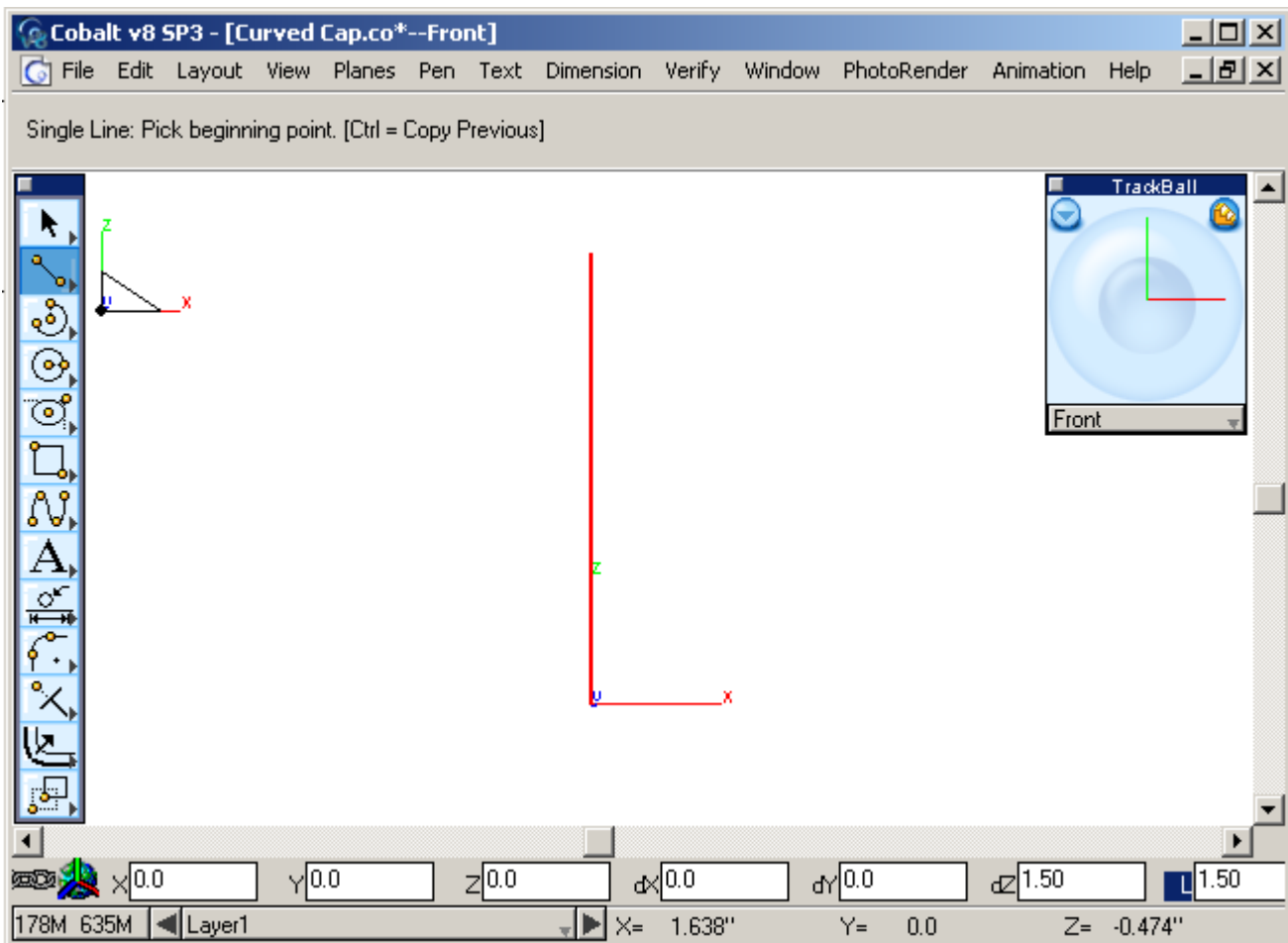
1. Go to Front view. If the Trackball is not displayed, activate it in the **Window>Trackball** menu. The top view is on by default, it is shown in the bottom of the Trackball. Click the gray arrowhead on the bottom right side of the Trackball to display the drop down menu and choose the Front view.



2. Choose the **Single Line** tool from the main tool palette.

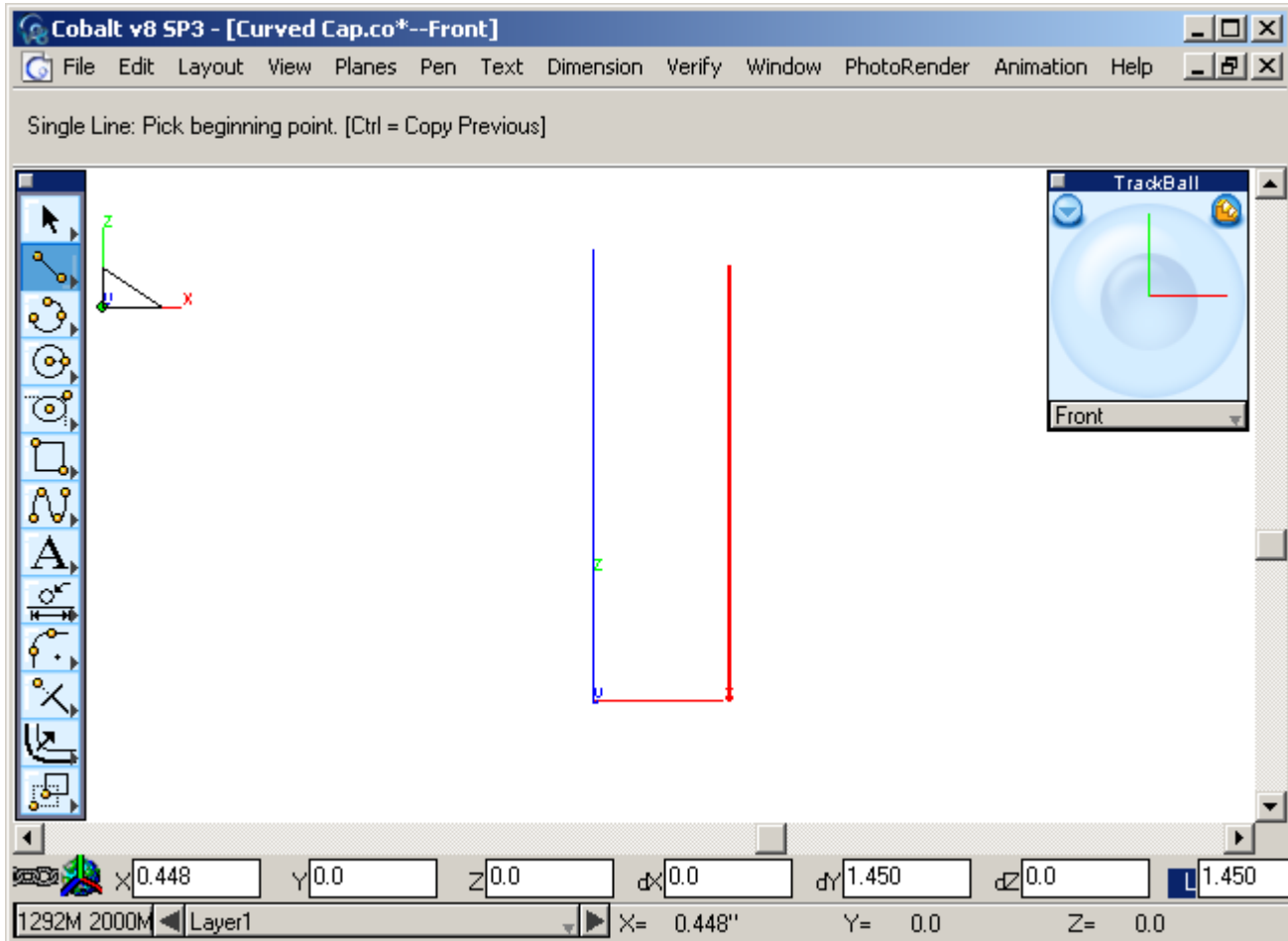


3. Draw a vertical line. First click in the Origin of the working area for the beginning point, drag the line up some distance and click again for end point of the line. The line appears to be red since it is selected for possible modifications.
4. At the bottom of the window, in the Status Line, type **1.5** in the **L** field and press ENTER key.
5. If the line goes off the screen, zoom in by scrolling back the middle mouse button or select **View>Zoom In**.

