

**Welcome to**  
**SOLIDWORKS**  
**Organic**  
**Shapes**

By Anthony and Adrian

# Introduction Slides



Adrien

Hyperxite

ASME Project Director



Anthony

ASME External Affairs  
Director

# TUTORIAL

Its time to apply what you've learned about Solidworks Surfaces, Organic Shapes, Profiles, and Boundaries to a real world example!

Open Solidworks and React with 👍 when you are ready to continue!



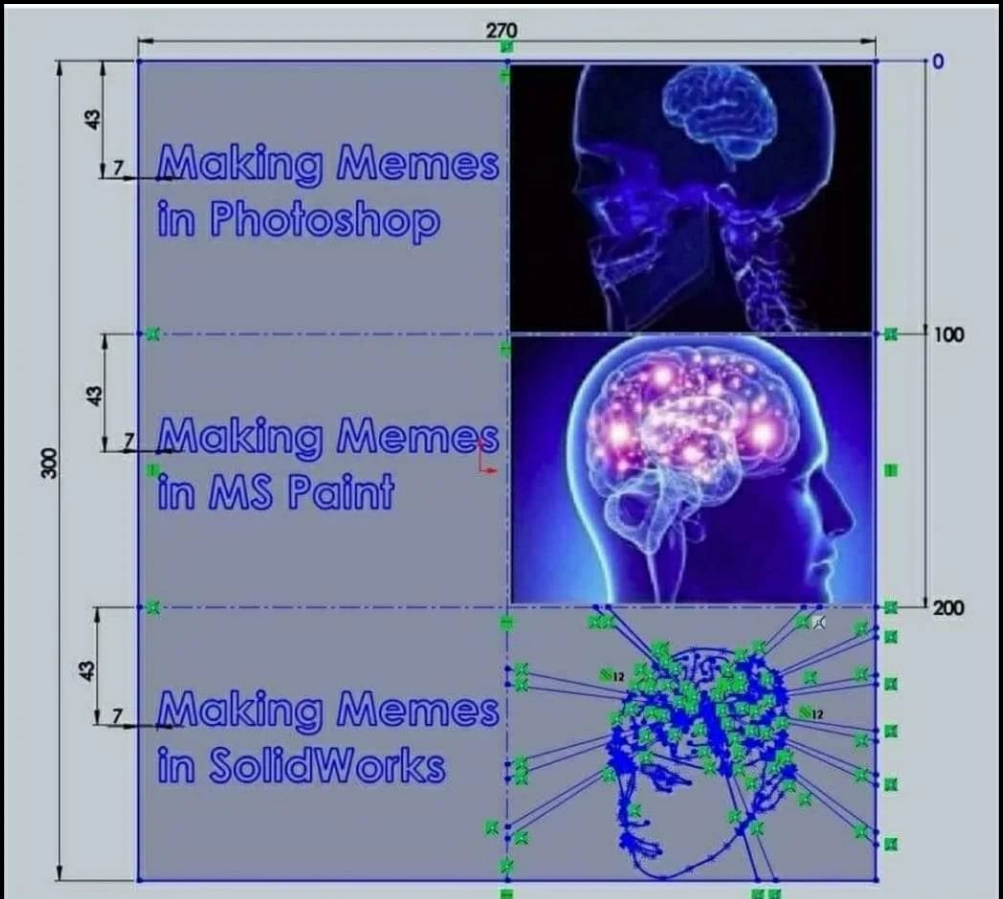
We are going to be making a bottle in Solidworks, that looks like this



# Start of Bottle Tutorial

To prepare:

