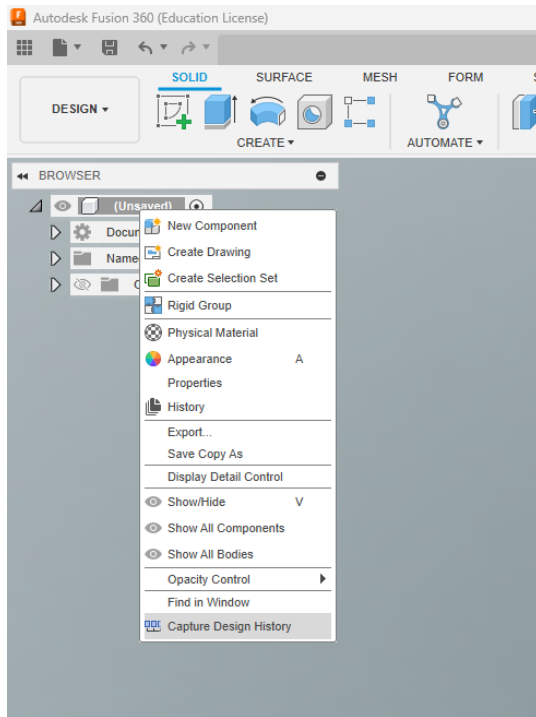
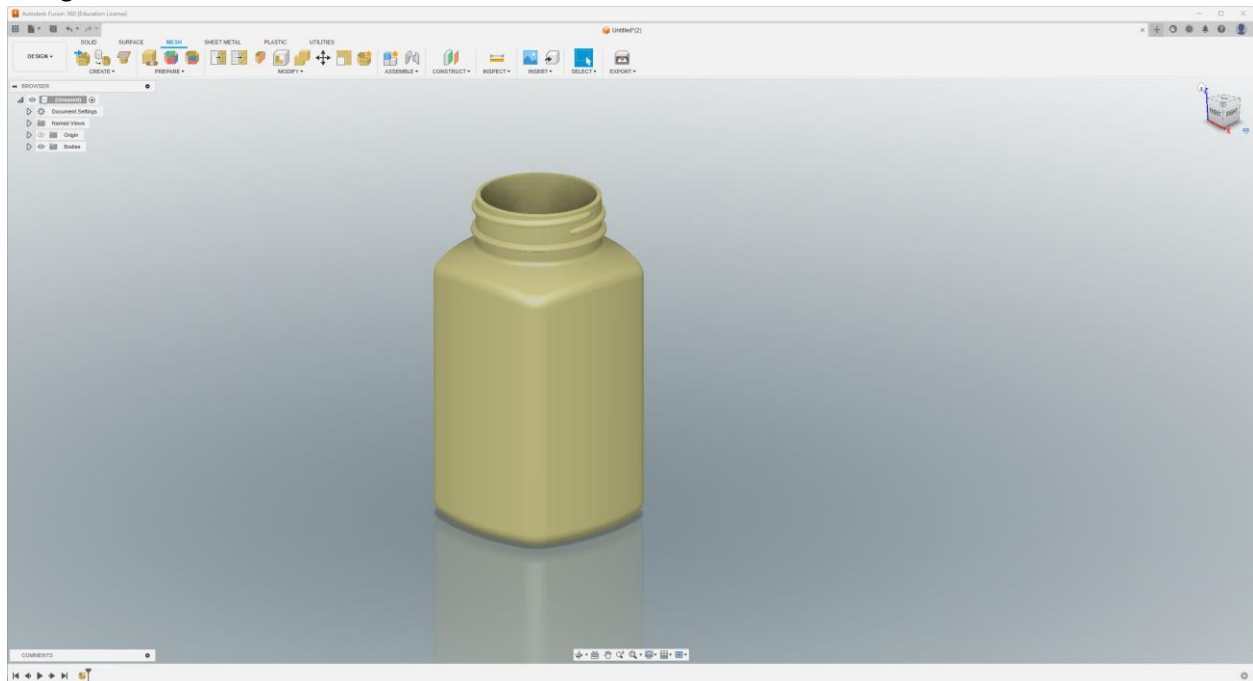


Bottle Modification using [Autodesk Fusion360](#)

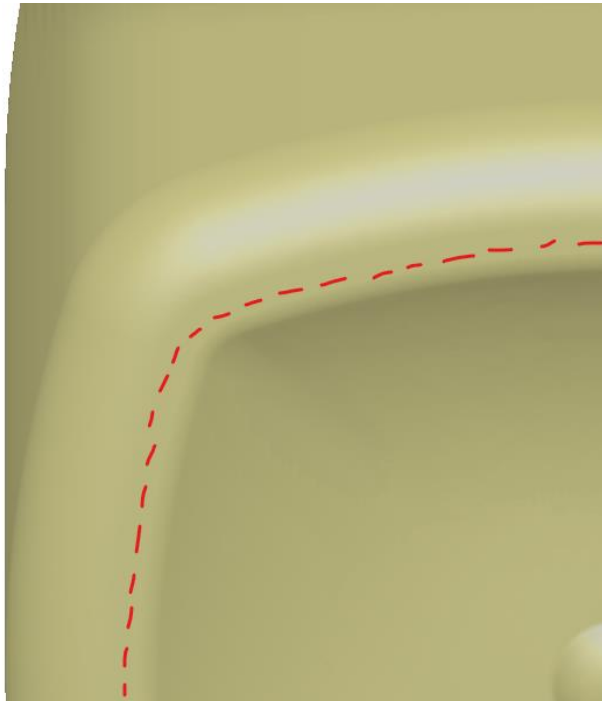
1. Open a new Design within Fusion360. Ensure that Design History is captured at the base of the page. This will allow retroactive edits to be made.