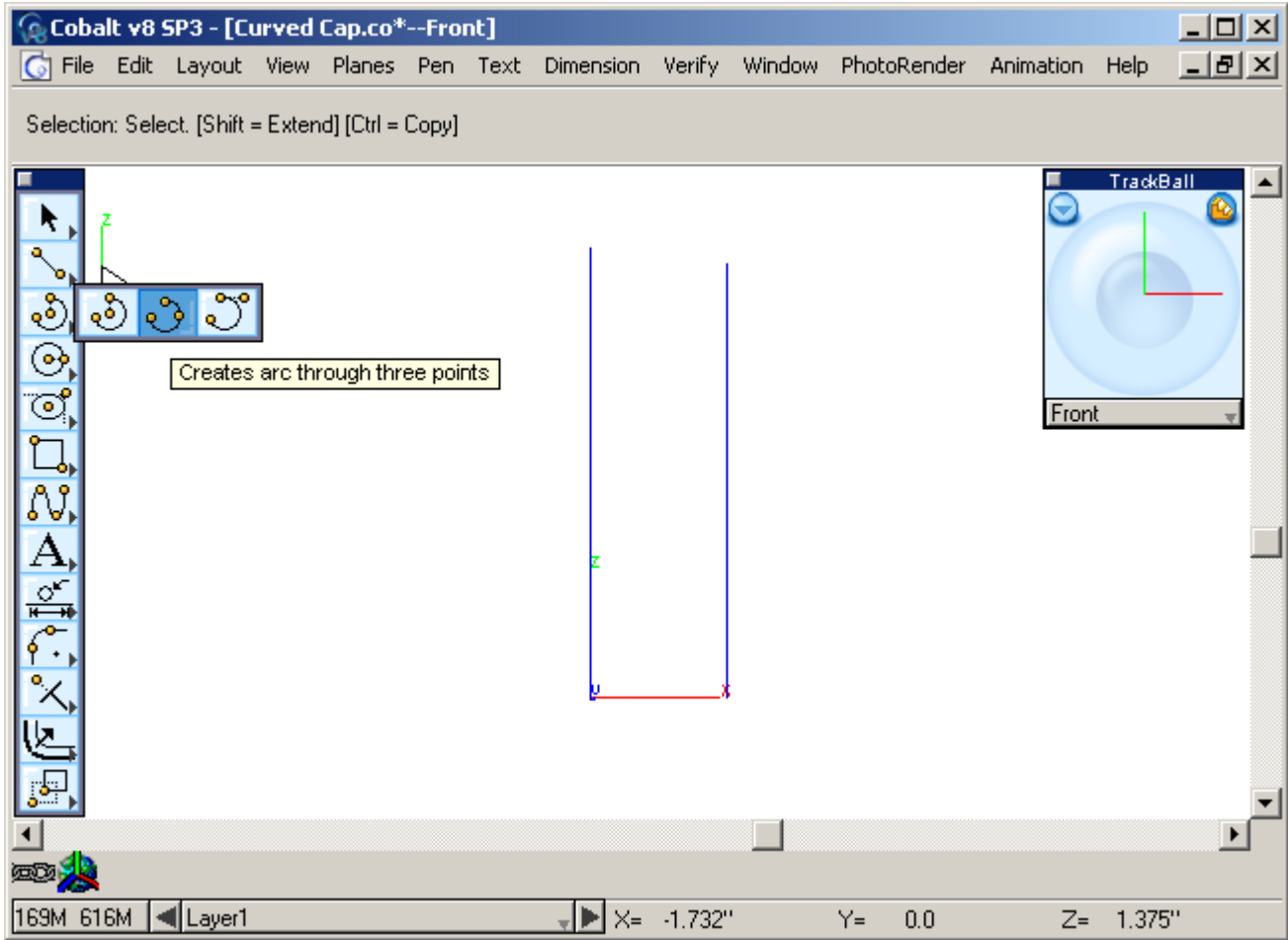


6. With the **Single Line** tool still active, move the pointer to the right from the origin for the distance about 0.45 along the x-axis (as you move the mouse its location coordinates are seen in the Location Bar in the very bottom of the window) and click for the beginning point of the second line. Drag upward and click for the end point.

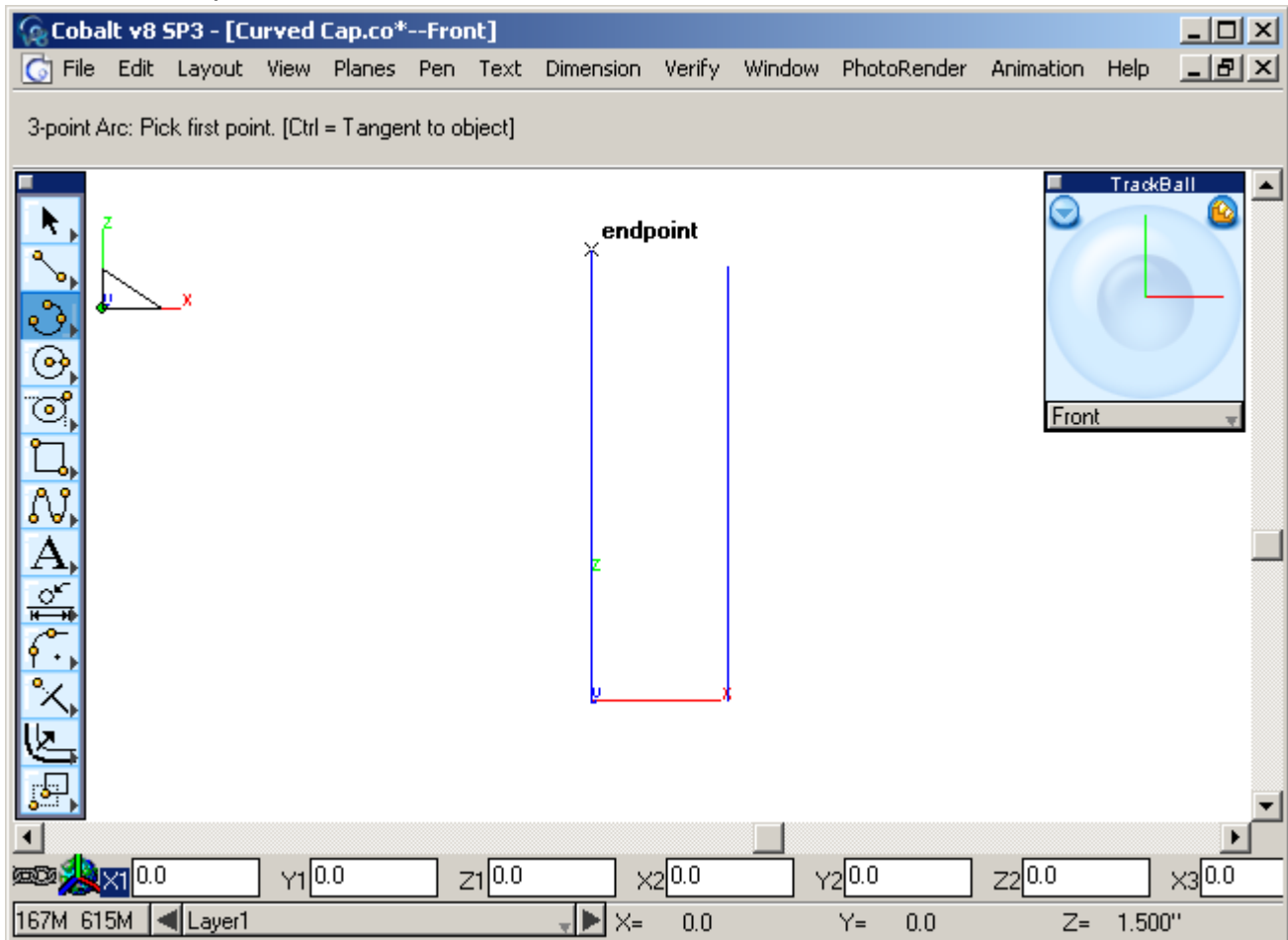
7. Type **1.45** in the L field in the Status Line. Press ENTER.