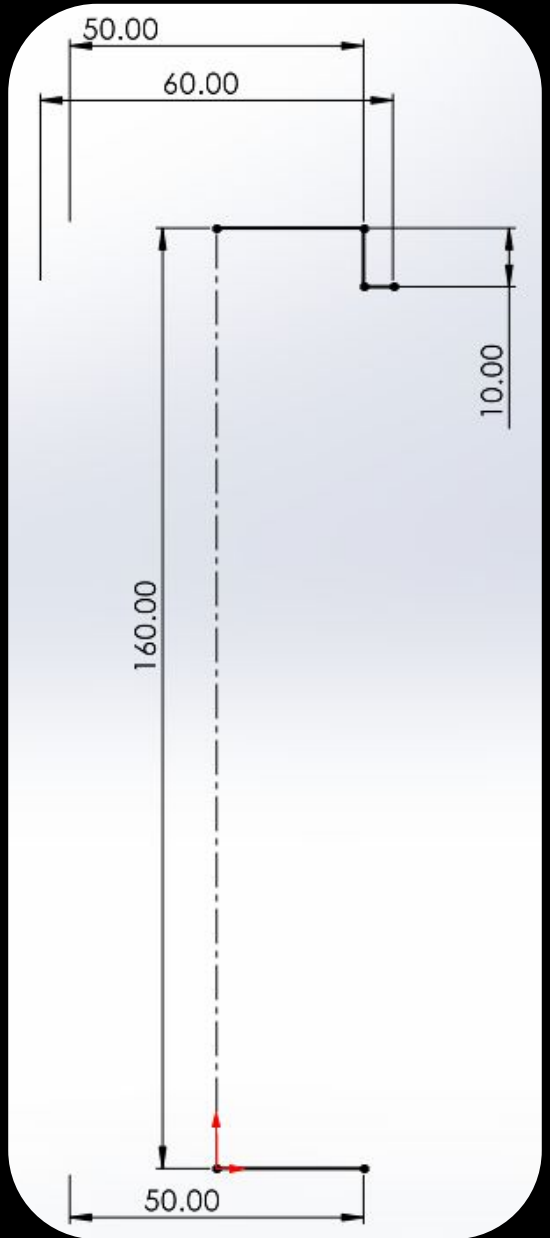
- Unit System MMGS
- First Sketch on the Front Plane
- Feel free to ask any questions
  - Solidworks hard :(
  - We here to make solidworks easier!



# Revolve Base Sketch

On the Front Plane  
Construct the following sketch:

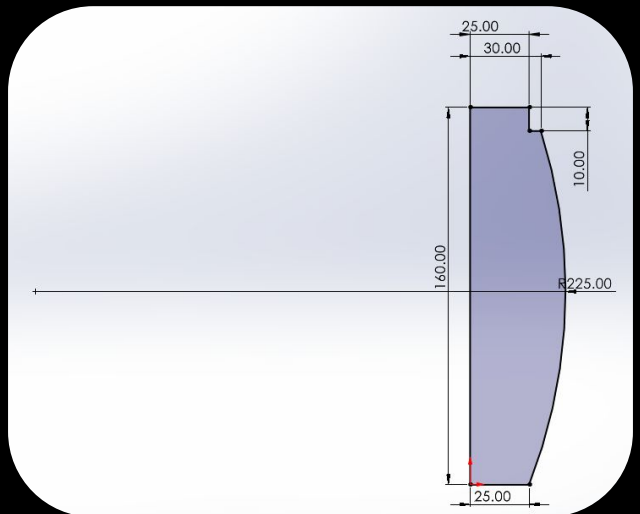
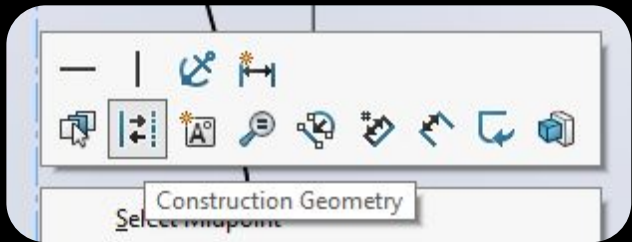
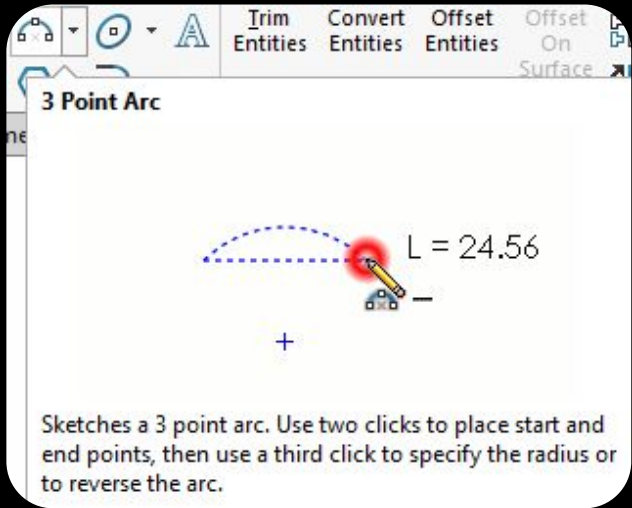
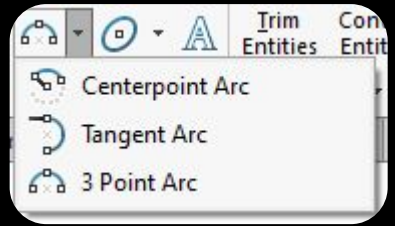
You can create diameters with the Smart Dimension tool by selecting the end points of the horizontal lines and the centerline



# 3 Point Arc Tool

Place the two end points (first two clicks) connecting the two end nodes of the sketch, and then use smart dimension to set the radius to 225 mm (Click on the arc and the radius tool will show up, like a circle).

