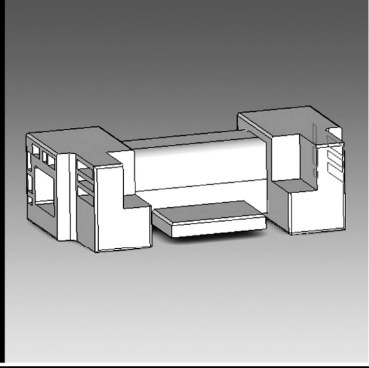
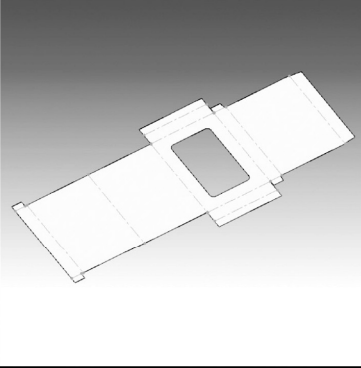
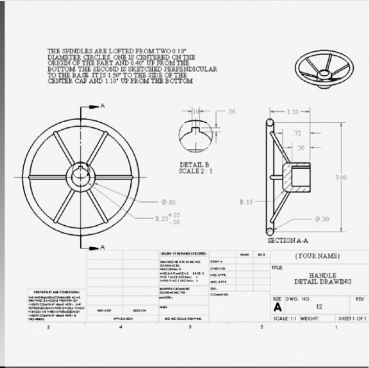
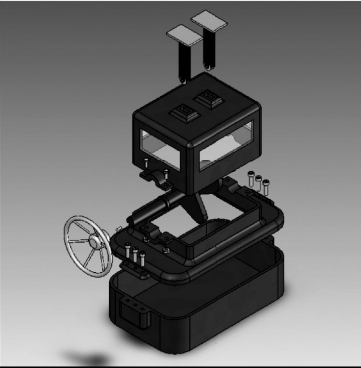
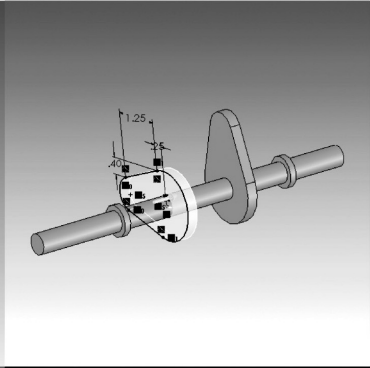
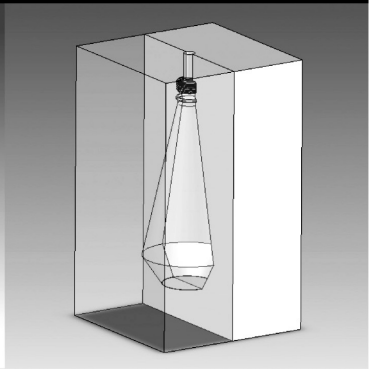
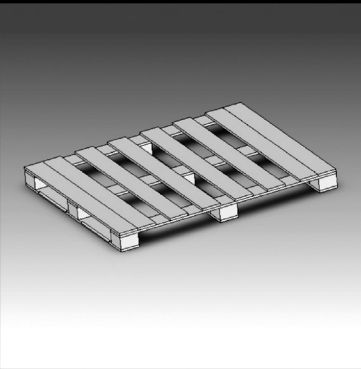
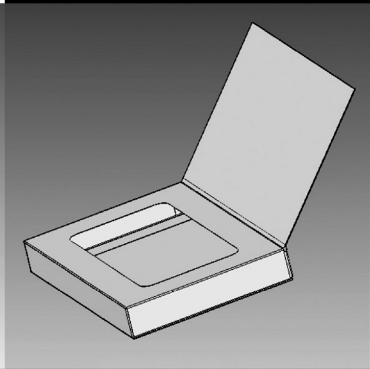
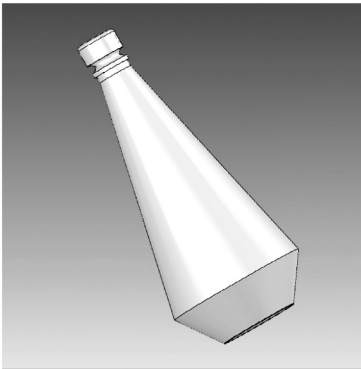
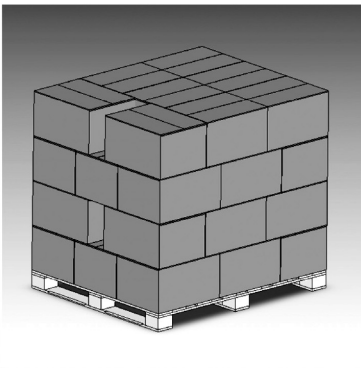


# Package Structure & Mechanism Design with SolidWorks

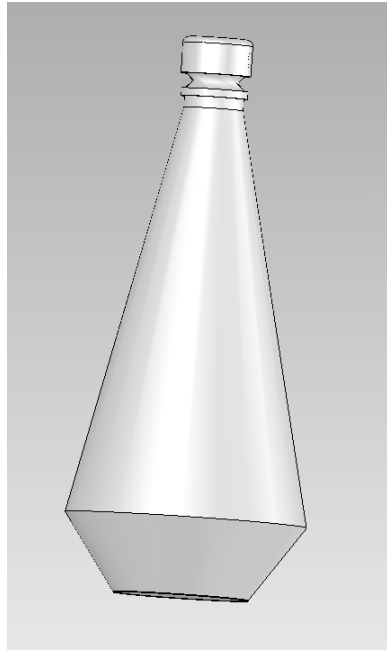
## 2010

Changfeng Ge

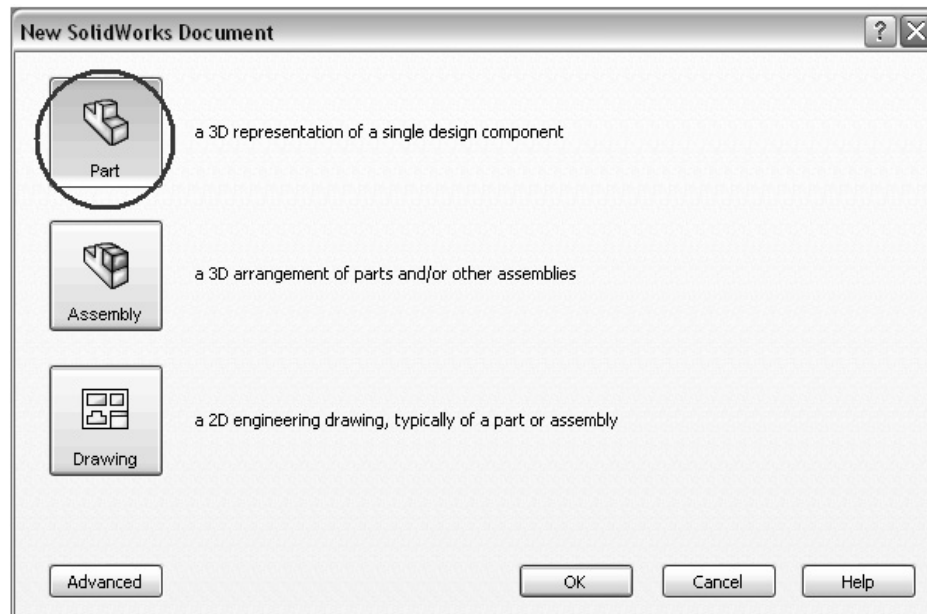
Rochester Institute of Technology



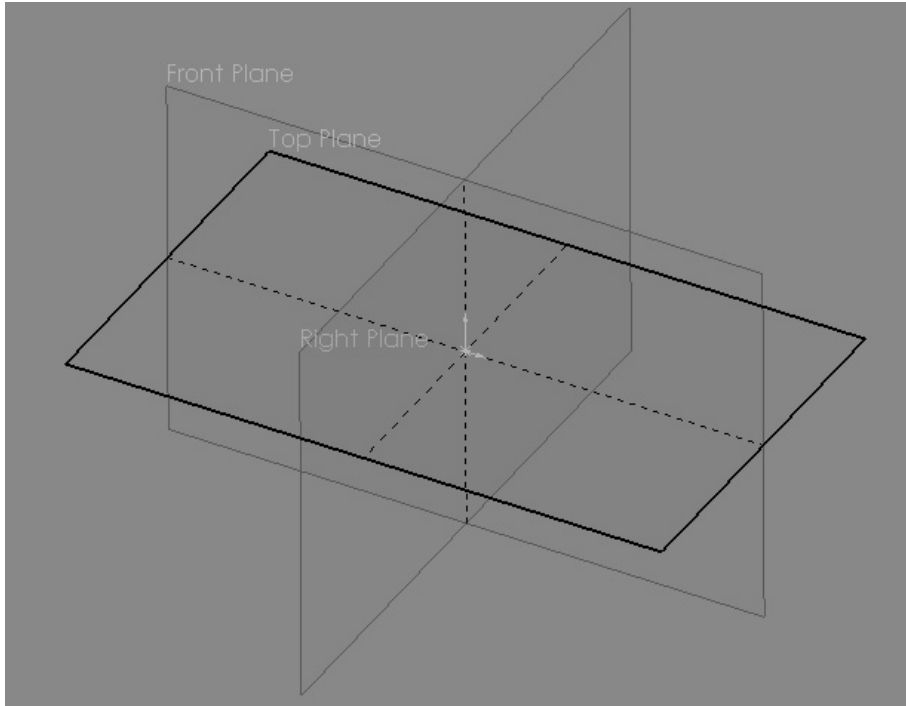
## Chapter 2 Design of a Bottle and a Cap



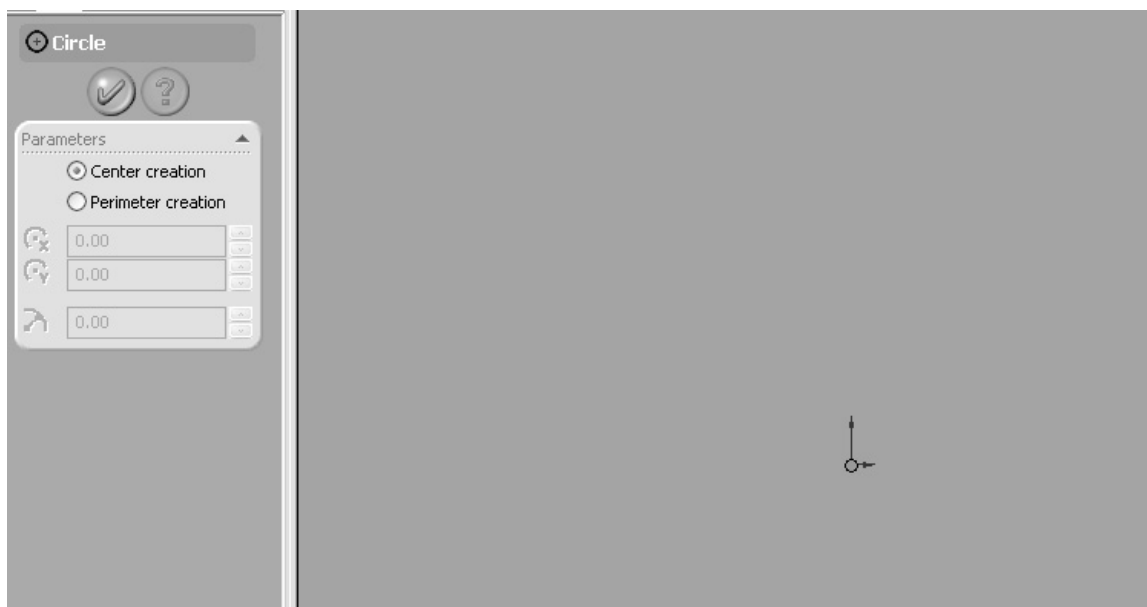
1. Start off with a new Part Document in SolidWorks.



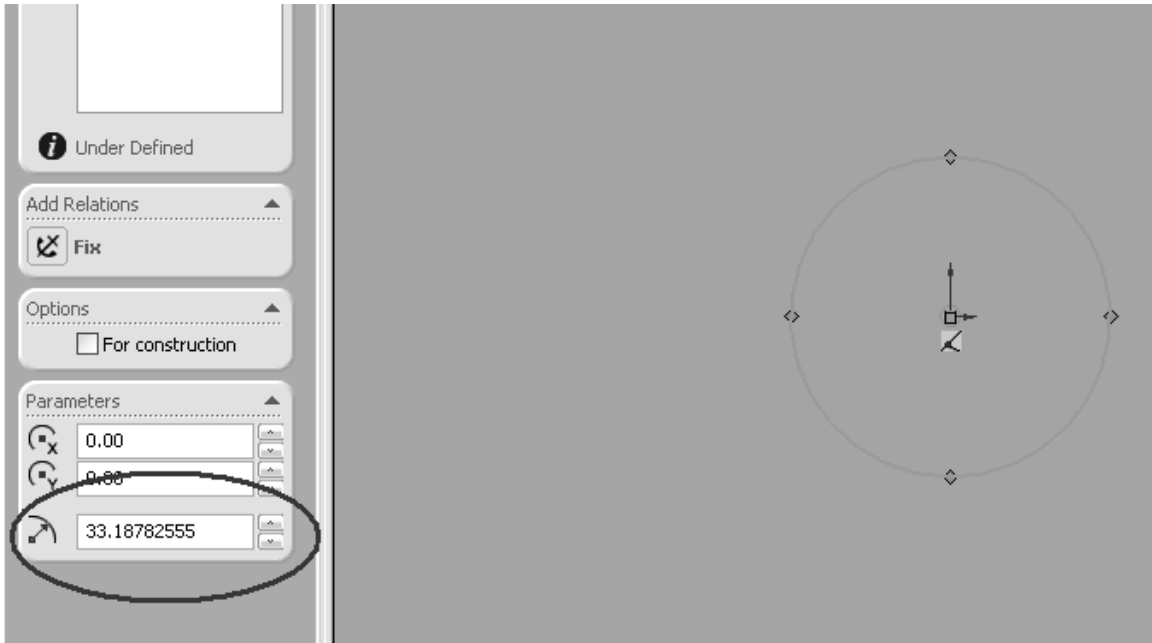
2. Next, choose which plane to sketch on. For this container, choose the top plane.