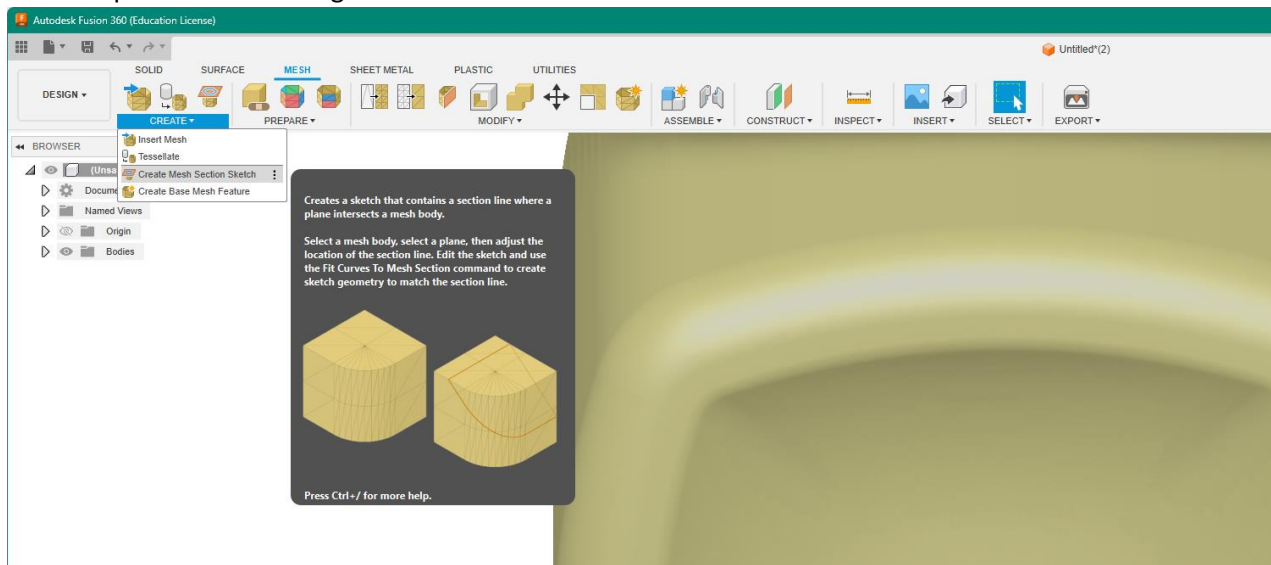
2. Navigate to the Mesh tab and insert the STL file into the scene.



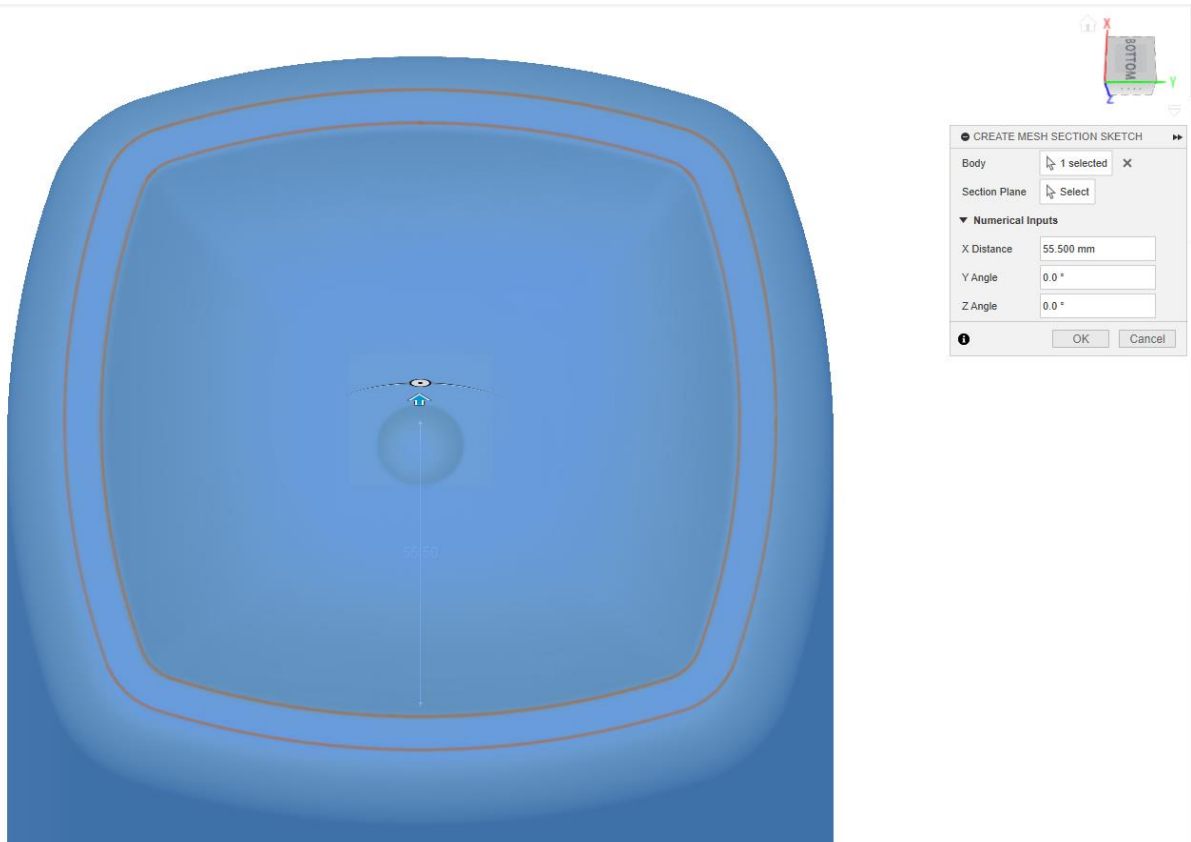
3. Determine where would be best to place the cut for the base. Make sure to leave enough space on the base of the part to place enough supports to adequately support part throughout the print. Following an existing profile for the cut can help to seamlessly reattach the base and looks best visually.