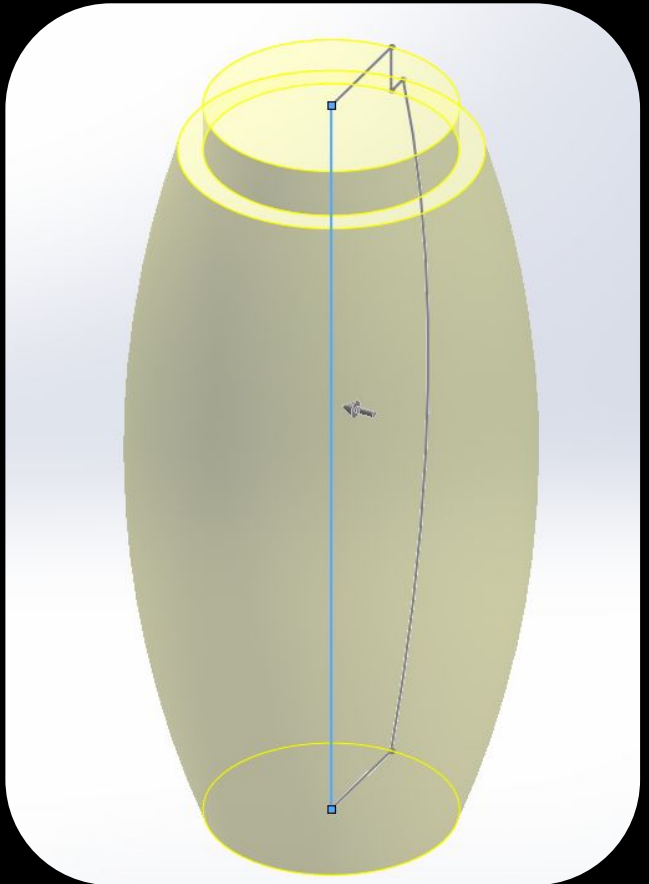
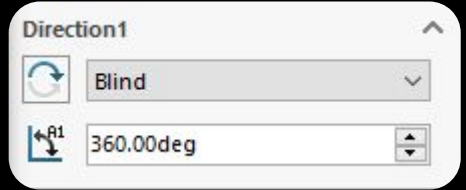
Also Convert Centerline to regular line.



# Revolve Boss Base Tool

Use the Revolved Boss Base Tool in order to create the three dimensional shape of our water bottle

Make sure to use 360 degree rotation!