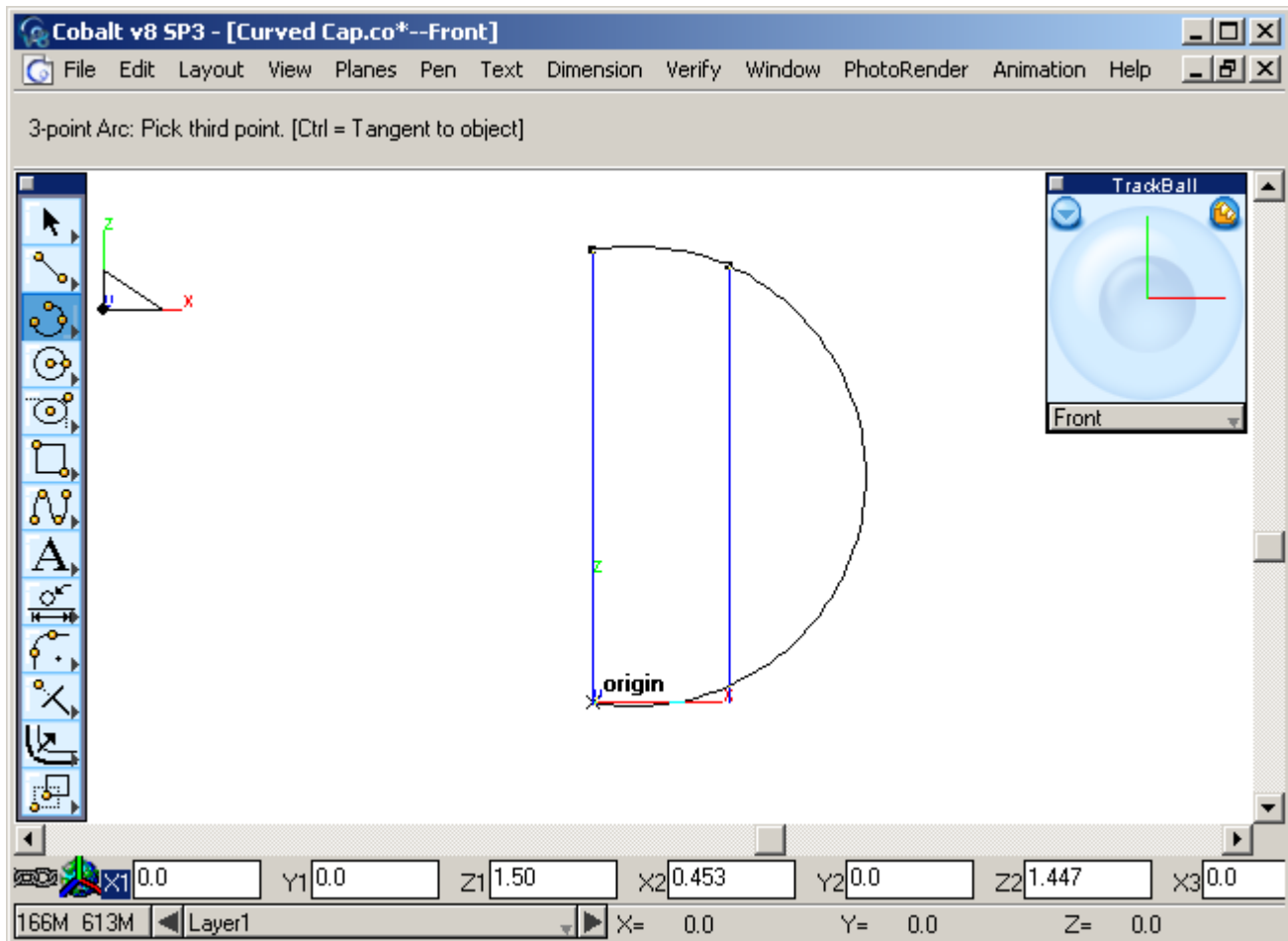
8. Create the curve for the top of the cap. Choose the **3-point Arc tool**.



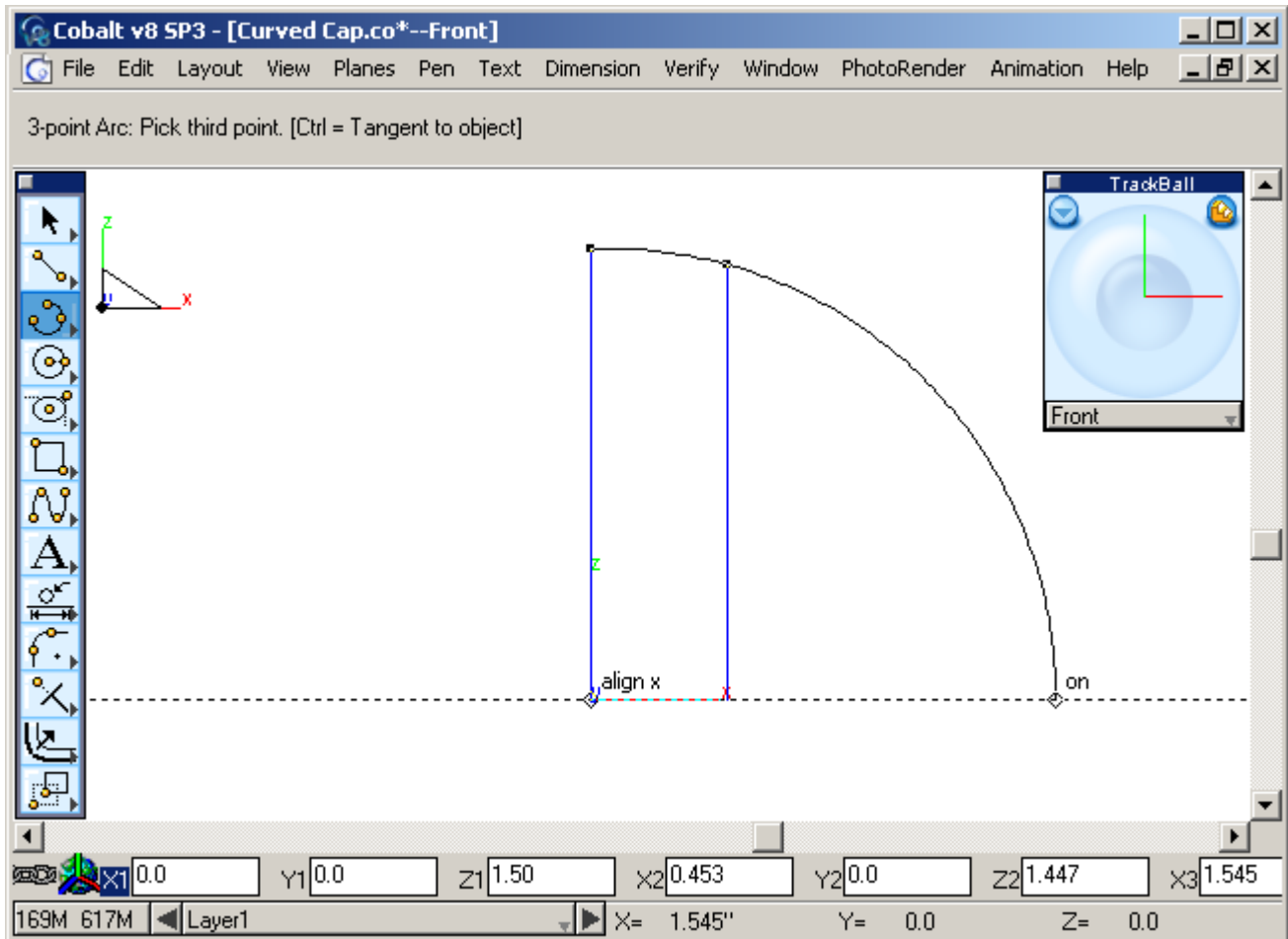
9. Begin drawing the arc by clicking the upper endpoint of the bigger line for the first point.



- Click in the upper endpoint of the second line for the second point. Move the pointer to the origin to activate the temporary construction line.