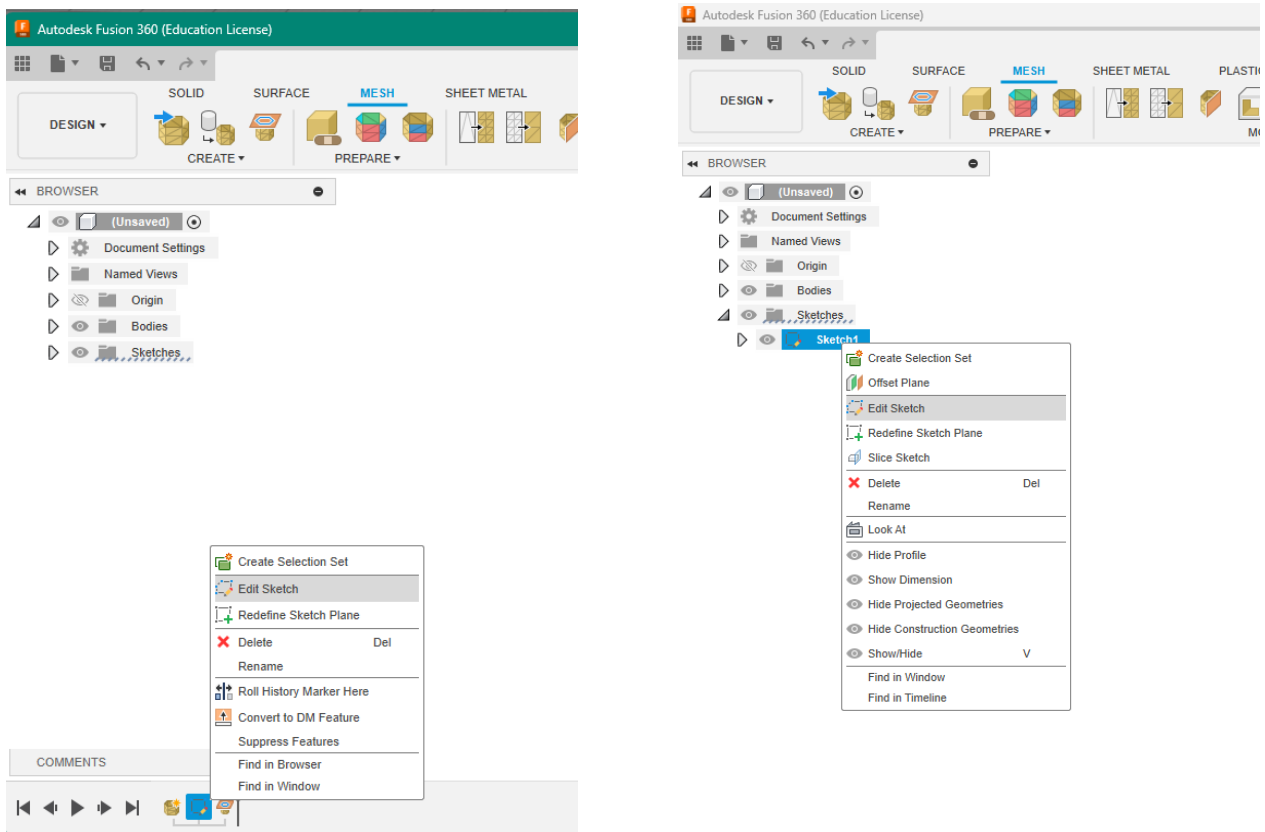
4. Create a profile sketch using the Create Mesh Section Sketch tool as seen below.



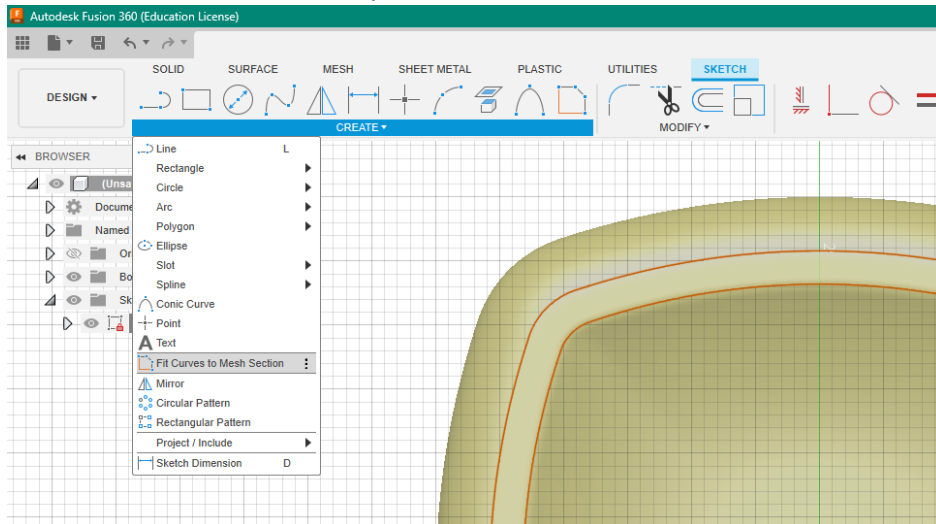
5. Use the Angle and Distance inputs to determine an adequate profile to create the base cut.