3. Choose the Circle tool and draw a circle beginning at the origin. Make sure you're making a center creation circle.

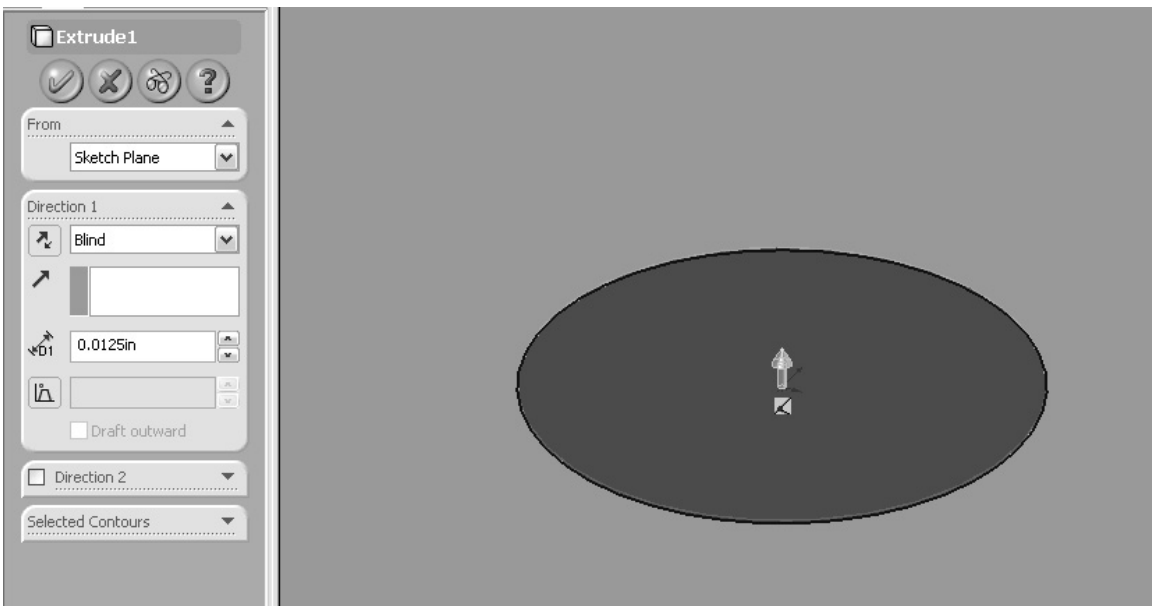


You can then adjust the Radius of the circle to 1.125in.

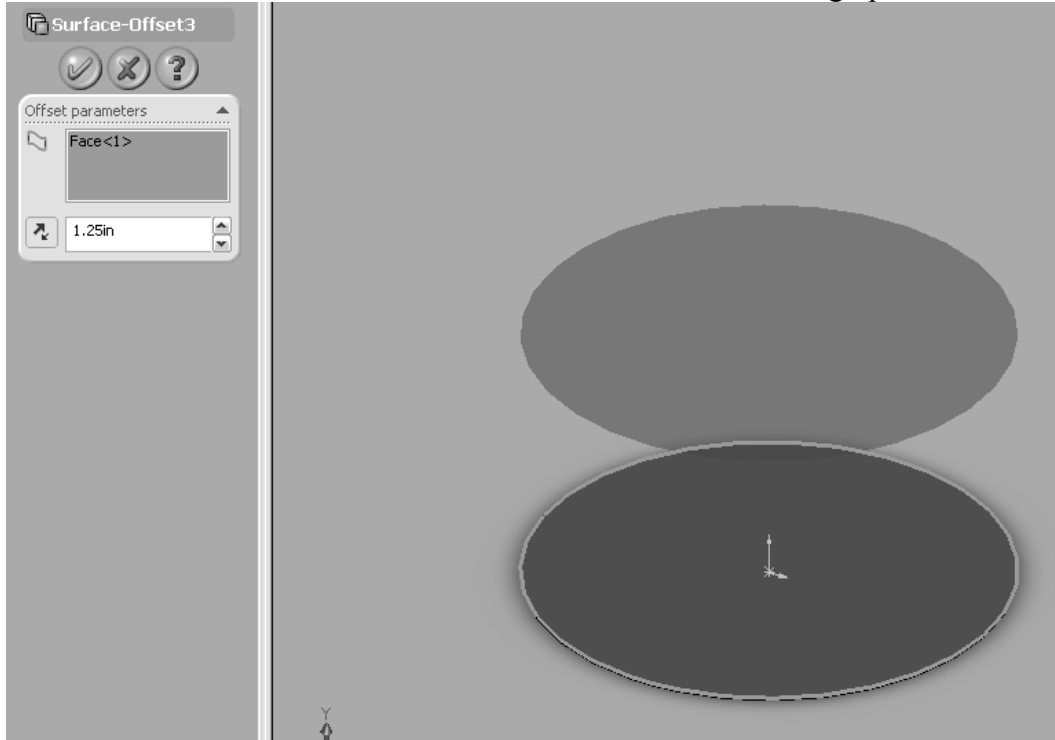


Click the Check button.

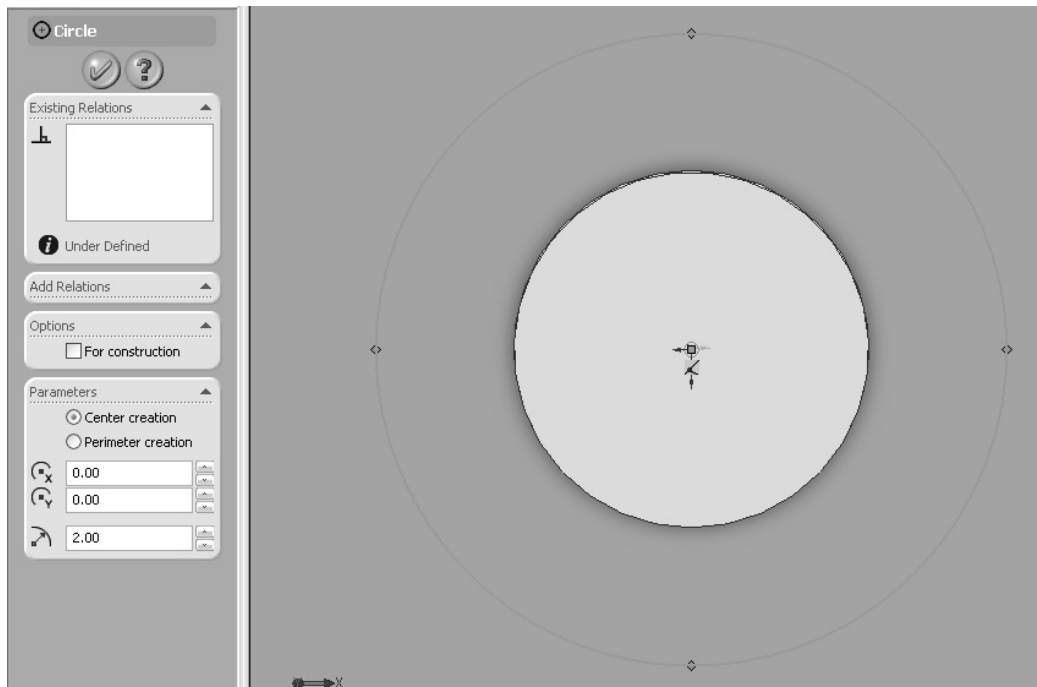
4. Click on the Extruded Boss/ Base button. You want to Extrude the thickness of the bottom of the bottle to 0.0125in.



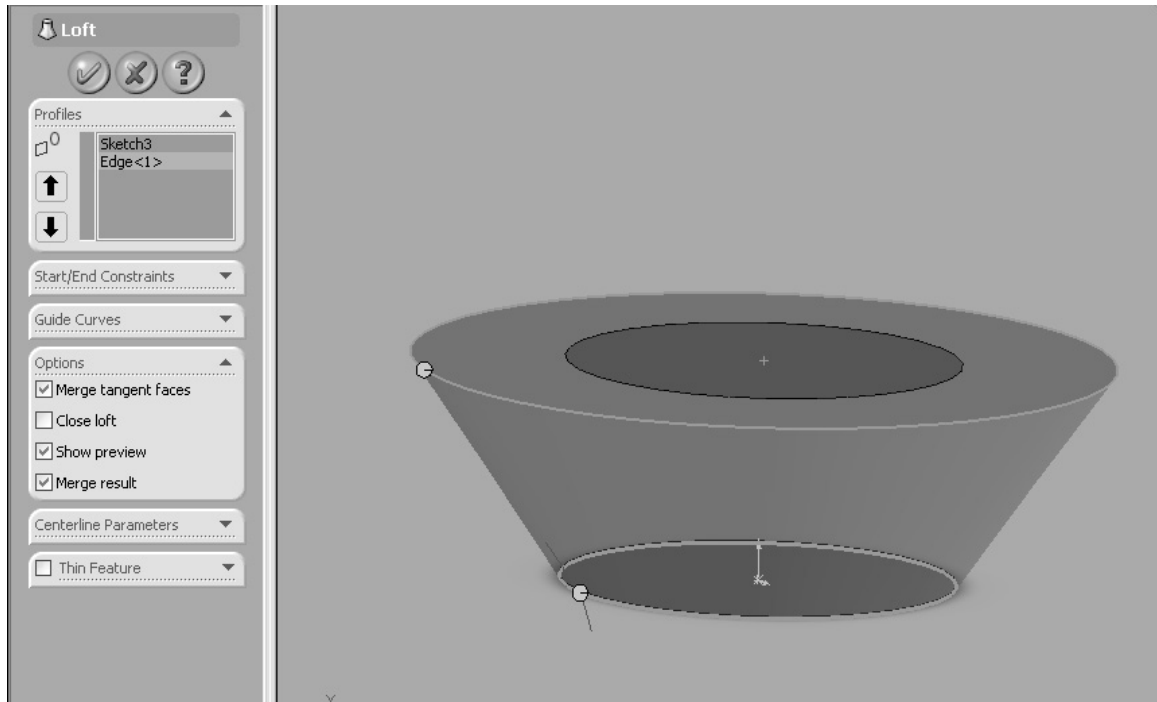
5. Next click View then Toolbars and check Surface button. And look for Offset from the Surface toolbars. Click on the first circle drawn and drag upwards to 1.25in.



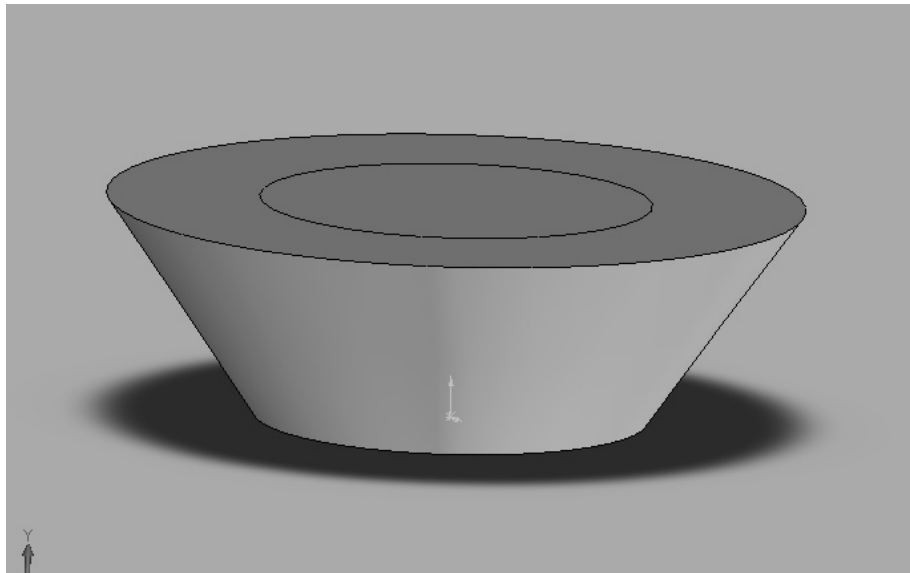
6. You just created a Lofted Surface. From here, you want to draw the next circle. Click on the Circle Tool. Starting on the Origin, create a circle with a 2in radius.



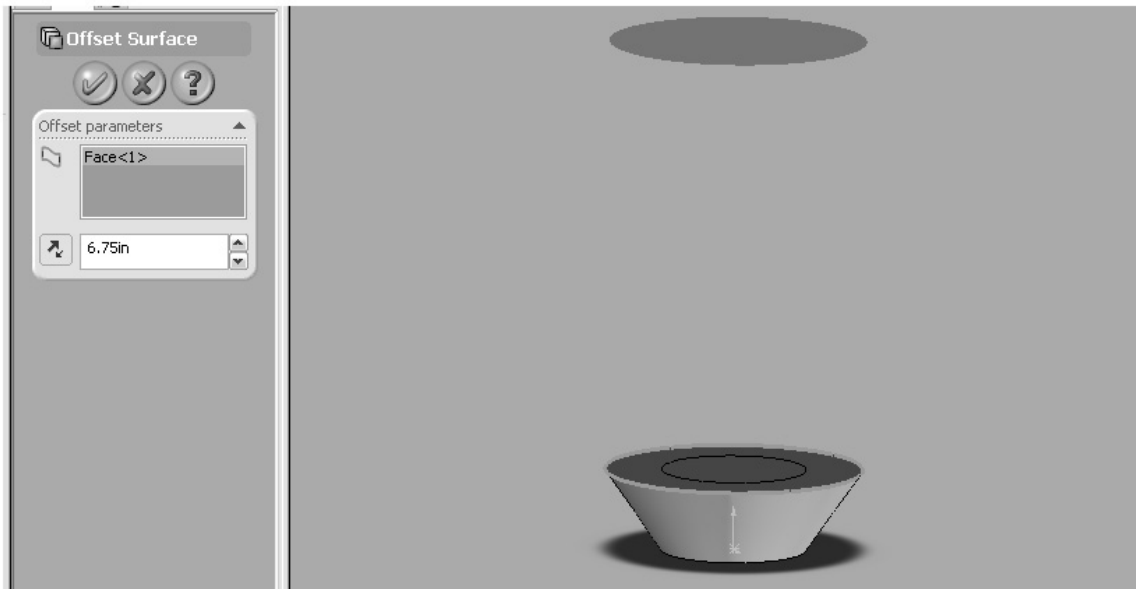
7. You can now create a Loft between two surfaces. Click on the Lofted Boss/Base button.