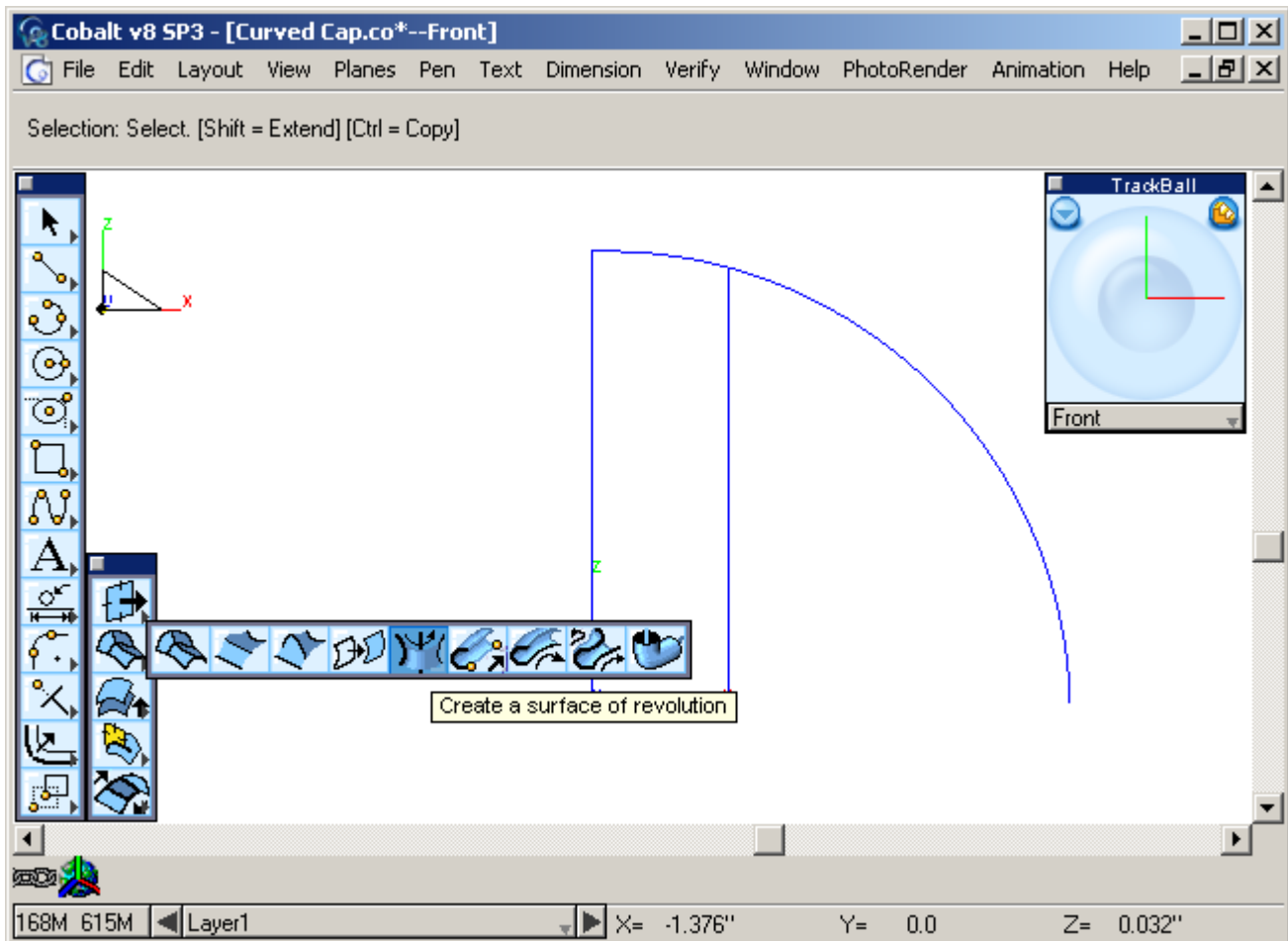


11. Move to the right along the x-axis and click for the third point.



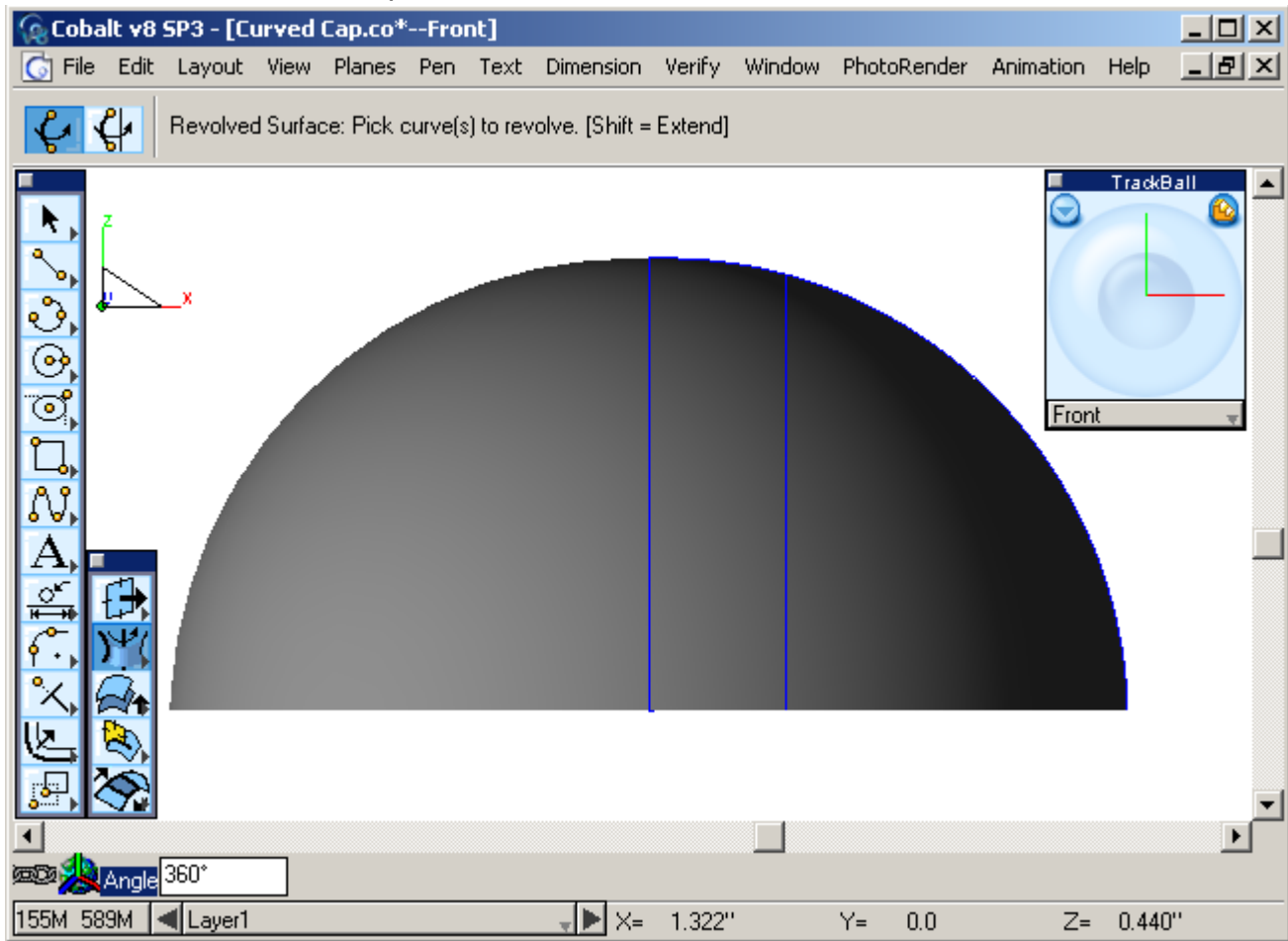
12. The arc appears. This will be the curve to revolve to create the upper part of the curved cap.

13. Create the shell dome. Choose the **Revolved Surface** tool from the **Surface Tools** palette.

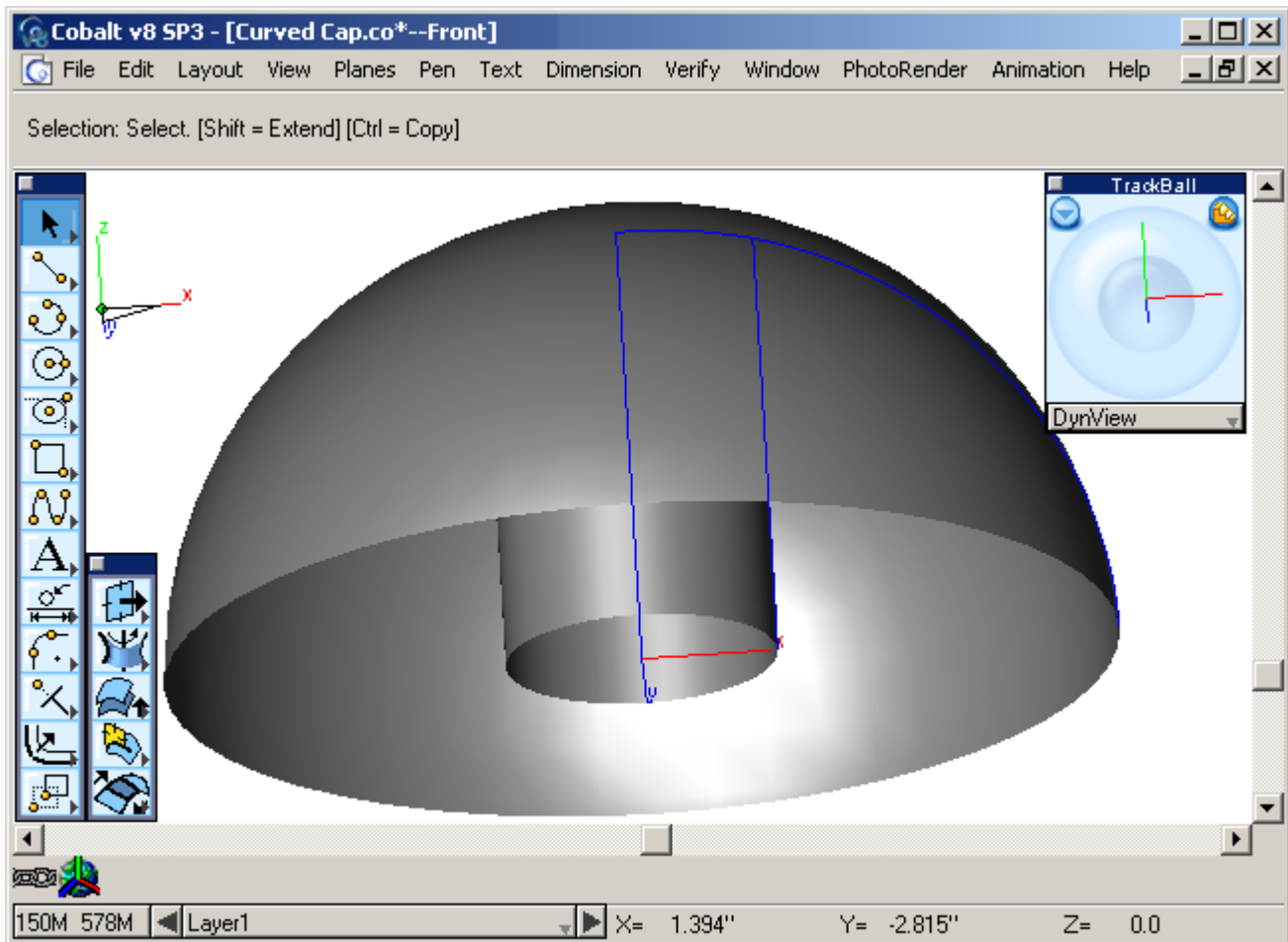


14. Select the arc for the curve to revolve.

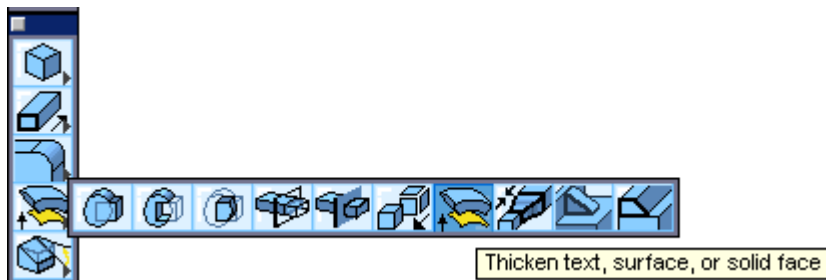
15. Click both endpoints of the longer line for the lathe axis. As a result you get a dome like in the picture below.