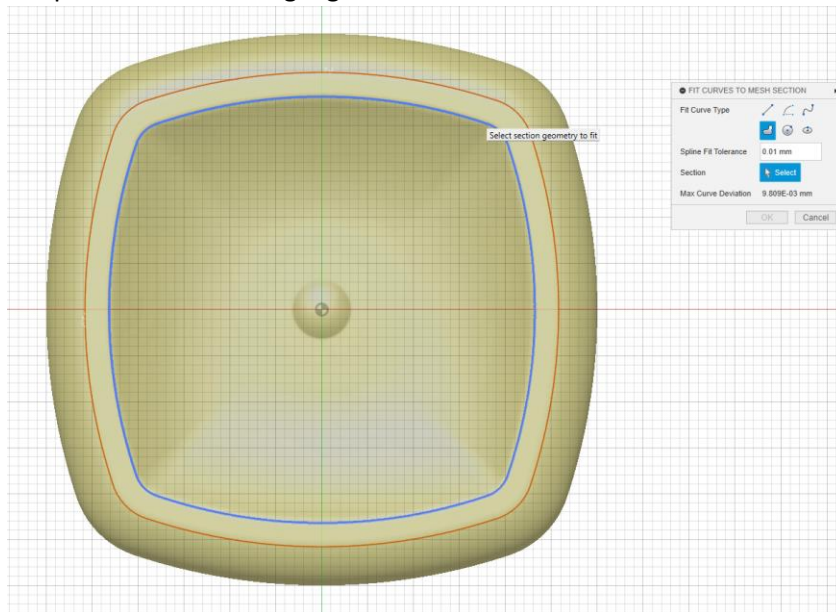
6. Edit the sketch that was created in Step 4 by using the design history bar or the navigation.



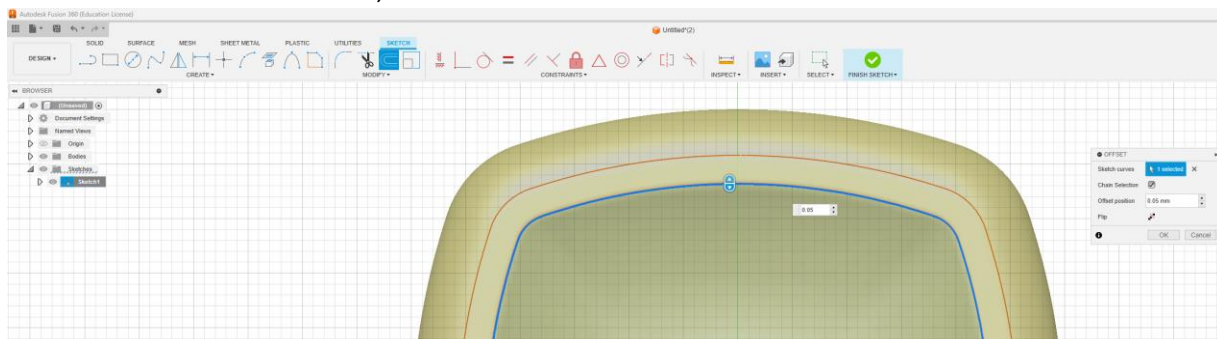
7. Once within the Sketch interface, select Fit Curves to Mesh Section within the Create menu in order to get usable sketch curves from the profile data.



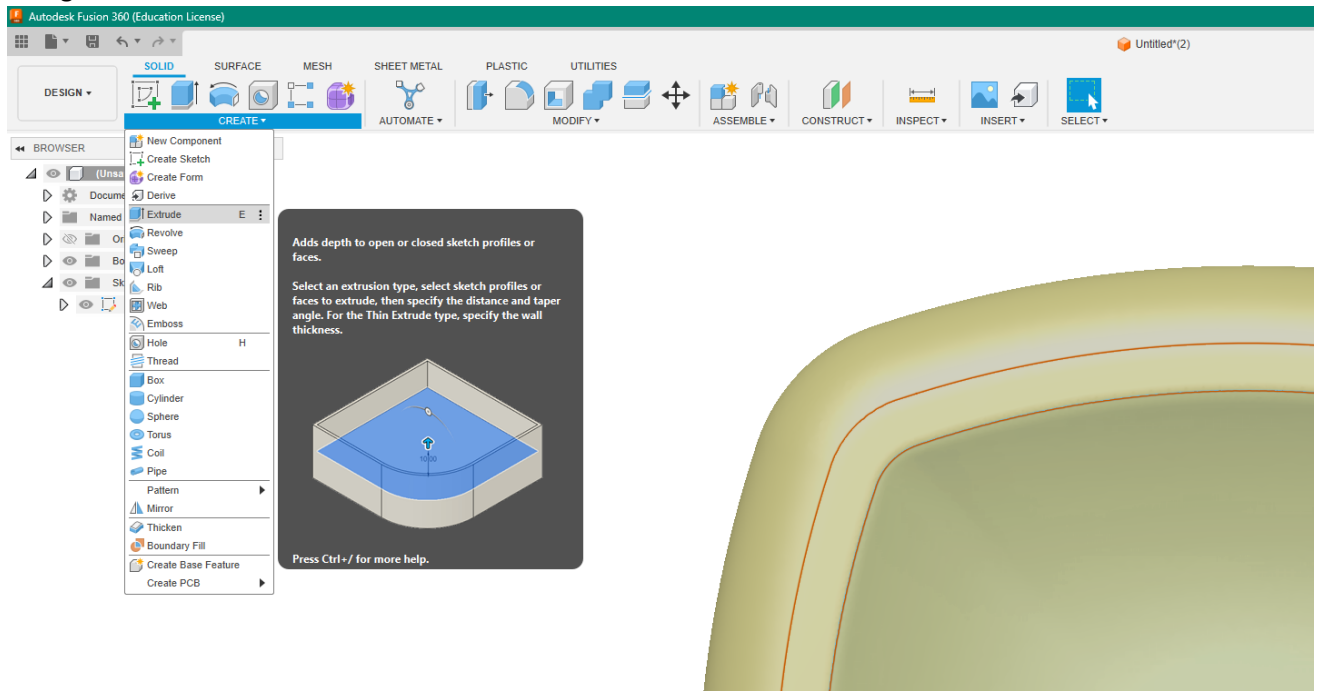
8. Select the profile where the base cut is desired using the Closed Spline Fit Curve Type. Once this tool is applied, the profile should be highlightable and able to edit.