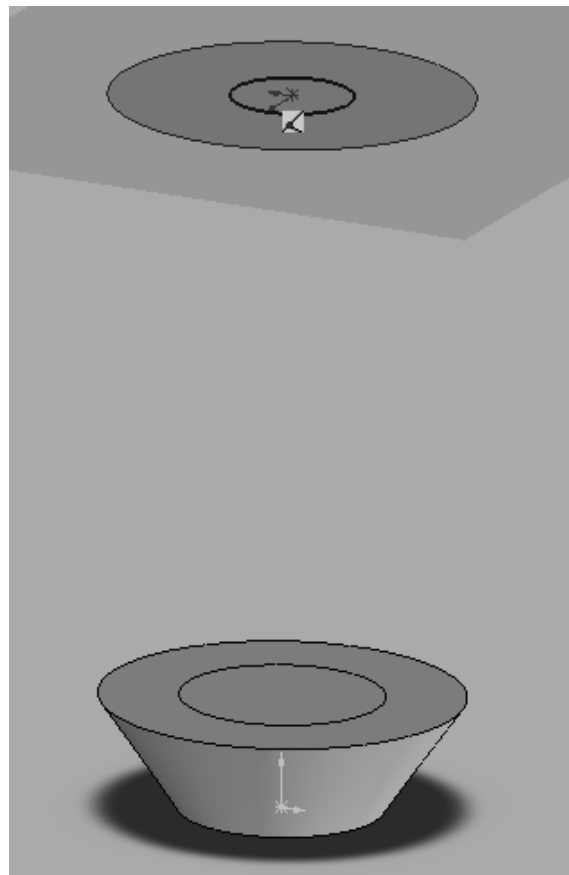
You should end up with something like this. If not, you might have to play around with where you click on the two surfaces.



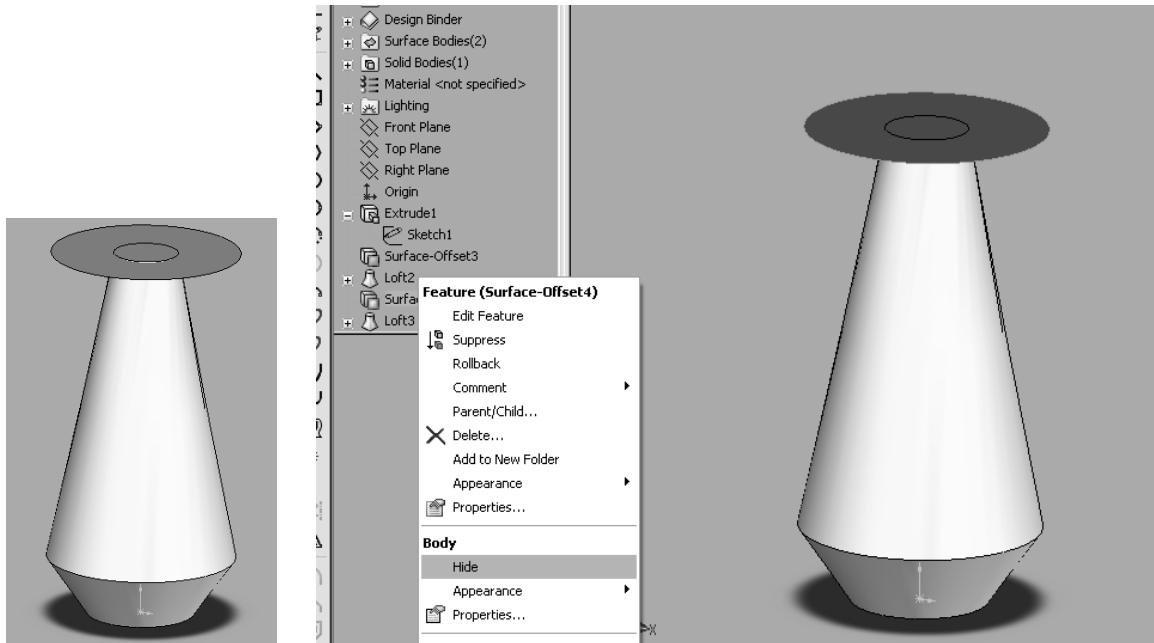
- Next create another Lofted Surface by clicking on the Offset Surface button. This surface should be lofted up 6.75in.



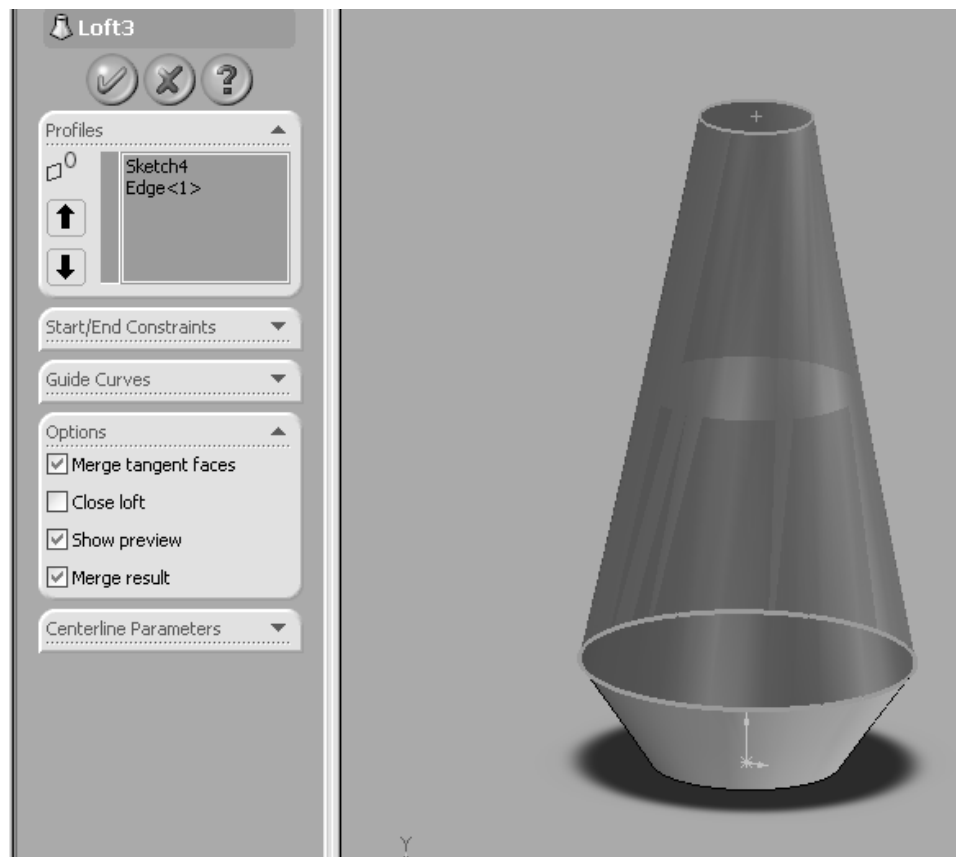
Draw a circle on the new surface, starting at the Origin, with a radius of 0.5in.



If the Lofted Surface is distracting, you can click the Hide option in the Feature Manager Design Tree.

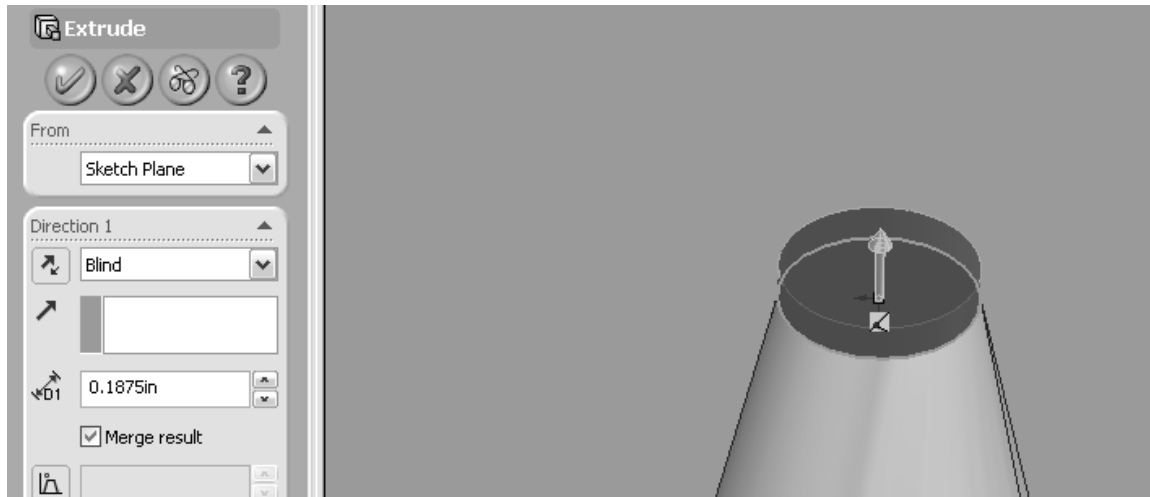


9. Create a Loft between the new surface circle and the base.





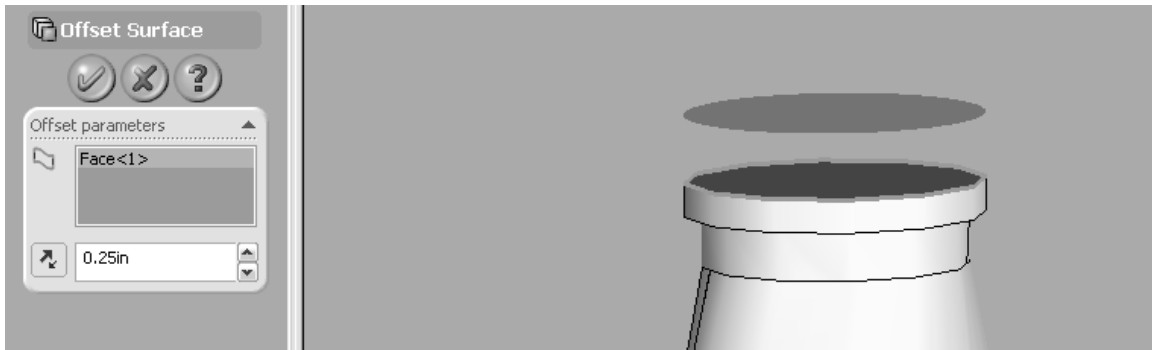
10. Draw a circle on the surface, starting at the Origin, with a radius of 0.5in. Next, Extrude the circle up 0.1875in.



11. Draw a circle on the top surface, starting at the Origin, with a radius of 0.5625in. Extrude this circle up 0.125in.

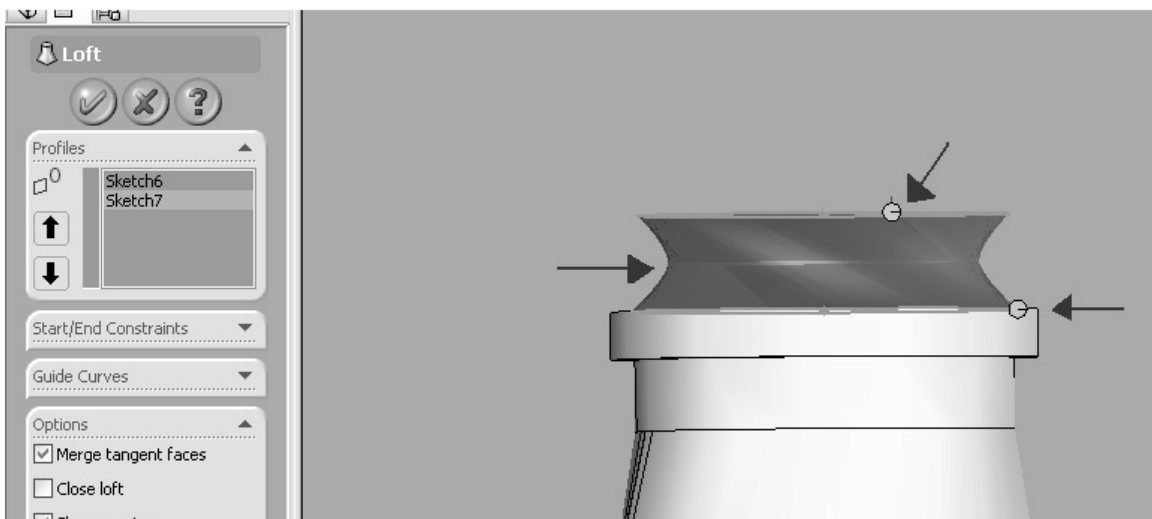


12. Draw a circle in the top surface, starting from the Origin, with a radius of 0.5in.  
Create an Offset Surface 0.25in away from the top surface.