16. Now create the inner cylinder surface. With the **Revolve Surface** tool still active, click the line on the left to revolve and the endpoints of the central line for the axis of revolution. Use the Trackball to turn the dome shell to see what is beneath it.

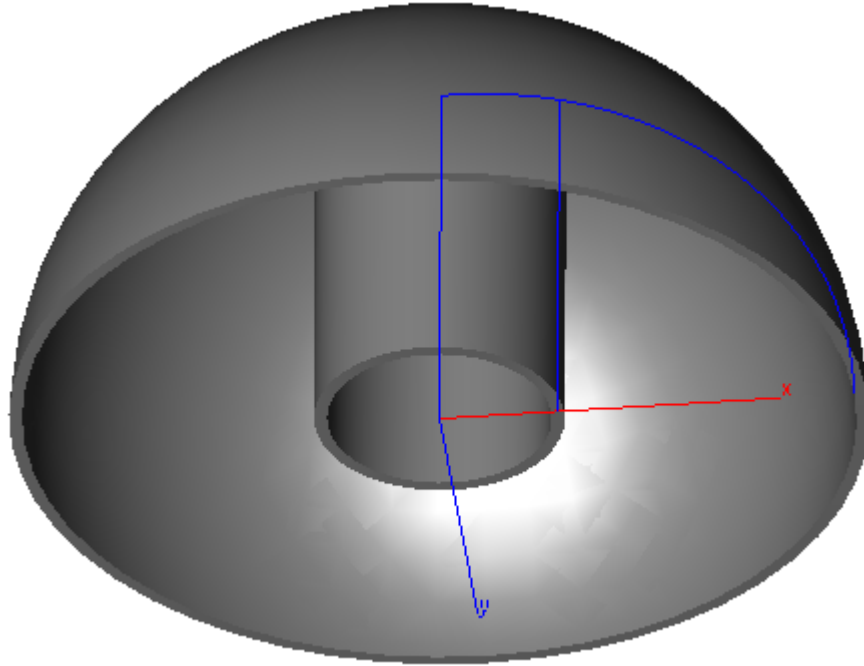


17. Thicken the dome-shell making a solid with the thickening direction pointed into inside the shell.
18. Select the **Thicken Solid** tool.

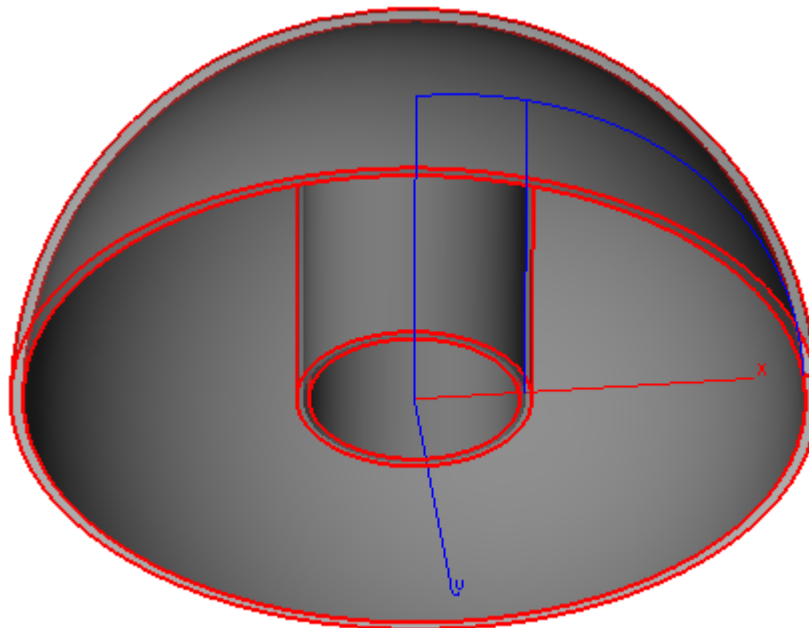
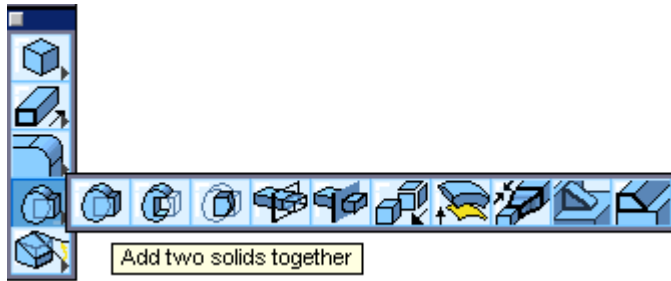


19. Insert **0.05** for the Thickness field in the Status Line. Tap the CTRL key (Windows) or the OPTION key (Macintosh) until the thickening direction is on the inside, if not already. Click the dome shell to thicken. Make sure that the thickening is done inside.

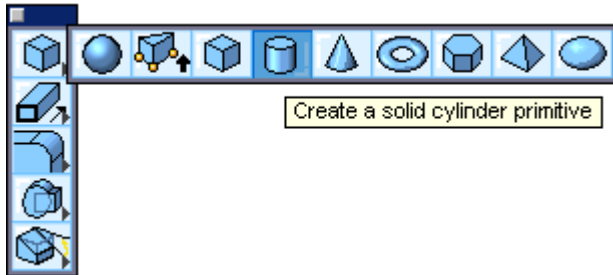
20. Repeat this for the inner cylinder. It should be thickened inside too. The result should look like the picture below.



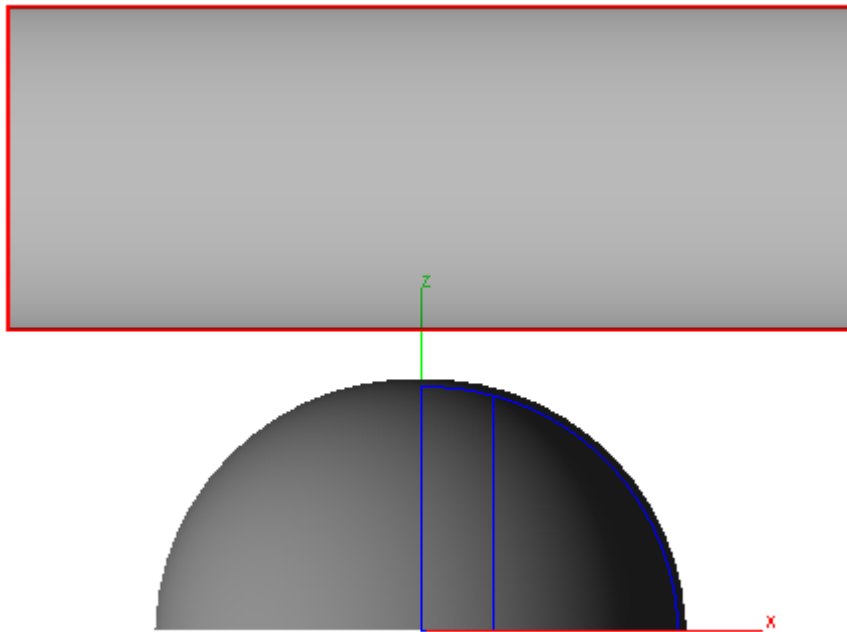
21. Merge the solid shell with the inner cylinder. Use the **Union Solid** tool.



22. Now it is necessary to build a solid cylinder whose side will create a recess on the cap. For this, choose the **Cylinder** tool from the **Solids** tool palette.

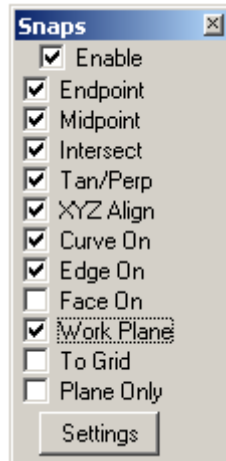


23. Click in the area on the left over the dome-shell for the starting point and then move the pointer along x-axis and click again on the right over the dome-shell for the end point of the cylinder. Enter **2** in the **D** field for the diameter in the Status Line in the bottom of the window.