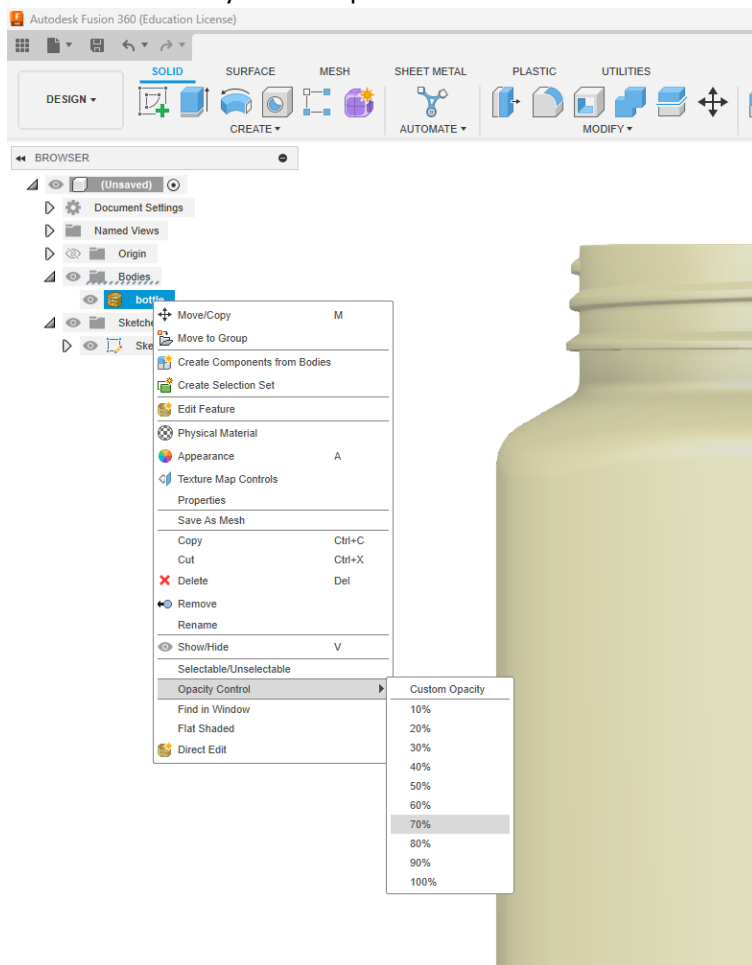
9. Use the Offset tool within the Modify menu to add an offset of 0.05 millimeters. Once this additional curve has been created within the sketch, select Finish Sketch to return to the Mesh interface.



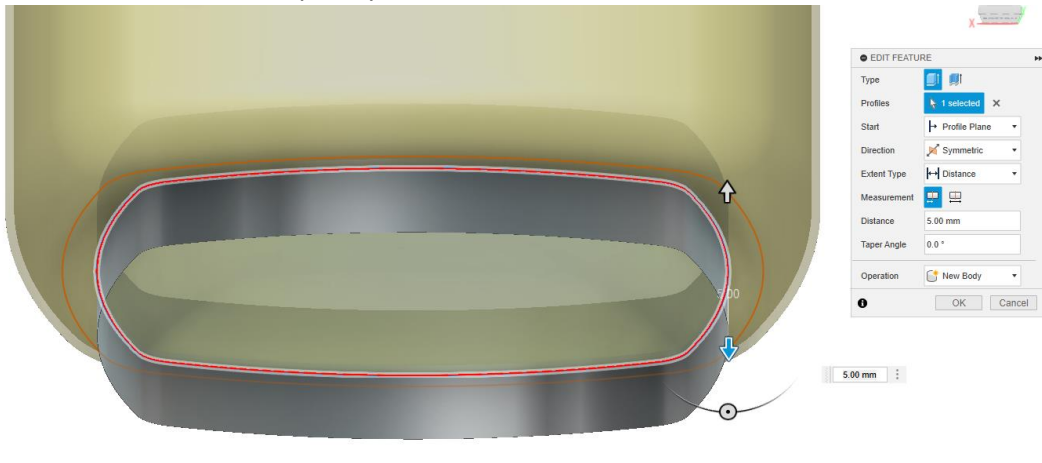
10. Navigate to the Solid interface and select Extrude from the Create menu.



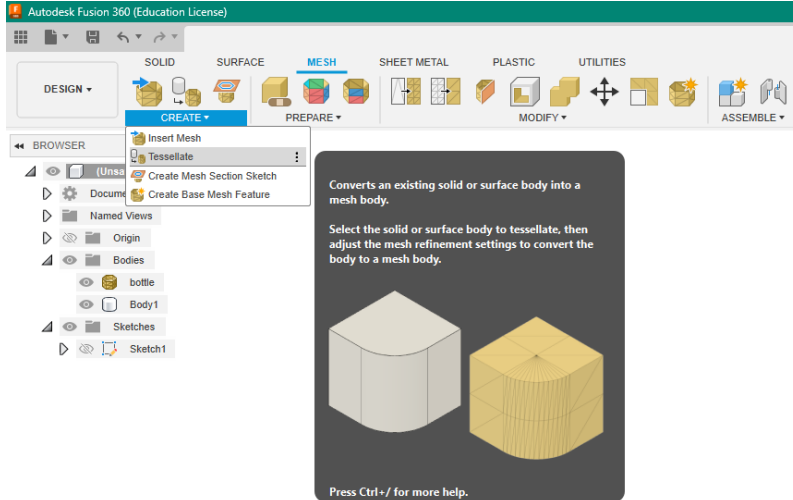
11. Change the opacity of the bottle mesh to 70% in order to ensure that the cut created in the following steps interfaces correctly with the part.