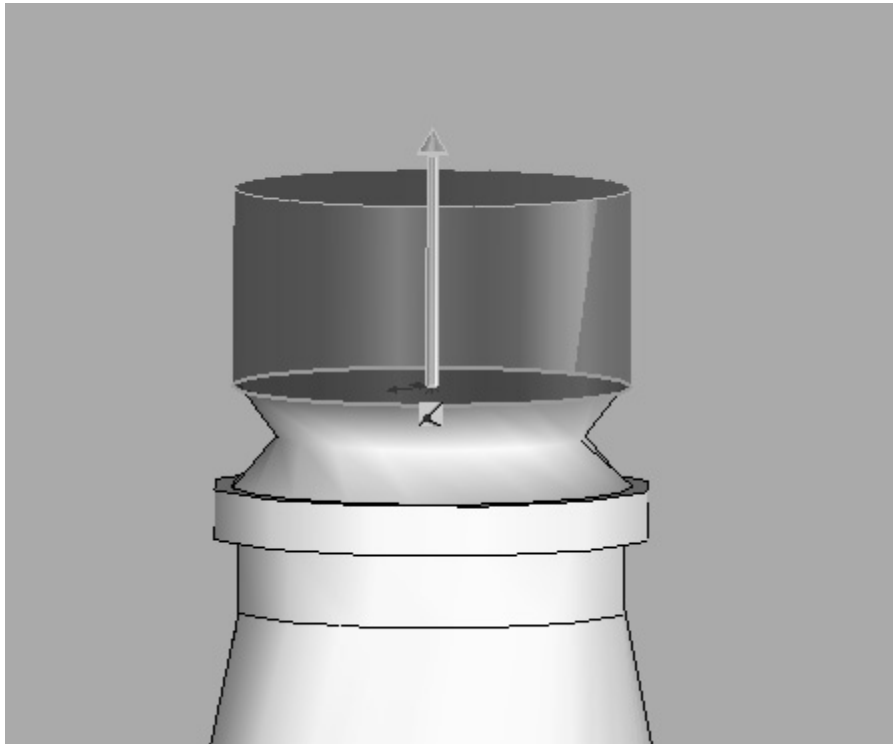
Draw a circle on this new surface, starting from the Origin, with a radius of .4875in.

13. Use the Lofted Boss/Base tool to connect these two circles.

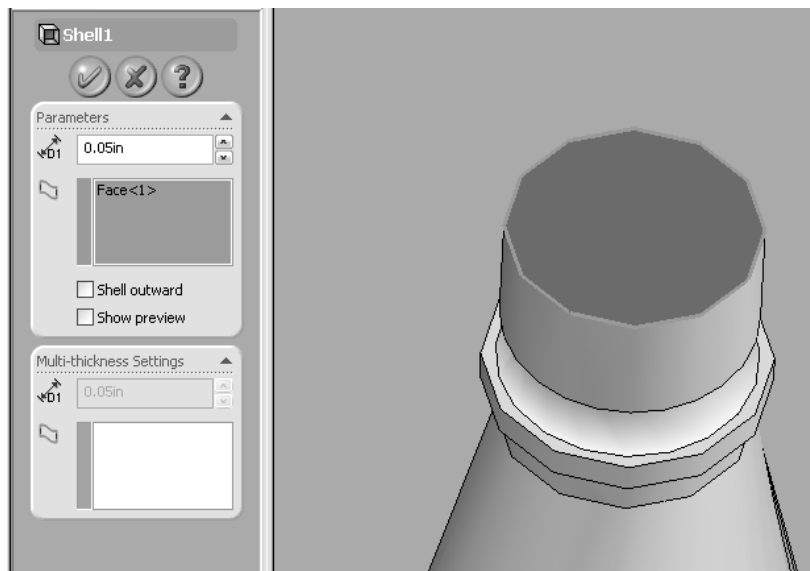


Be sure to click on different places on each circle to create the indent between the two circles.

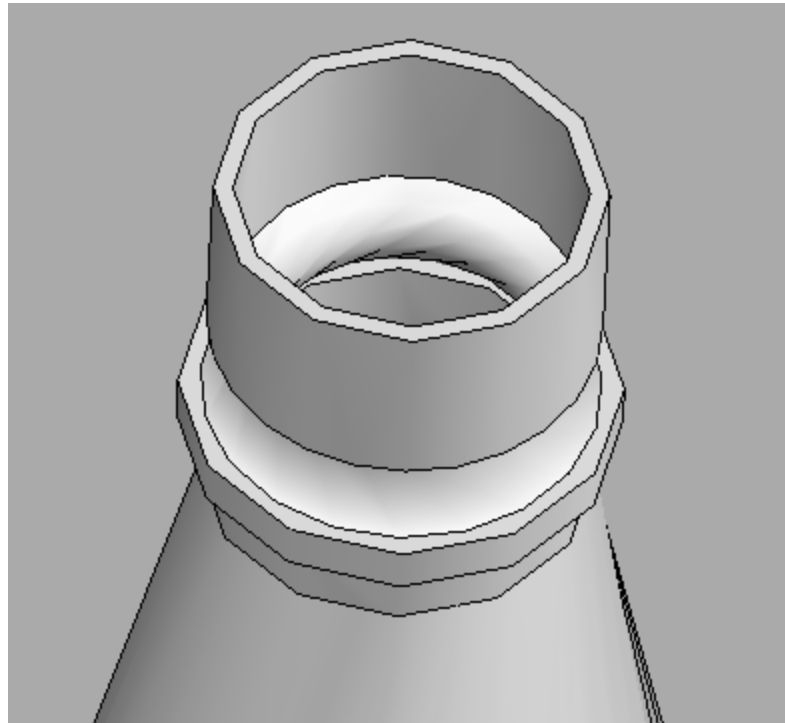
14. Draw a circle, starting at the Origin, with a 0.5in radius on the top surface.  
Extrude this circle up 0.5in.



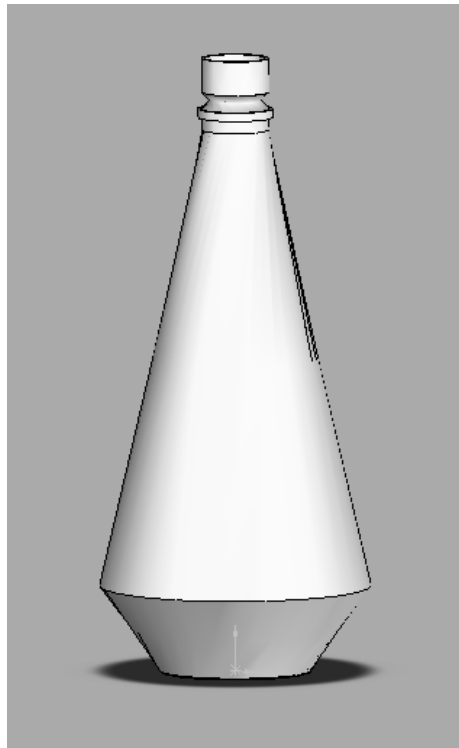
15. The body of the bottle is now complete! The final step to finish it off is to make the bottle hollow. Click on the Shell tool.



Change the Shell thickness to 0.05in. After shell, you should check if there is any lofted surface inside the bottle, if yes, you should remove it by click “Hide” as the lofted surface is used ONLY for construct the bottle.

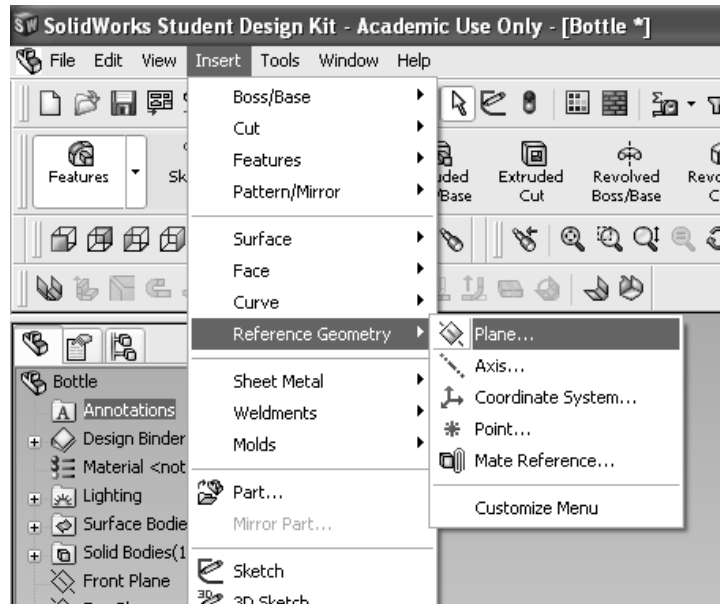


The bottle is now complete!

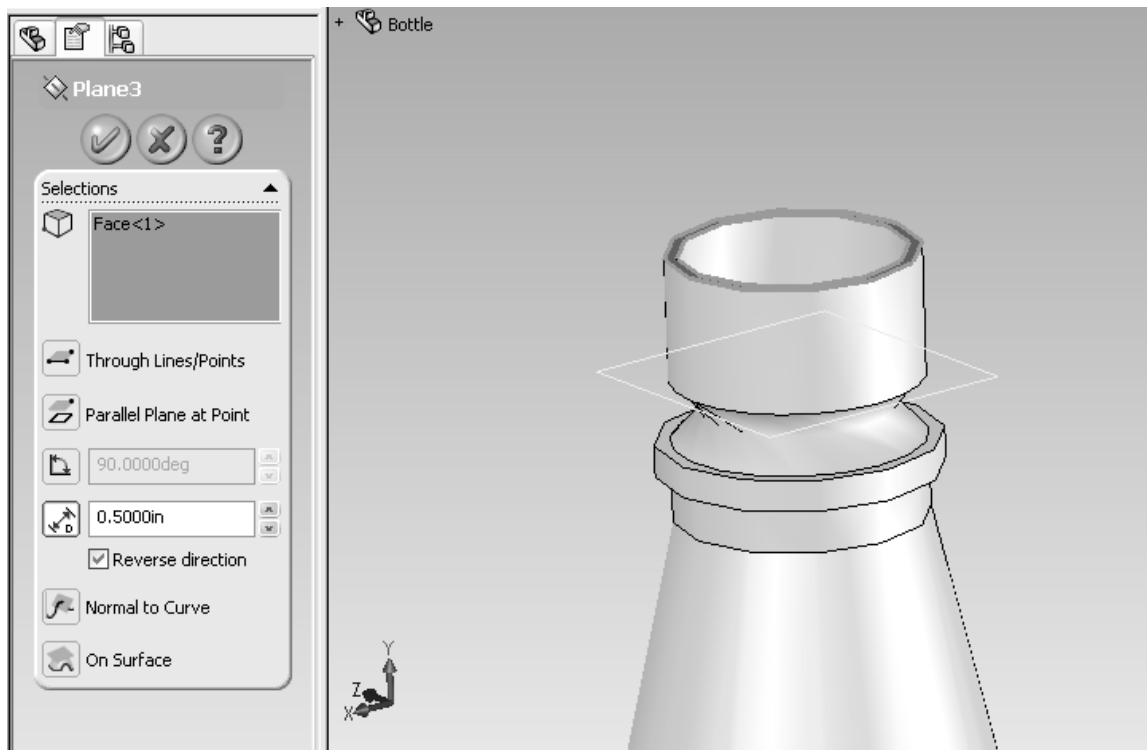


Now we will add Screw Threads on the bottle and Bottle Cap.

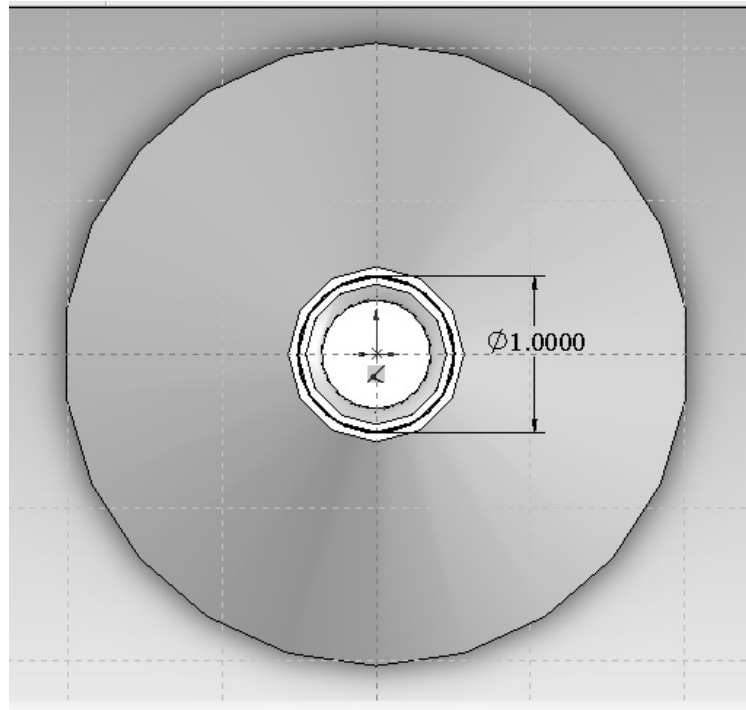
16. Create the circle that dictates how wide the Helix will be. Inset a Reference Plane on the top surface of the bottle.



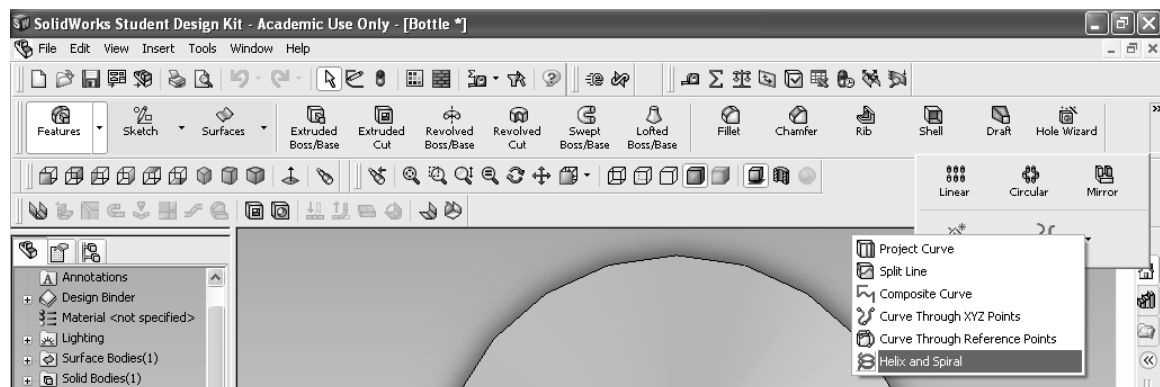
17. Now select the top surface of the bottle neck and set the distance to 0.5 in.