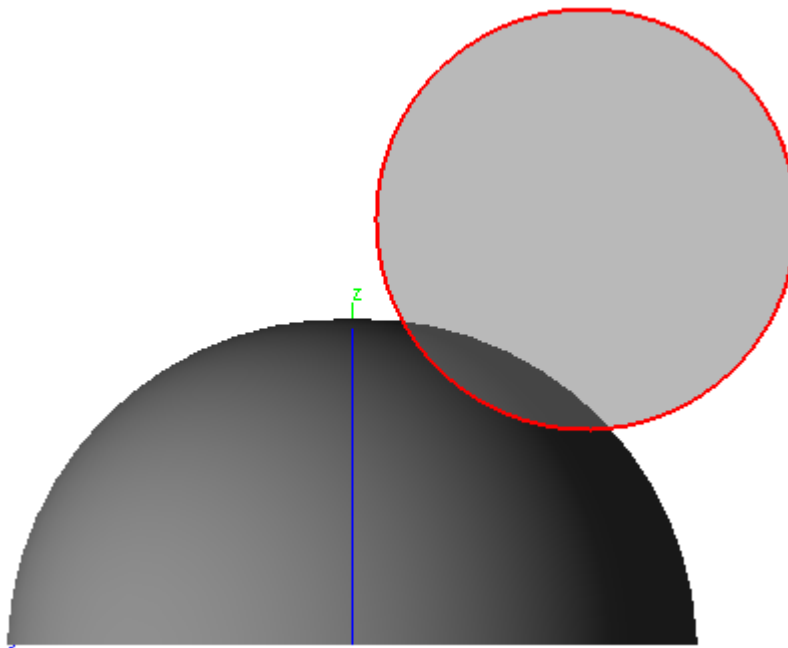


24. Choose the **Select tool** and Left Side view in the Trackball.

25. Go to **Window>Snaps** and check the Work Plane option.



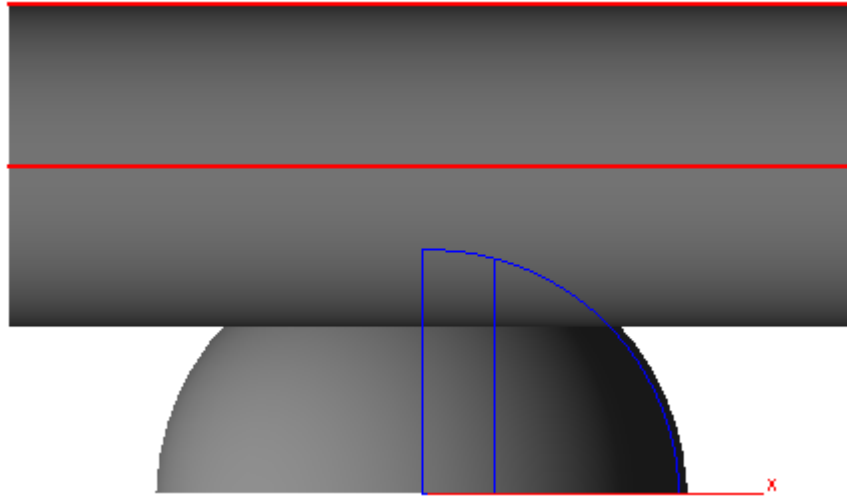
26. Grab the cylinder and relocate it according to the picture below.



27. Return to the Front view and uncheck the Work Plane option in the Snaps window.

28. Create a hollow cylinder that will be sized and placed identically to the one we already have. Choose **Single Line** tool. Draw two lines: first one along the center of the cylinder, from one vertex to another; the second; along the upper

edge of the cylinder from one quadrant to another. The lines are in red in the picture below.

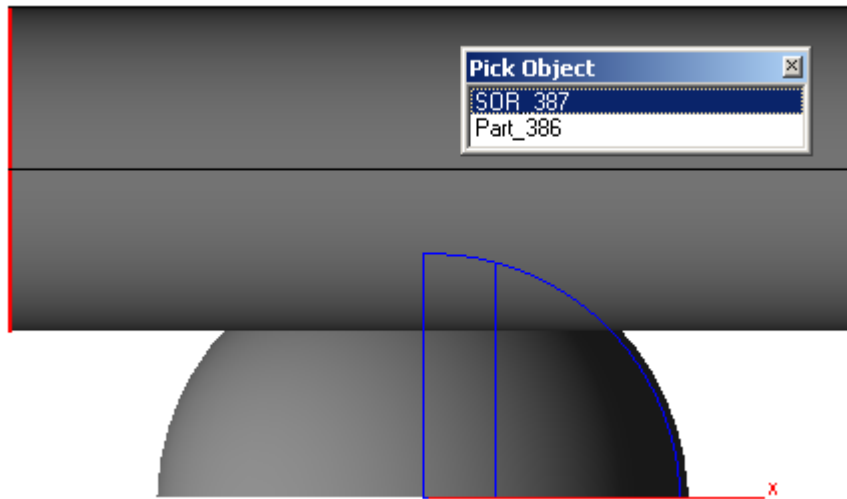


29. Use the **Revolve Surface** tool.

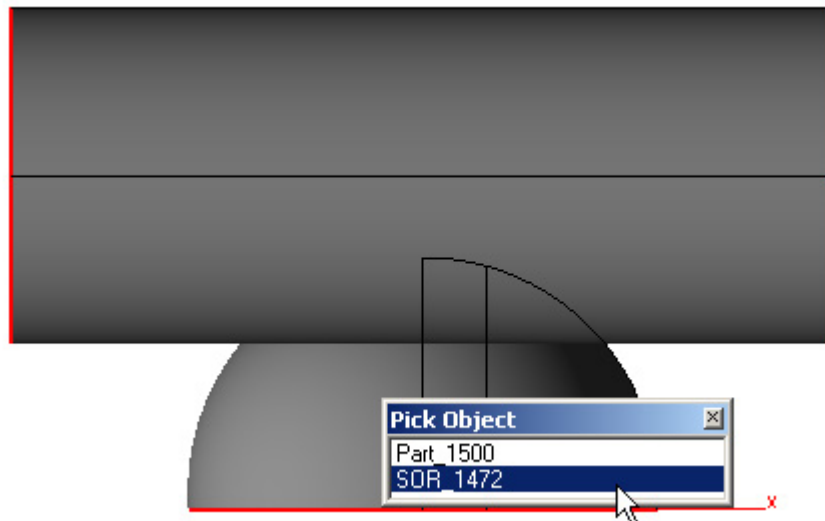


30. Pick the upper line that we just created as the profile to revolve. Then, click both endpoints of the center line.

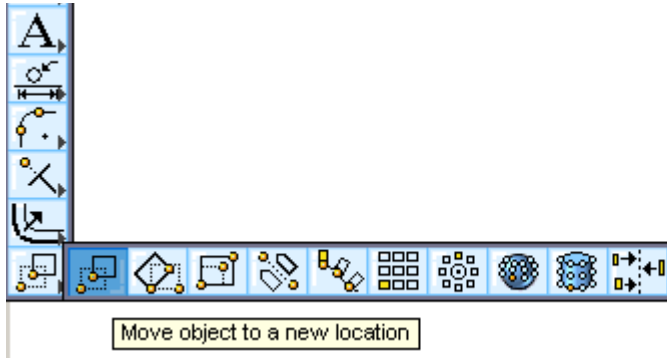
31. To make sure that the surface has been created, click the cylinder with the **Selection** tool and see the two objects: a surface and a solid part in the Pick Object window.



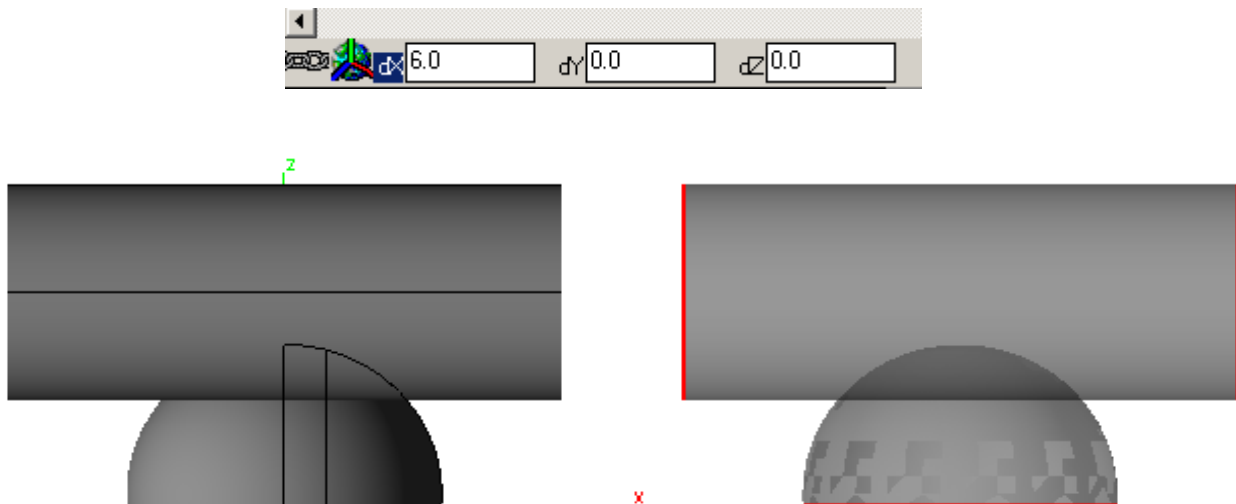
32. Hold down the SHIFT key and click the surface that creates the cylinder and then click on the cap surface from the Pick Object window as in the picture below.