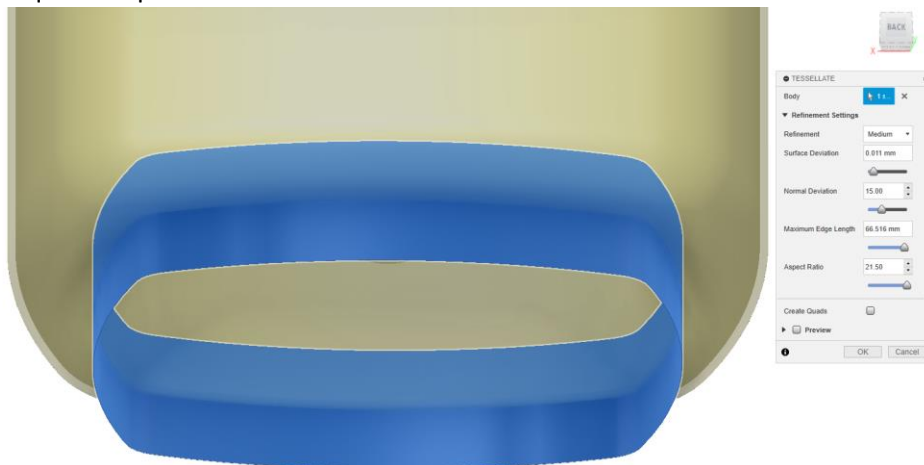
12. Select the profile just created in the previous sketch at Step 8. Ensure that only the profile between the two sketch curves is selected and not the center of the inner curve. Select Symmetric for Direction and ensure that the extrusion intersects completely with the bottle.



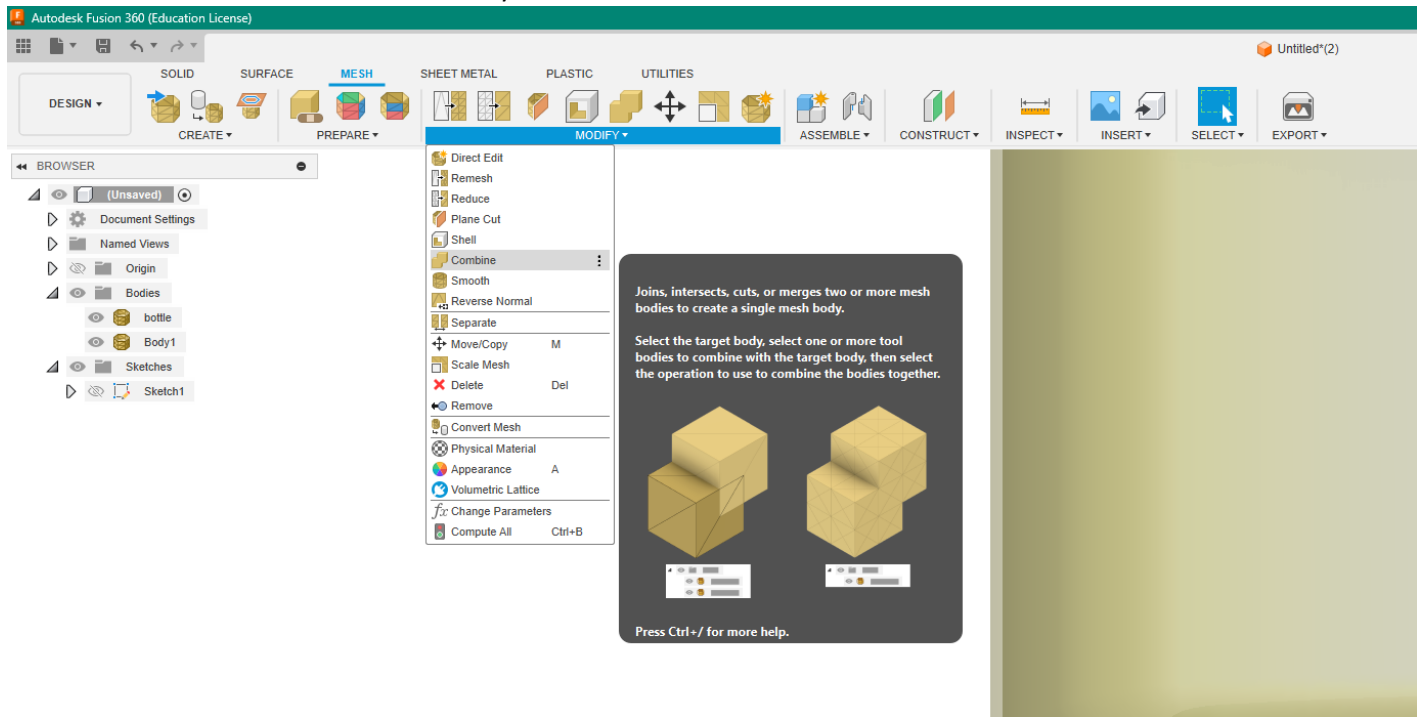
13. Convert the body just created to a mesh so that it can interact with the mesh bottle. To do this select, navigate to the Mesh interface and select Tessellate within the Create menu.