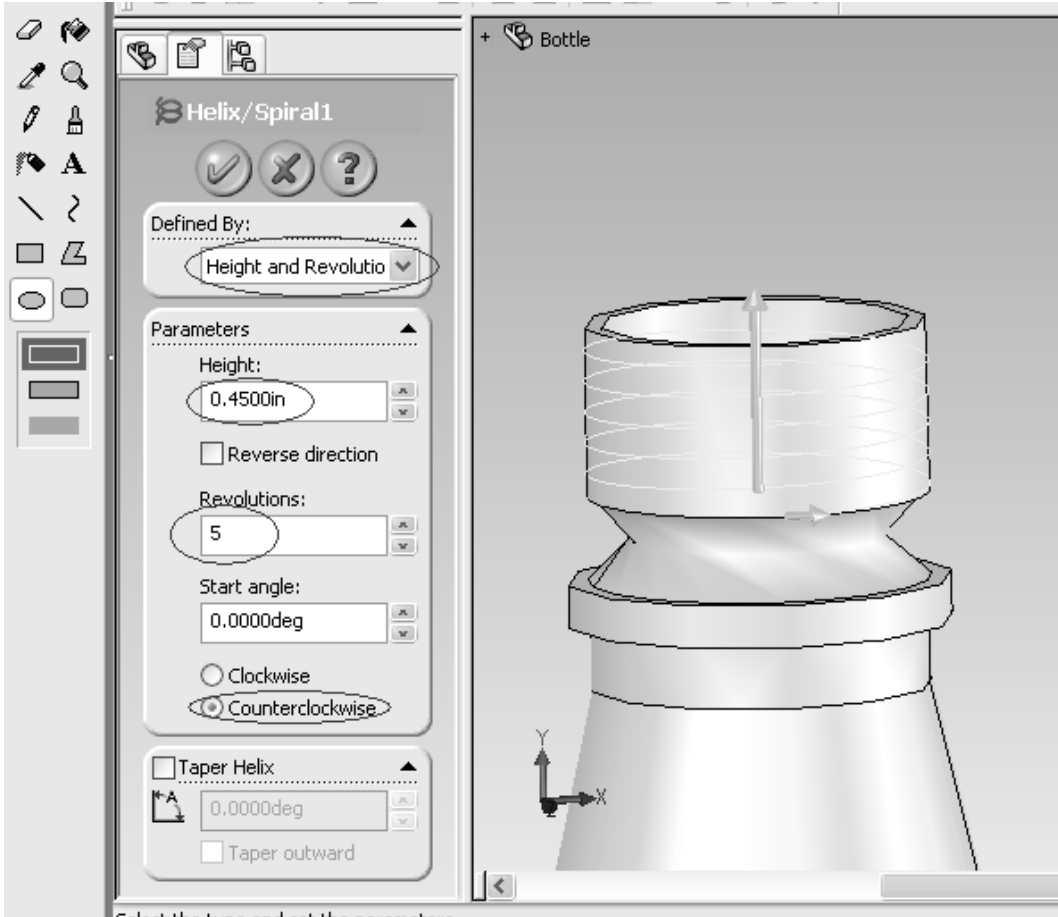
18. Insert a New Sketch on the plane we just created and draw a circle with a diameter of 1.0 in.



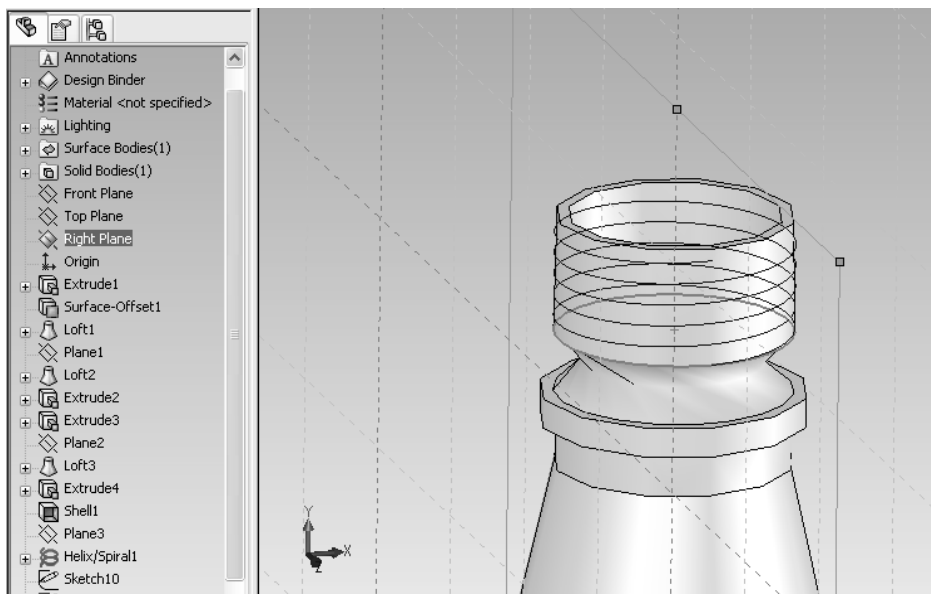
19. Exit that sketch and click on the Helix tool under the Curves menu.



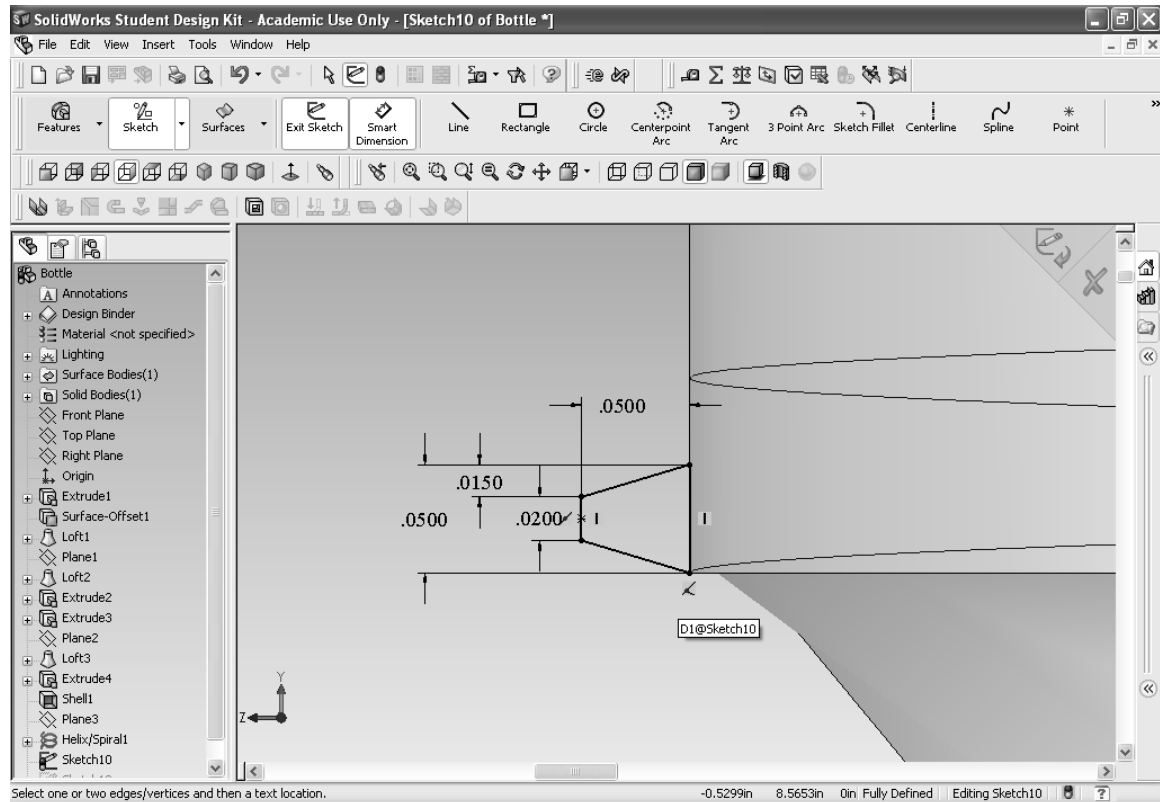
20. Set the parameters the same as below and click OK. We now have the path that the screw will follow.



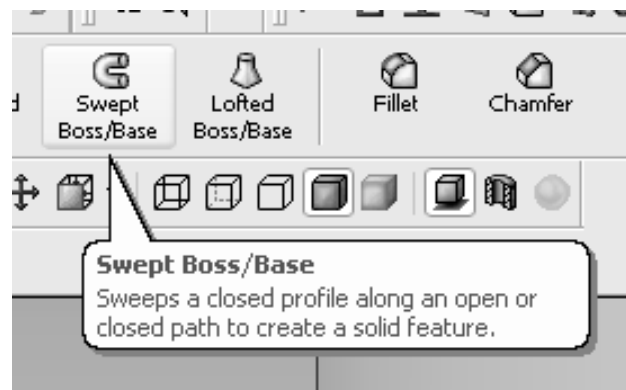
21. Create a New Sketch on the Right Plane.



22. Look at the Right Side and zoom in to the bottom edge of the neck where the helix begins and create a sketch coincident with the bottom edge and with the dimensions as shown.

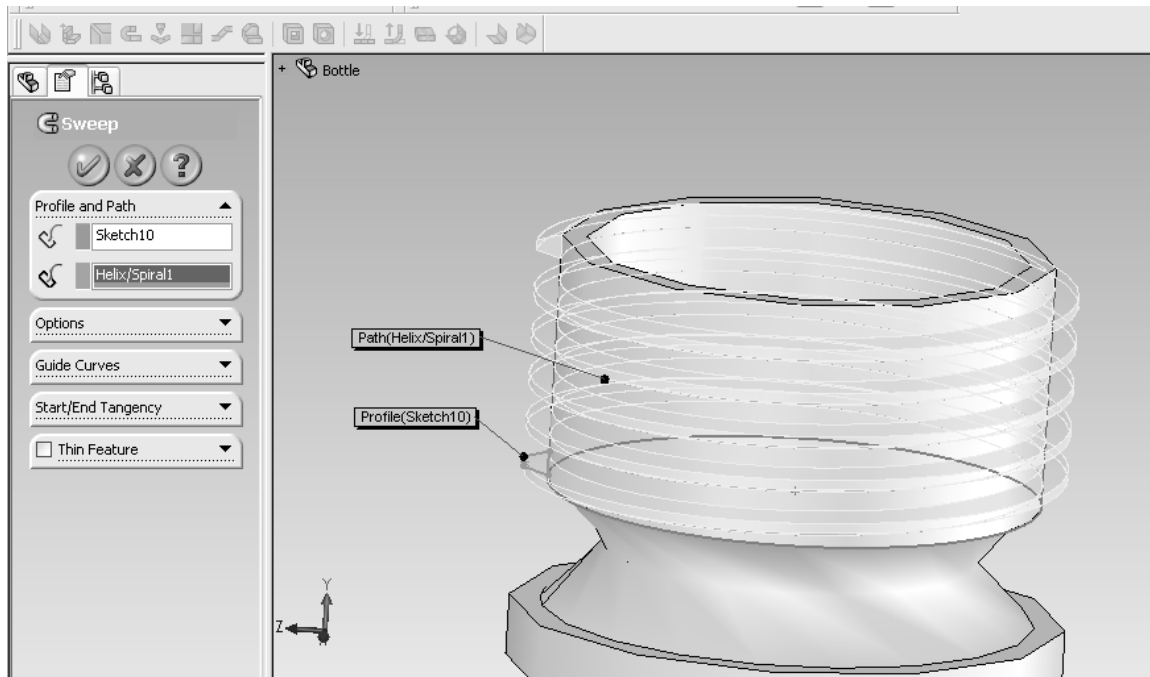


23. Now exit that sketch and select the Swept Boss/Base tool.

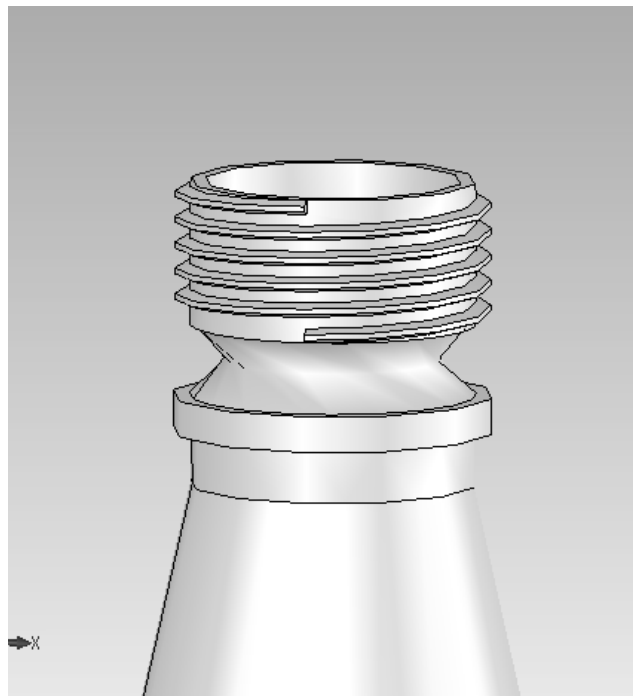




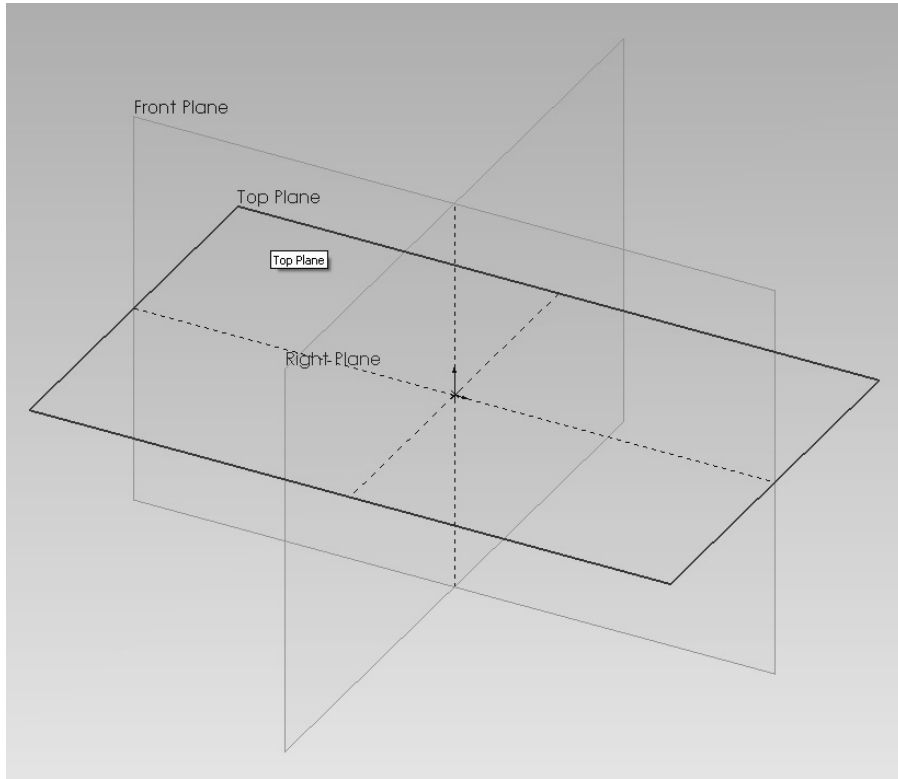
24. Select the sketch we just made in step 22 for the profile and the helix for the path.