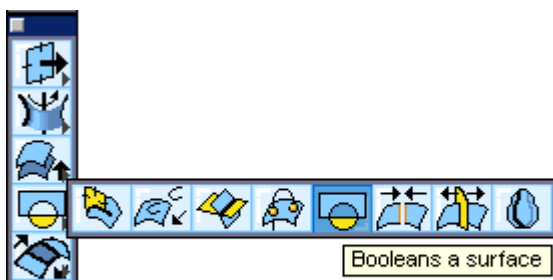
33. Select the **Move** tool from the main tool palette.



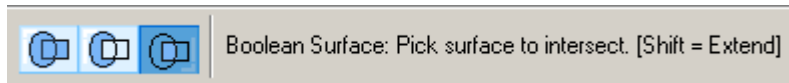
34. Click to the left of the model along the x-axis for the beginning reference point. Move the mouse to the left again along the x-axis and click for the ending reference point. Type **6** for the **dX** value in the Status Line as shown below and press ENTER. This allows relocating the surface objects along the x-axis.



35. To make the intersection of the surfaces we just moved, use the **Boolean Surface** tool.



36. Specify the **Intersection** option in the Message Line on the top of the window.

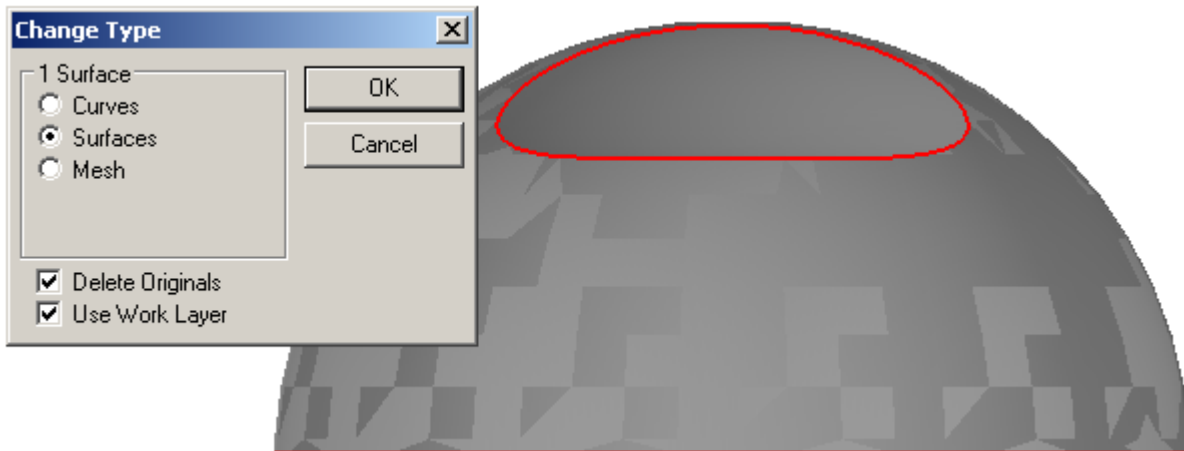


37. First click the cap surface and then the cylinder surface. This creates the recess on the cap surface as in the picture below.

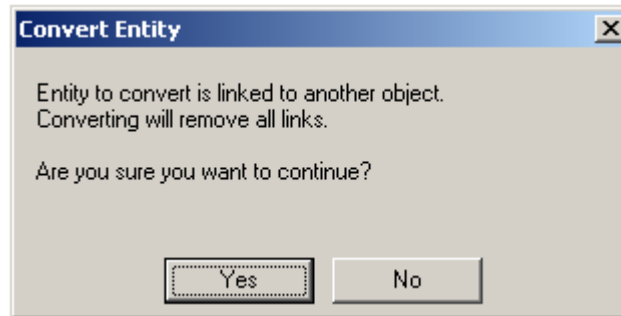


38. Drag to select the recessed cap surface.

39. Choose **Edit>Change Object Type**. Set the radio button next to the Surfaces option and click OK button.



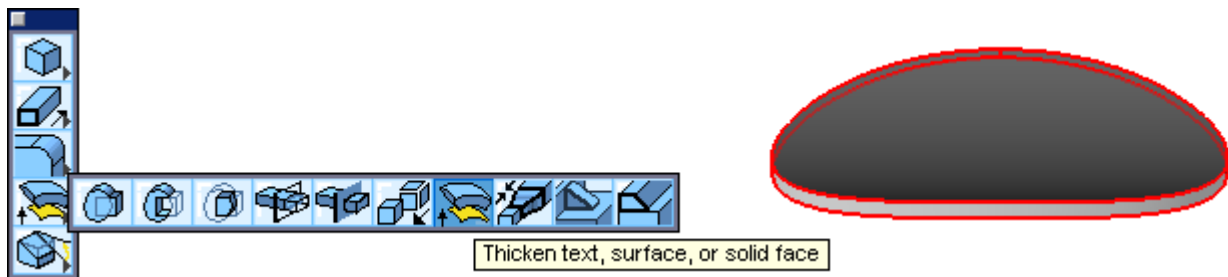
40. The dialogue window cautions that converting the object removes the links. Click Yes button.



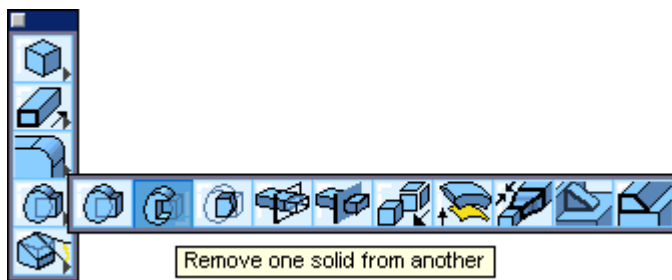
41. With the **Selection** tool, click the lower part of the cap and delete it. You get the petal like this.



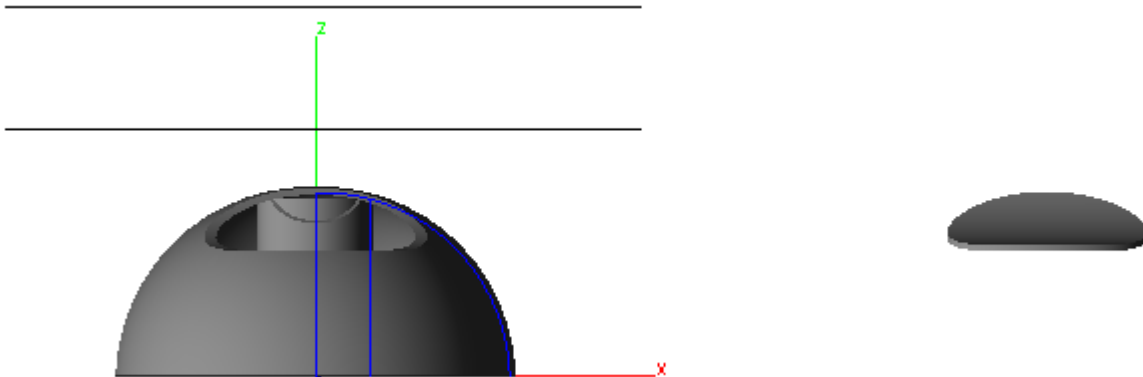
42. Thicken it with the **Solid Thicken** tool. Make sure to hold the CTRL button to thicken it inside.