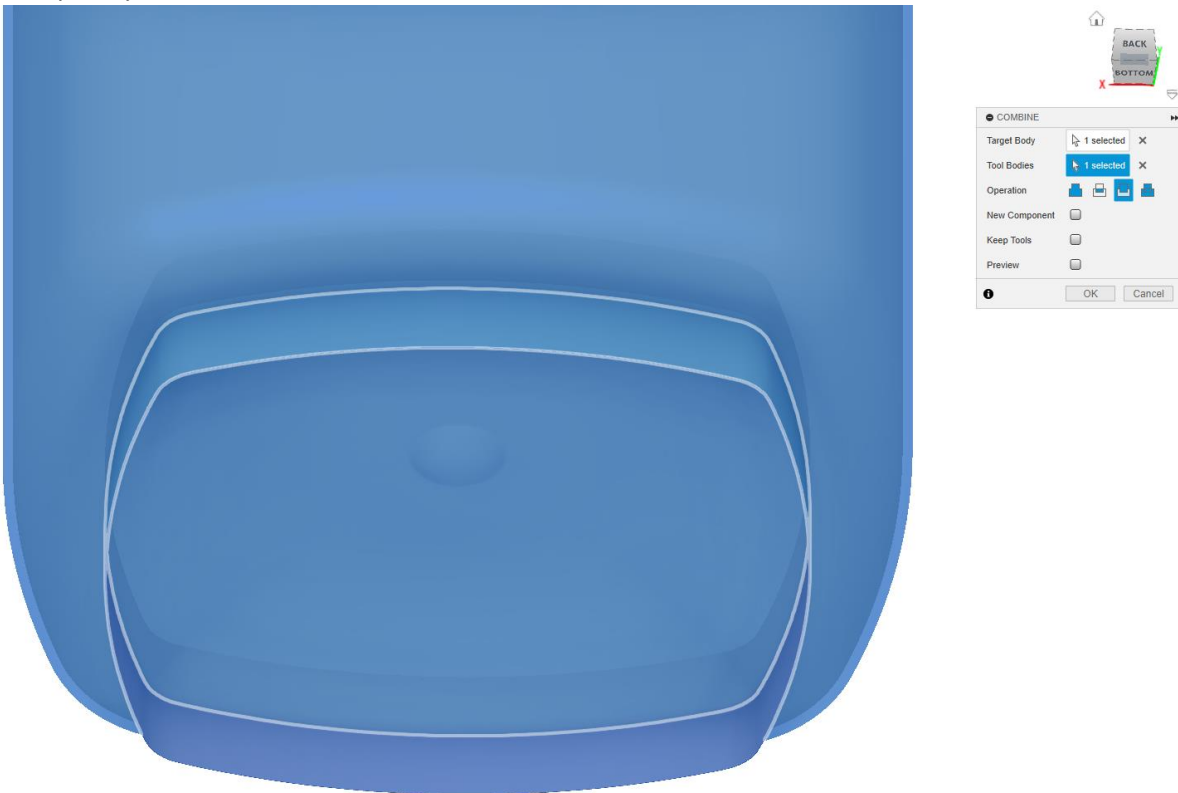
14. Select the body just created to convert to a mesh. The default inputs are okay for this conversion since the geometry is quite simple.



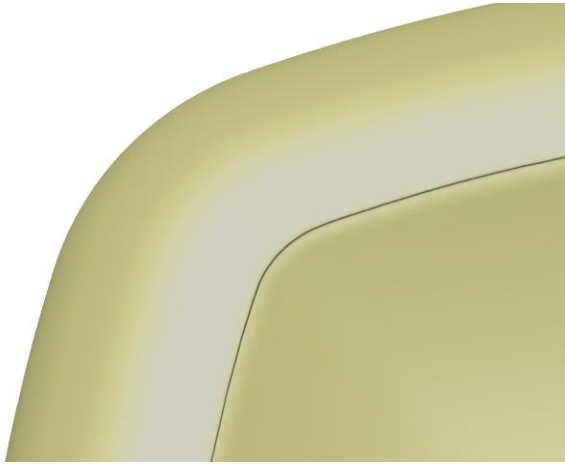
15. Select the Combine tool within the Modify menu.



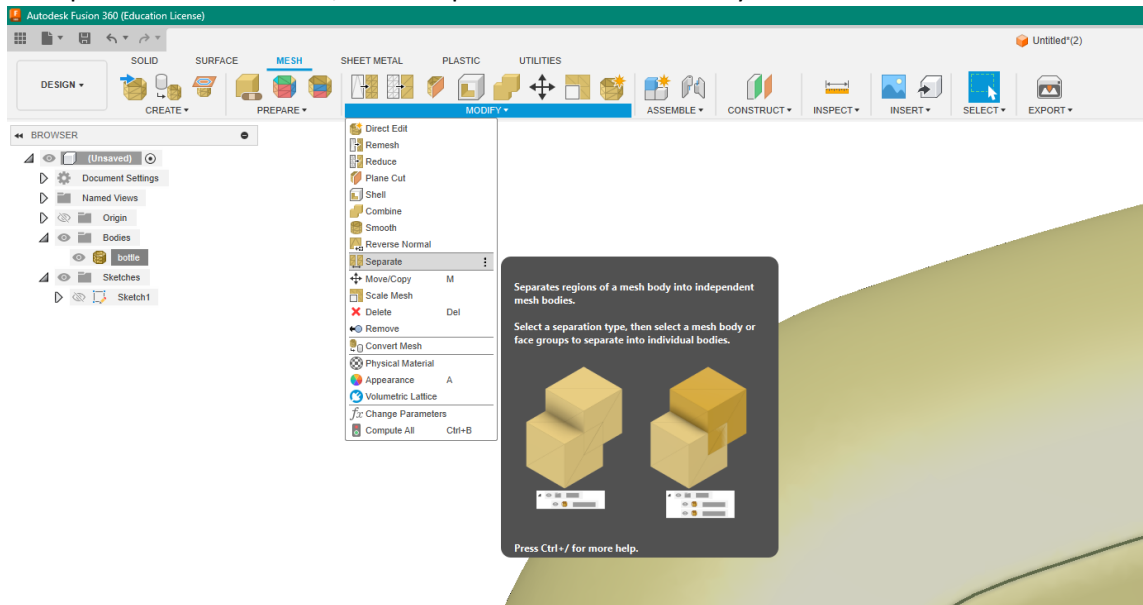
16. Select the bottle as the Target Body and the body just created as the Tool Body. Make sure that the Operation is set to Cut. It is not necessary to select any of the other options. This step can take a few minutes depending on the complexity of the mesh.