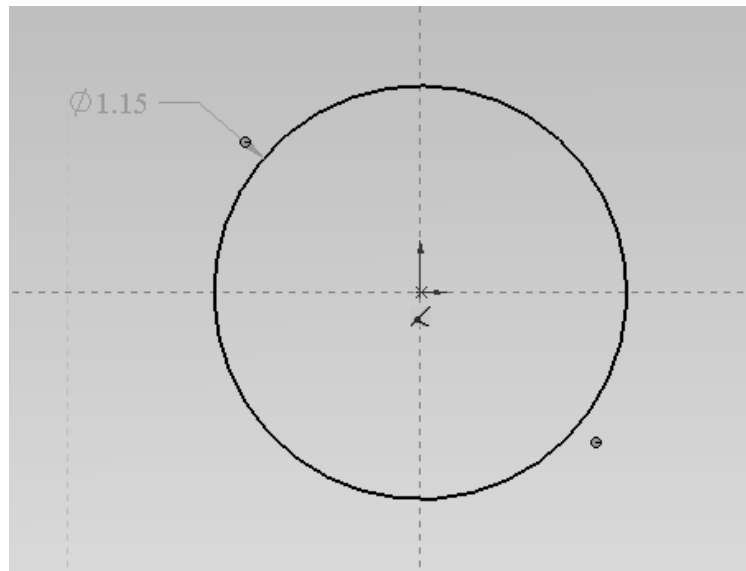
25. Click OK and the thread is complete.



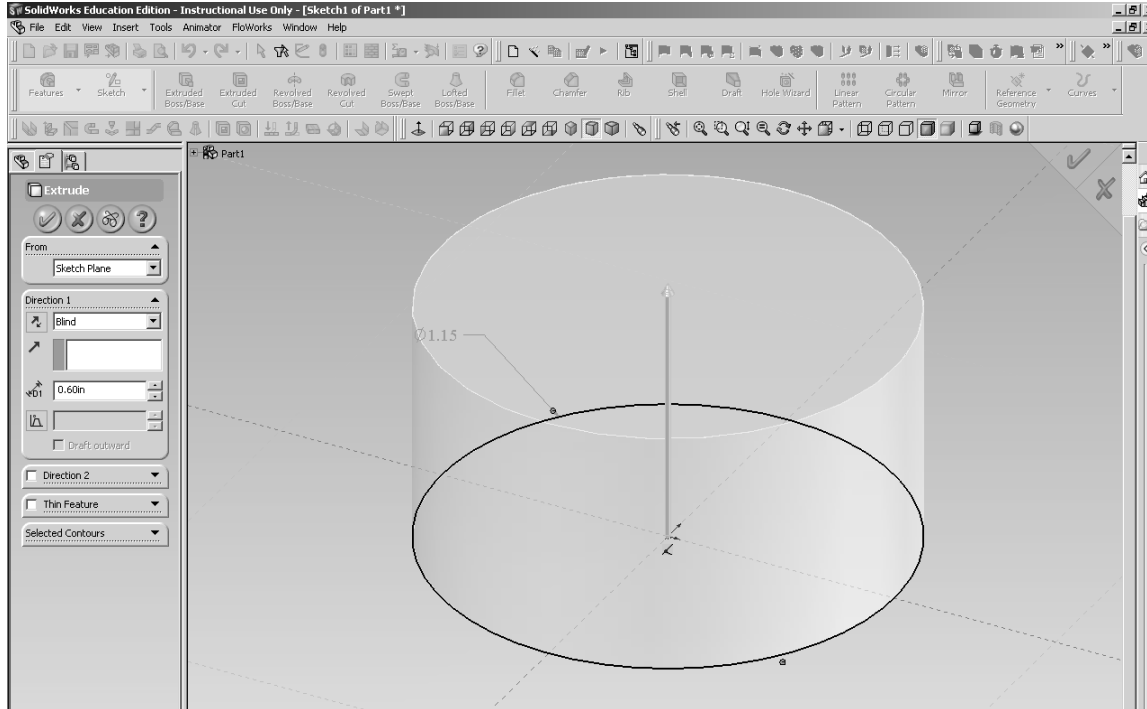
26. Create a New Part file and insert a sketch on the top plane. This will become the threaded cap for the bottle.



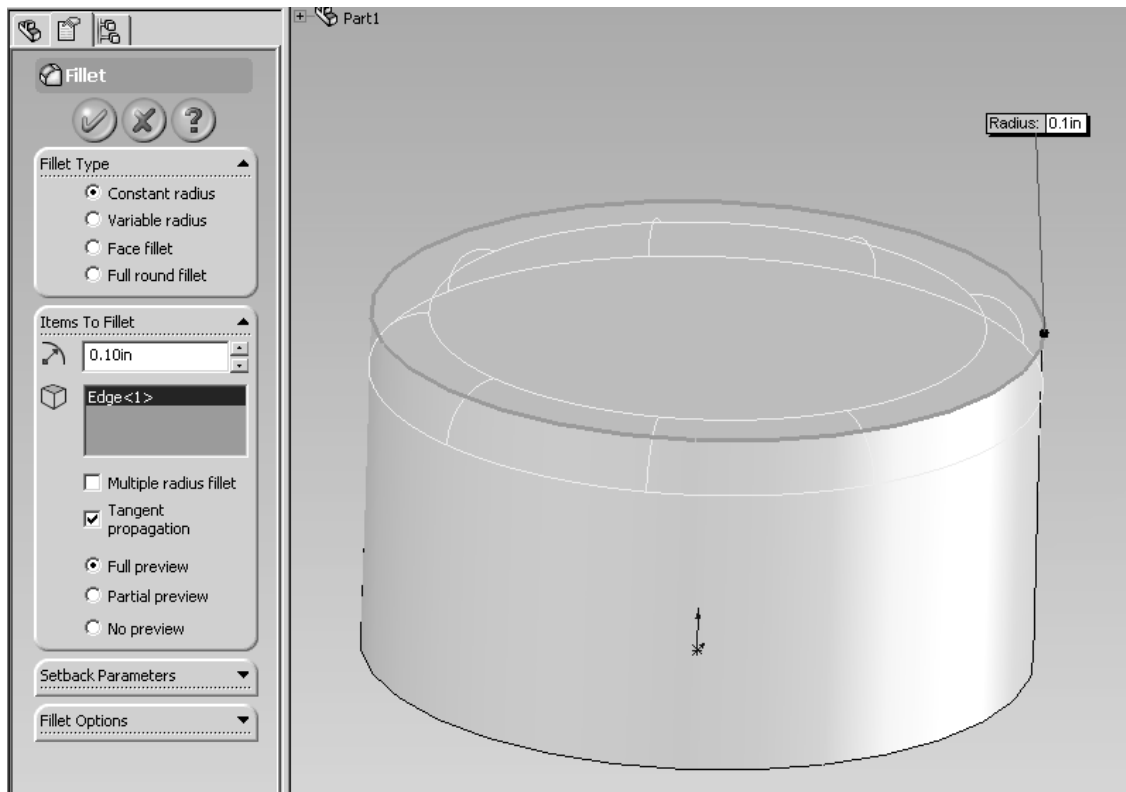
27. Create a circle with the center at the origin and a diameter of 1.15 in.



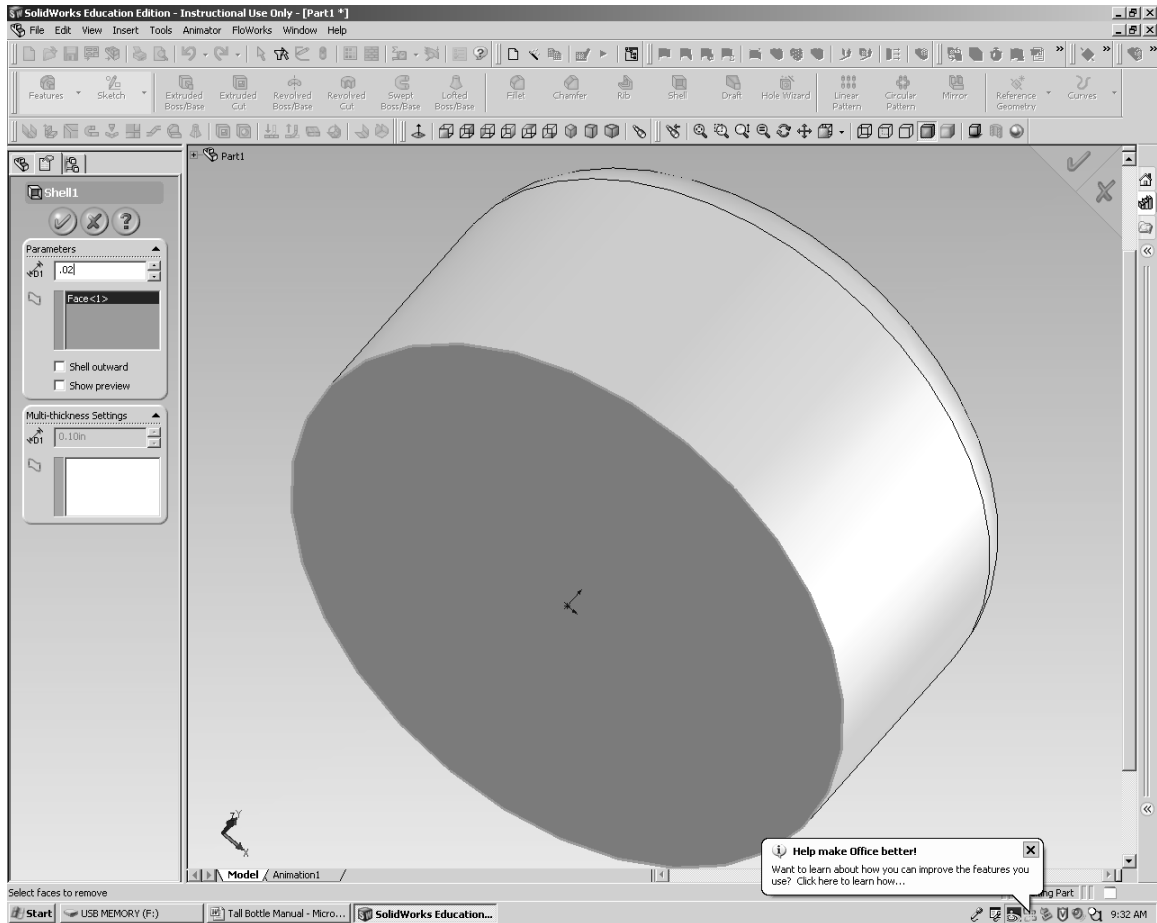
28. Extrude it 0.6 in. up.



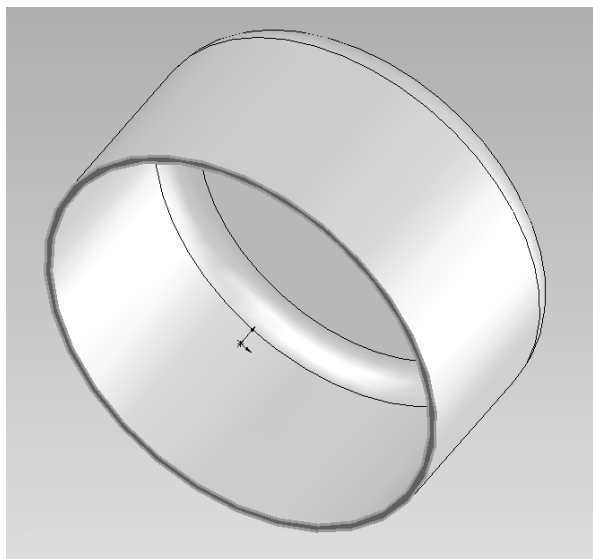
29. Add a fillet to the top edge with a radius of 0.1 in.