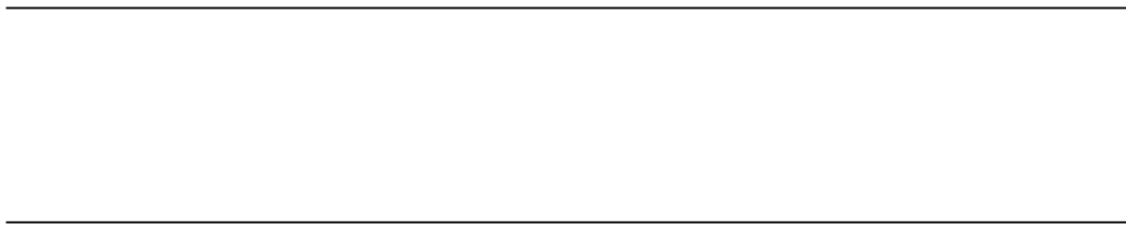
43. Subtract the solid cylinder from the solid cap in the model on the left. Use the **Subtract Solid** tool



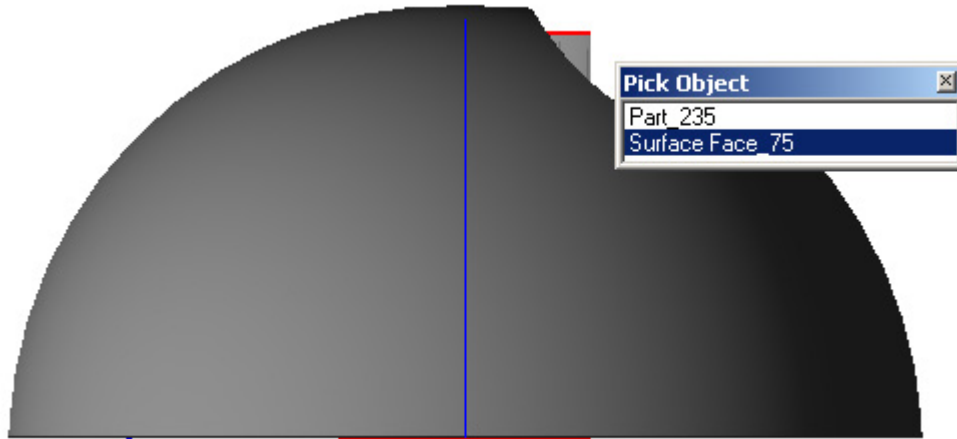
44. First click the solid cap as an object to subtract from. Then click the solid cylinder to subtract. The result looks like this.



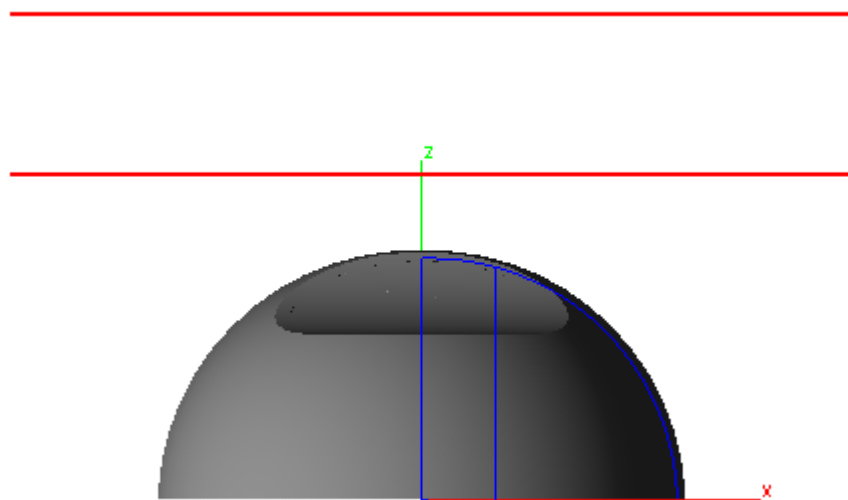
45. Choose the **Move** tool again. Drag the selection rectangle around the petal-shaped object. Click in the area slightly to the left from the petal, move the cursor further to the left along the x-axis and click again. Make sure that the Drafting Assistance prompt appears to confirm that you moved strictly along the x-axis. Type **-6** in the **dX** field in the Status line and press ENTER key.
46. The petal-shaped object relocates to the cap.



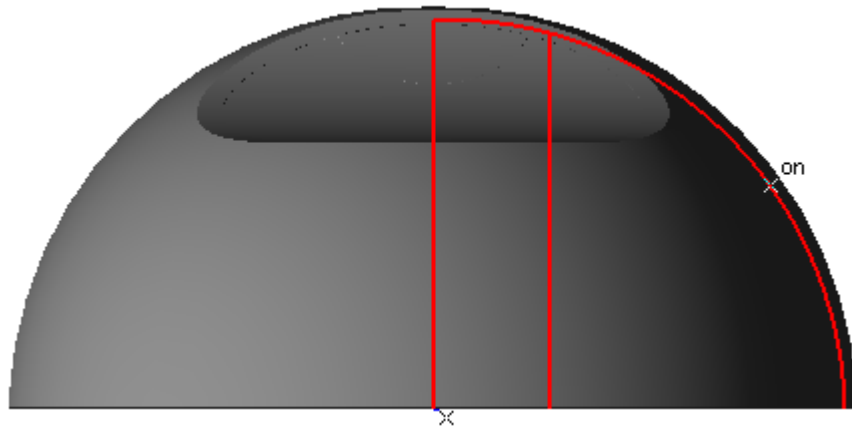
47. Delete all the extra objects now, including the surface that created inner cylinder,



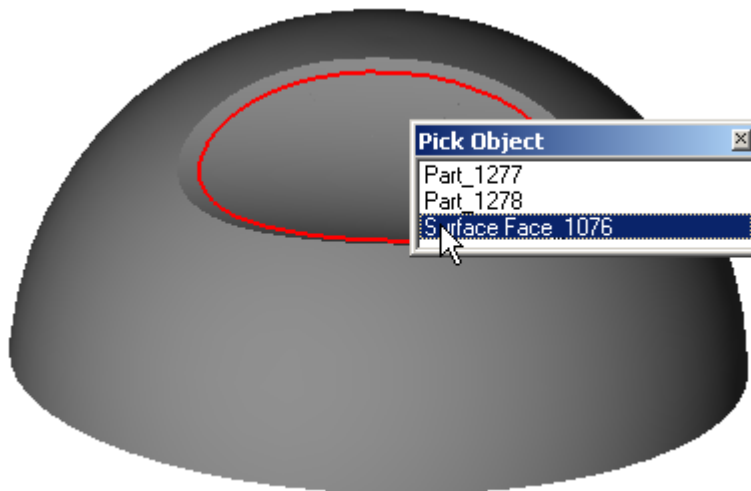
48. auxiliary lines,



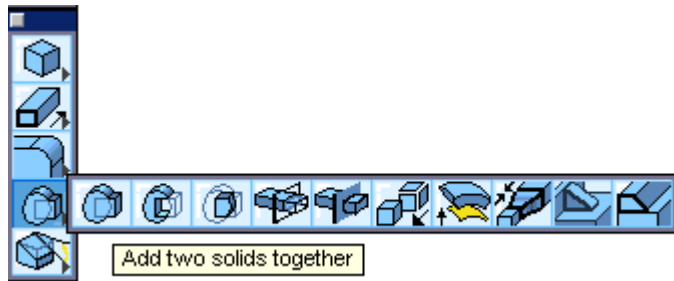
49. the lines and arc used to create the revolved cap surface,



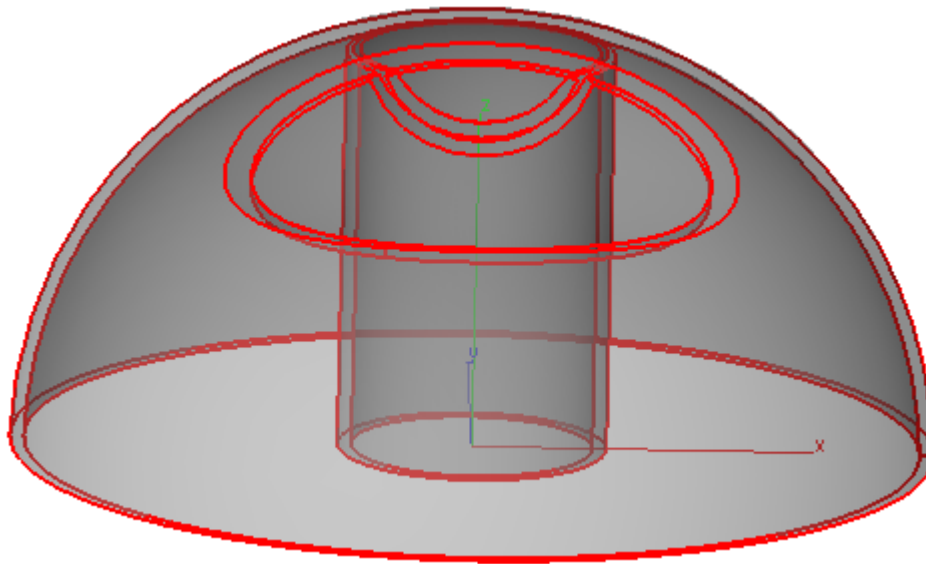
50. and the petal-shaped surface.



51. Unite the petal-shaped solid with the cap solid. Use the **Union Solid** tool.



52. Click the cap solid first, then the petal-shaped solid. The model becomes a single unit. If you click it with the **Selection** tool, it selects as a whole object.



53. Now you can add a color to the object. Drag to select it and right click. Choose the Color option from the context menu and specify one of the predefined colors or click More... to open the color palette. In the picture below you see the final curved cap in predefined blue.