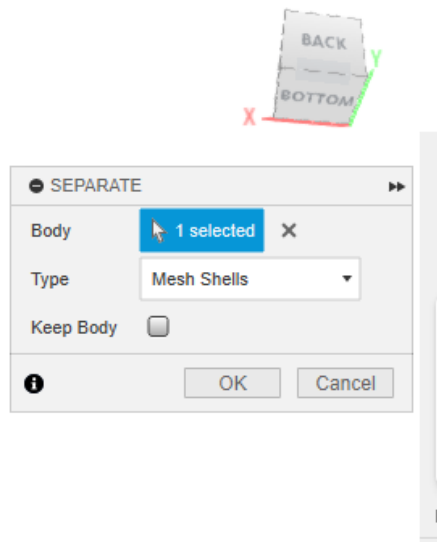
17. Once the previous step is completed, ensure that a small cut appears where the cutting body intersected with the bottle.



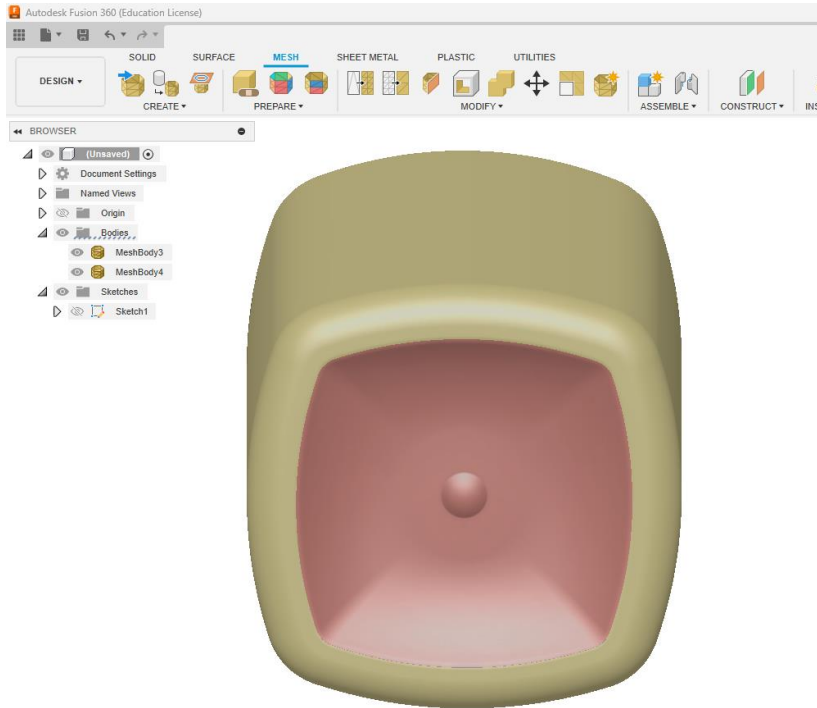
18. To separate the two bodies, select Separate within the Modify menu.



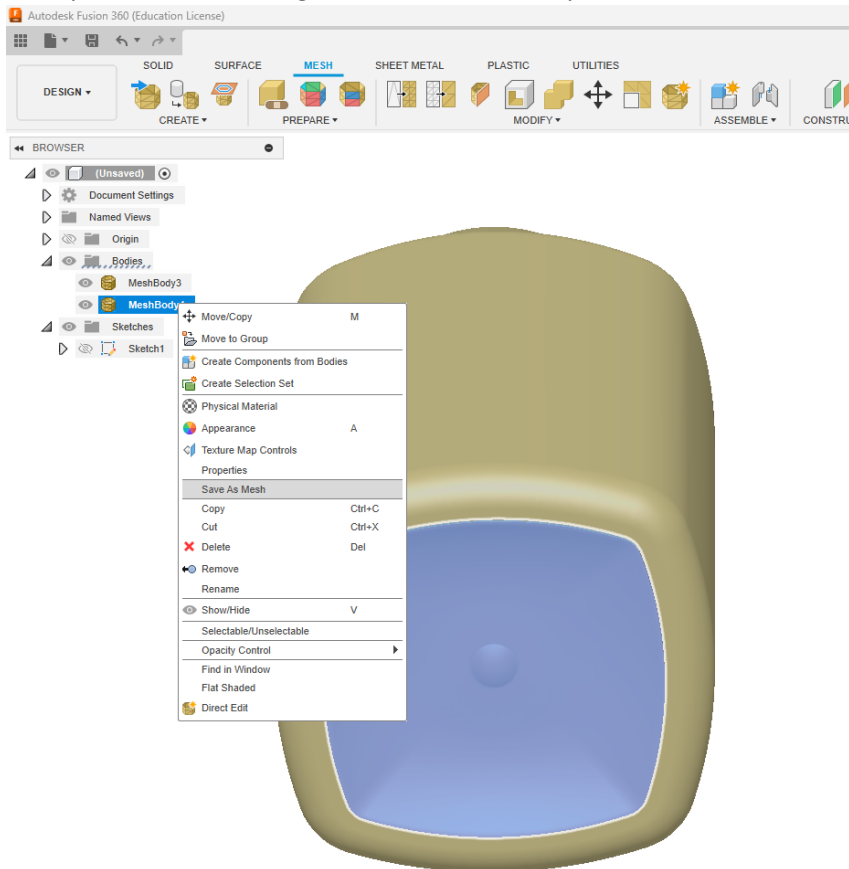
19. Select the bottle and ensure that the Type is set to Mesh Shells.



20. The bottle should now be separated into two parts, the bottle and the plug. They will appear as different colors and be shown as two bodies within the navigation menu.



21. To export the meshes, right click on one of the parts on the list and select Export as Mesh.



22. Ensure that the Format is STL and Unit Type is Millimeter. Select OK and save the mesh to desired location.
Repeat the step with the other part.

Select one component or a body