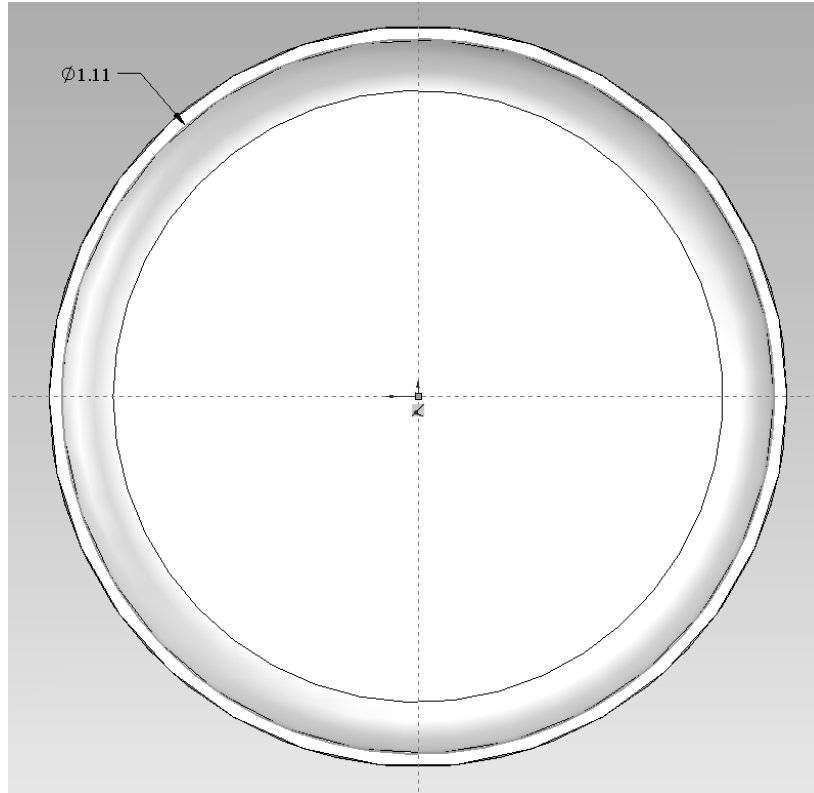
30. Now use the Shell tool to hollow out the bottom of the cap. Make the wall thickness 0.02 in.



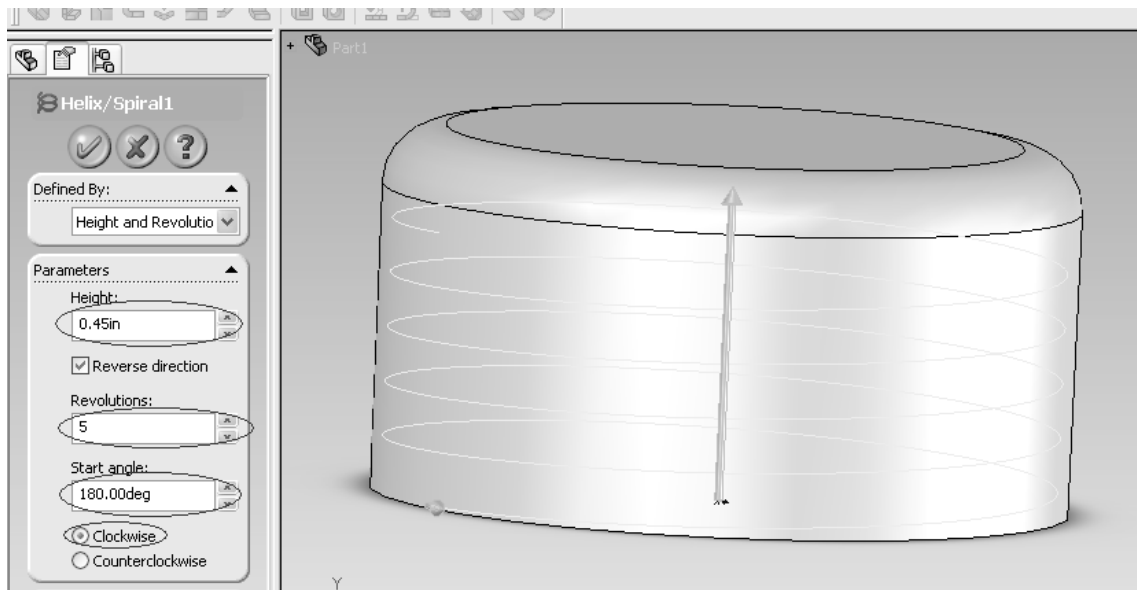
31. Insert a sketch on the bottom lip of the cap.



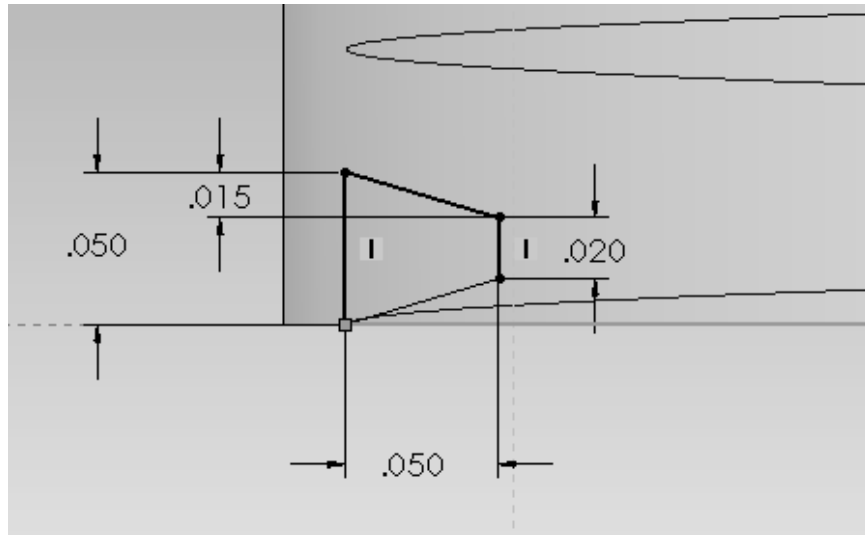
32. Draw a circle with a diameter of 1.11 in. This is the guideline circle for the helix.



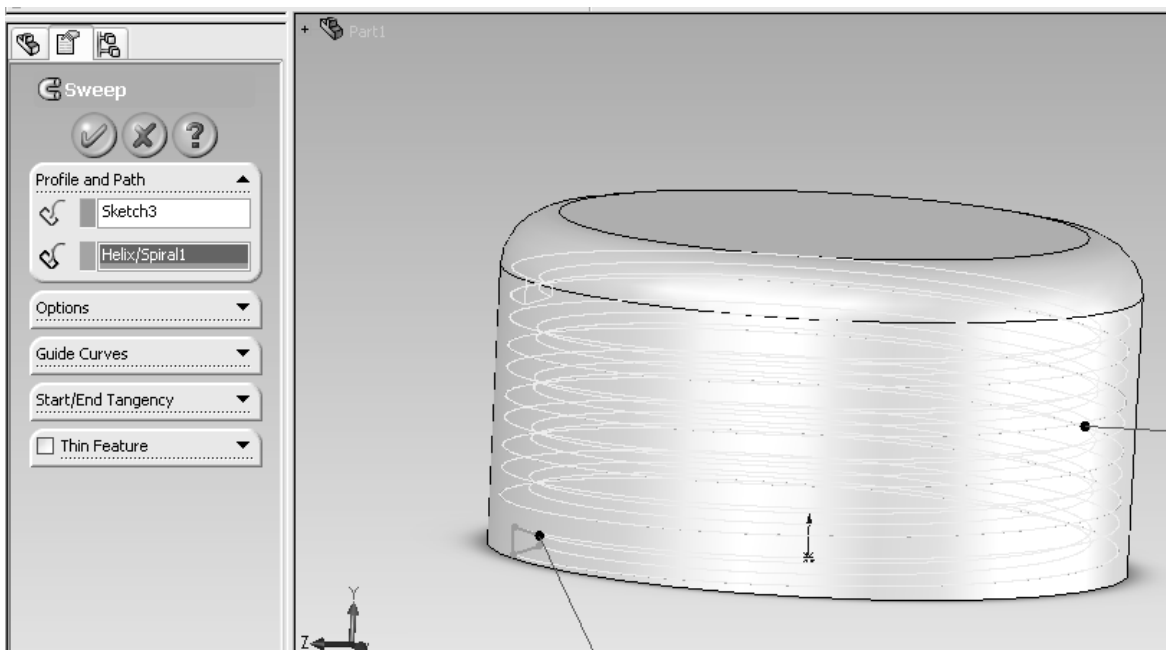
33. Exit the sketch and create a helix with a Height of 0.45, with 5 Revolutions, set the start angle to 180 degrees and turning Clockwise.



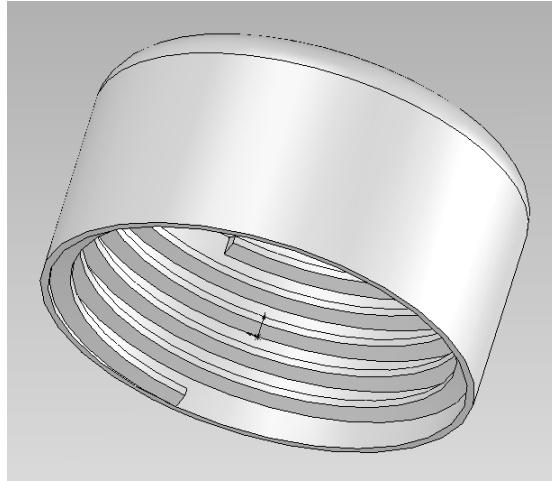
34. Create a new sketch on the Right Plane and zoom in to the bottom edge of the cap where the helix begins and create a sketch coincident with the bottom edge and with the dimensions as shown.



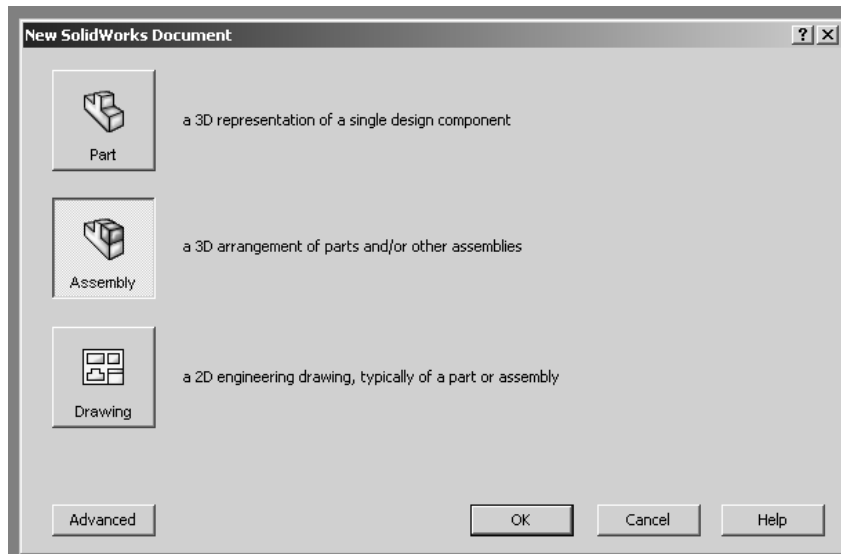
35. Exit the sketch and click on the Sweep tool. Again select the profile and the Helix for the path. Click OK.



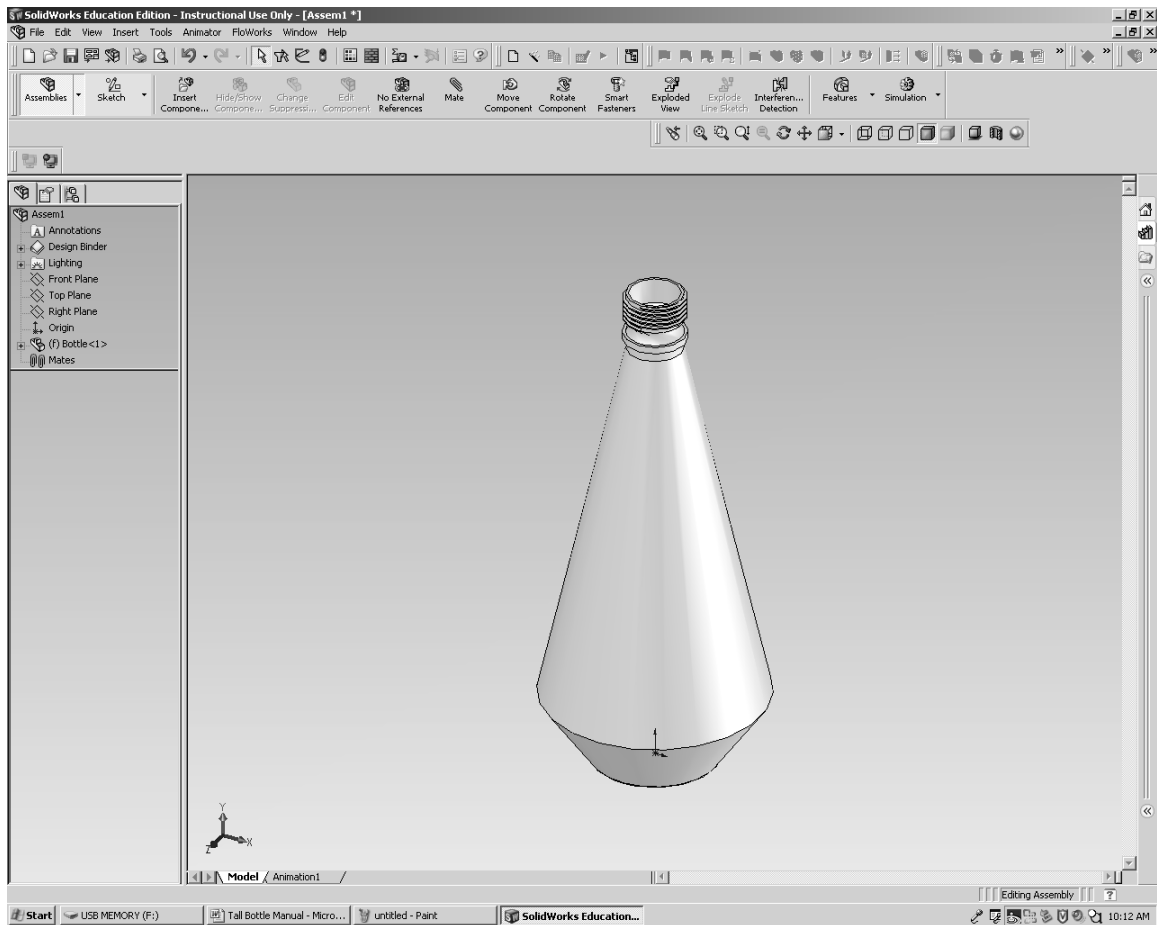
36. We now have a finished threaded cap. Now we will assemble the threaded cap and bottle together.



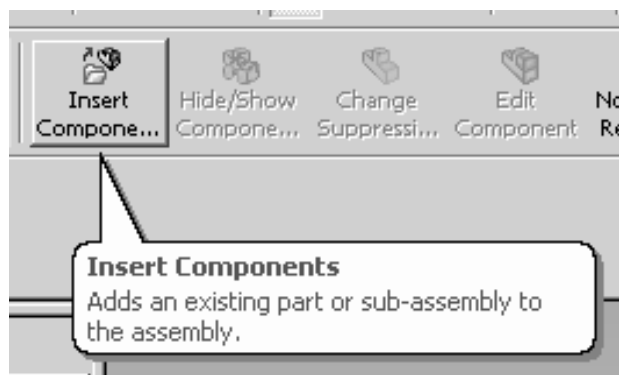
37. First, open a new Assembly File.



38. Under Part/Assembly: to Insert, click Browse and find your bottle. Open it. You can drop it anywhere on the screen. Notice if you roll your pointer over the origin, the part's origin snaps to it. This anchors the part.

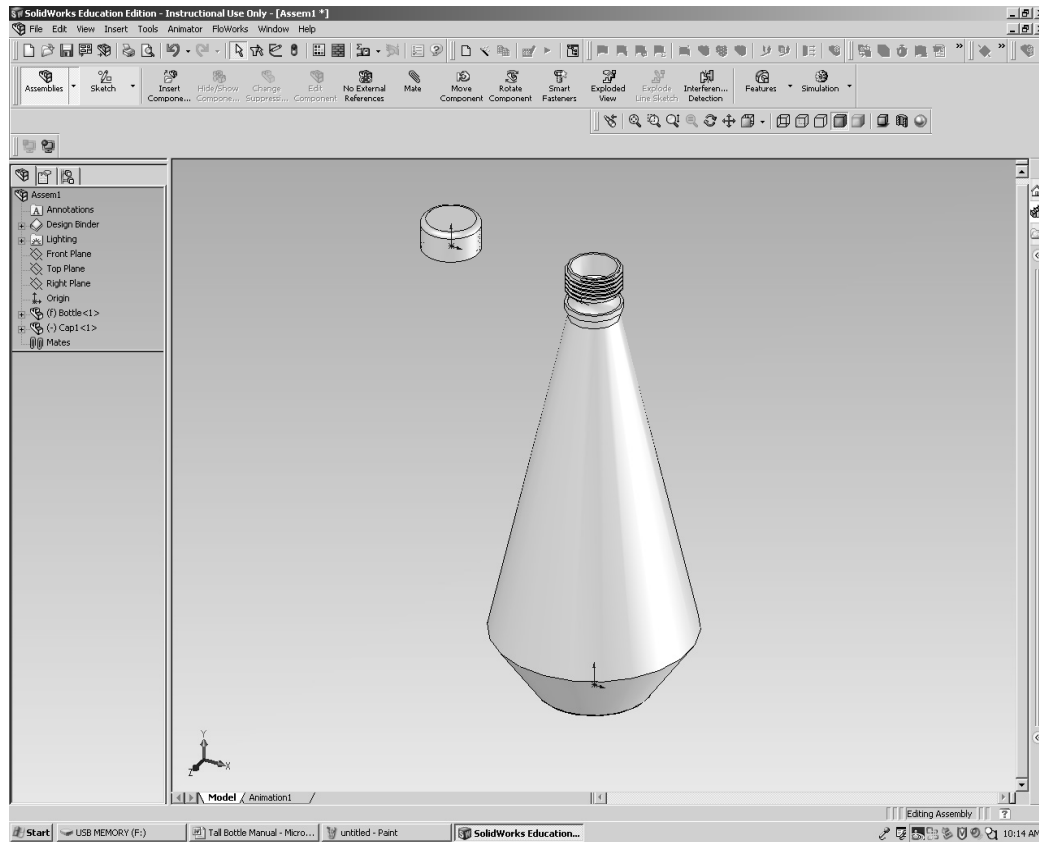


39. Now click Insert Components

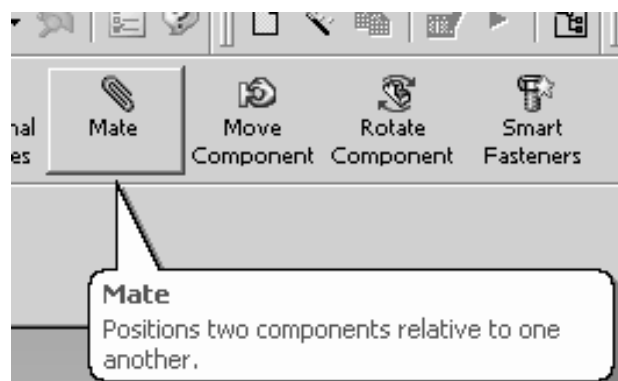




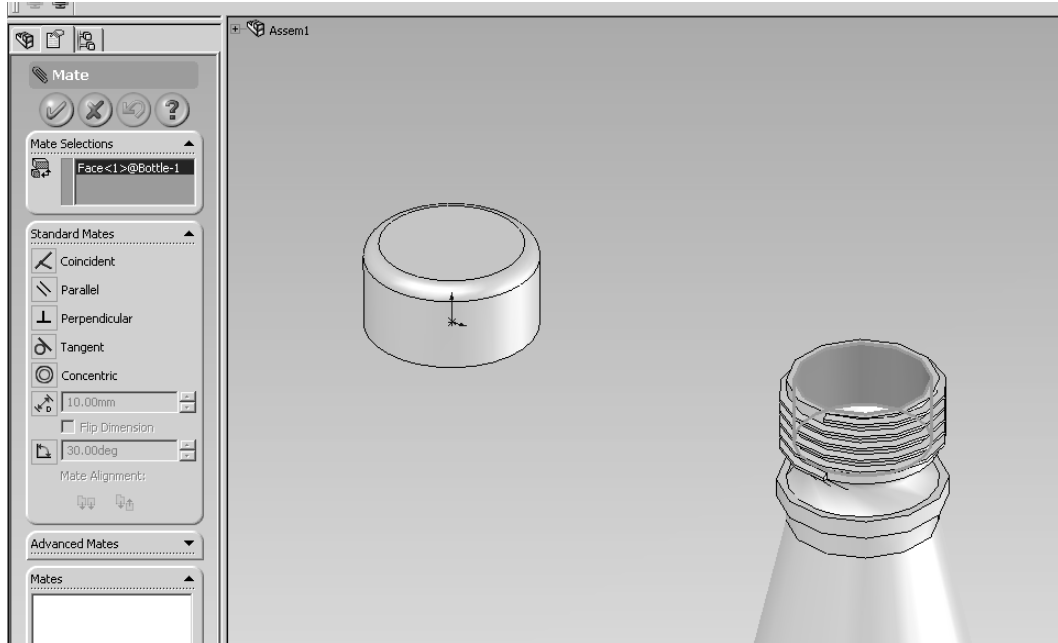
40. Find your cap file, open it, and drop it somewhere on the screen.



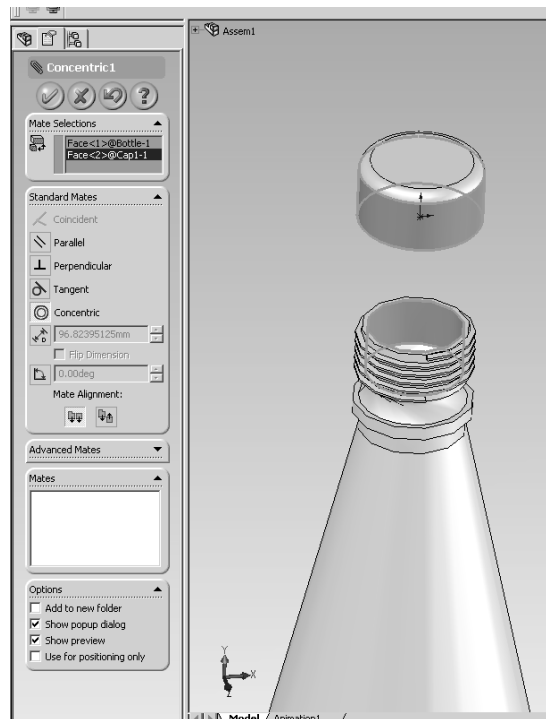
41. To bring the parts together we will use the Mate tool.



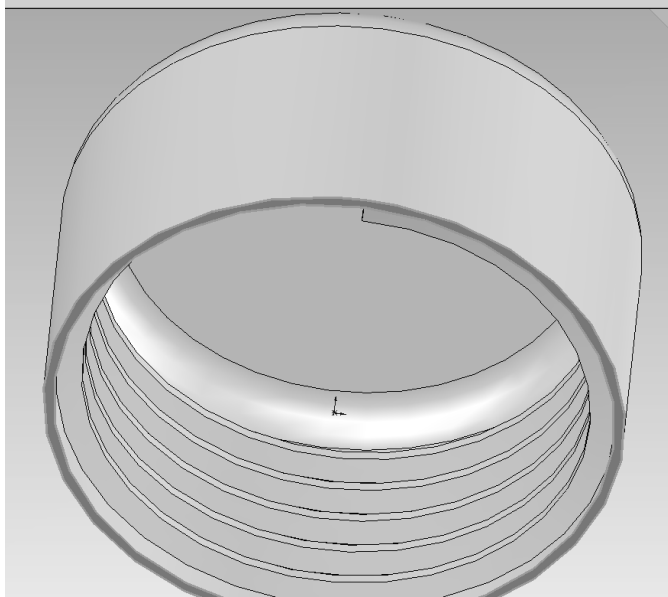
42. Click on the Mate tool and select the inner part of the bottle neck.



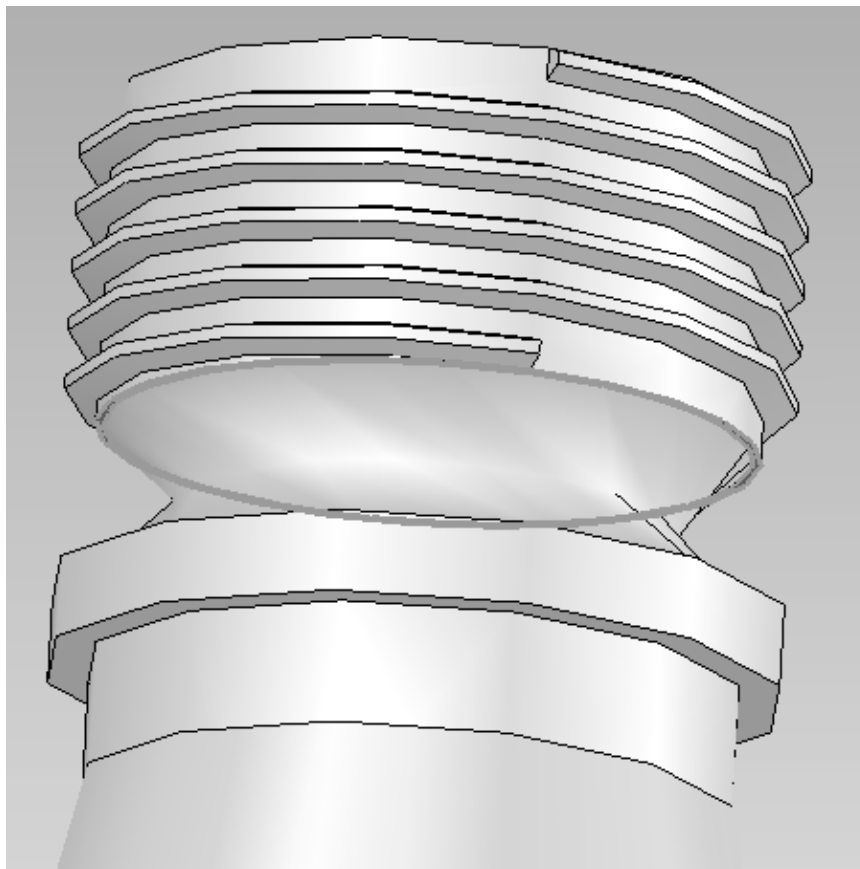
43. Next select the outer side of the cap. The mate automatically mates the parts with the mate it thinks you want. If it does not make the parts Concentric, do so under standard mates.



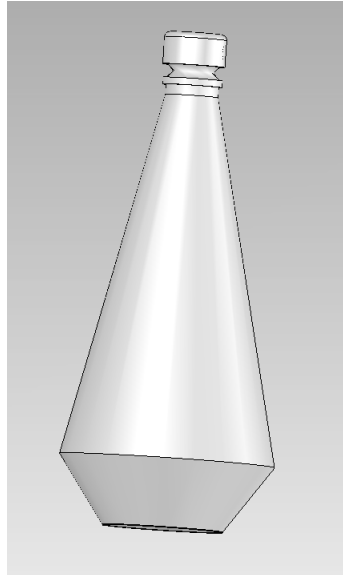
44. Click OK once. This sets the mate. Now rotate the parts so you can see the under side of the cap and select the bottom lip. You may have to zoom in.



45. Select the small lip under the threads for the second surface. Make sure the Mate is Coincident and click OK twice.



46. The cap and bottle are finished.



Stretch your thinking:

Instead of using Offset surface, please use Reference Plane to draw the bottle by creating reference plane in the aforementioned Step 5, 8 and 12.