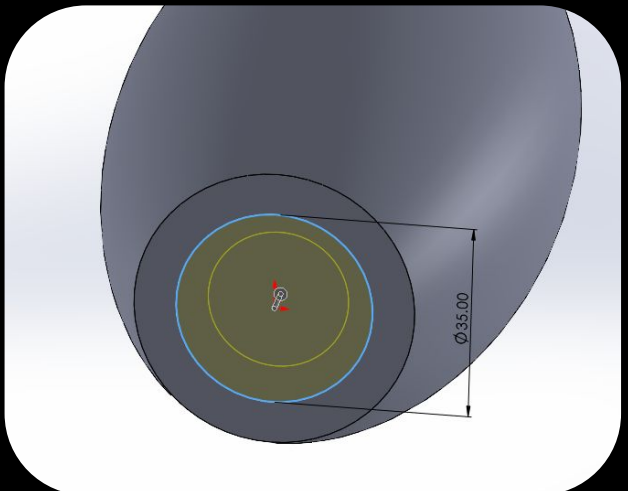
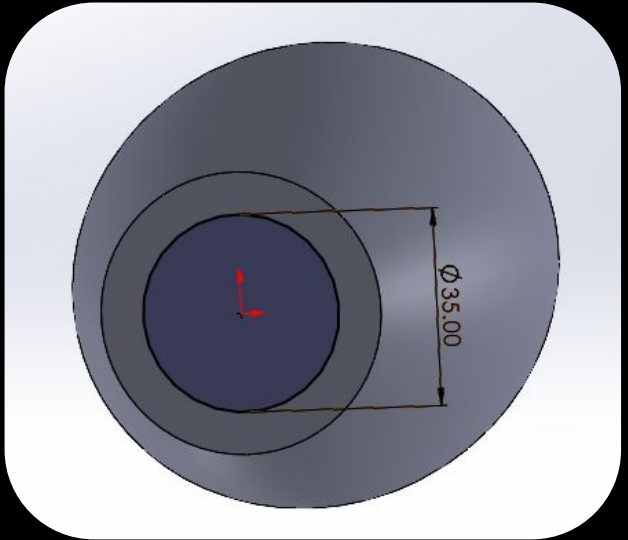
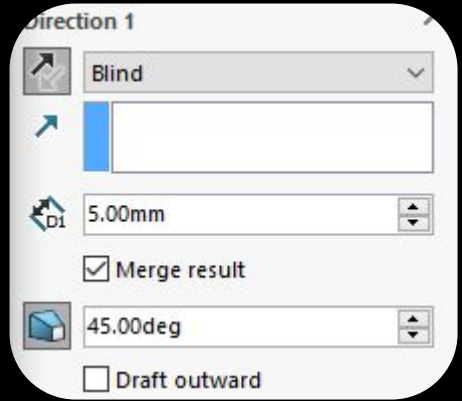
# Bottom Contour (Punt)

Extruded Cut  
on the bottom  
face of our  
water bottle

Make a sketch  
of a center  
aligned circle  
of 35 mm  
diameter

Use the  
Extruded Cut  
Tool with a 5  
mm distance  
and 45  
degree draft

😬 ! A Punt  
allows a bottle  
to stand  
easier than an  
imperfect flat  
surface

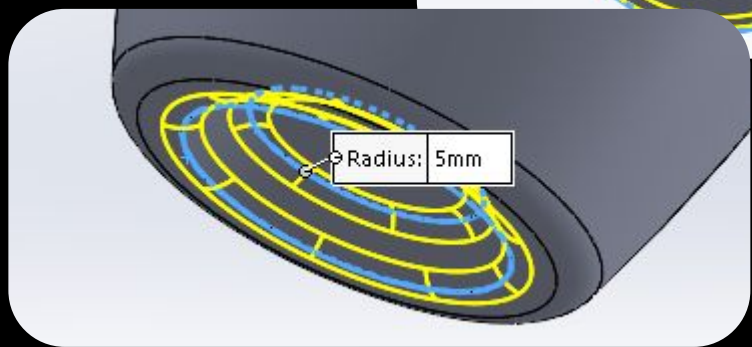
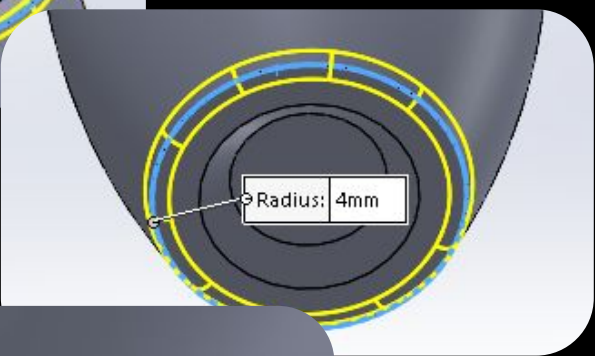
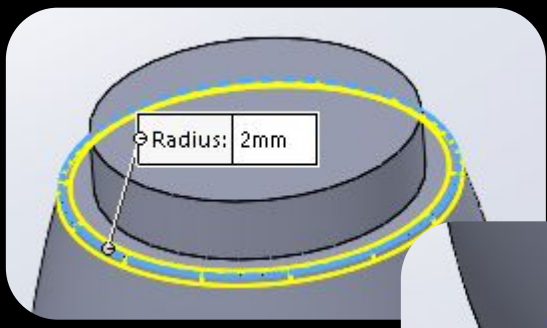


# Fillets are your friend

Sharp material objects are not great for things that are to be used by a human

We are going to add 3 different fillets to make our bottle more human friendly:

- Top: 2 mm fillet
- Bottom: 4 mm fillet
- Punt: 5 mm fillet



# Recessed Contour

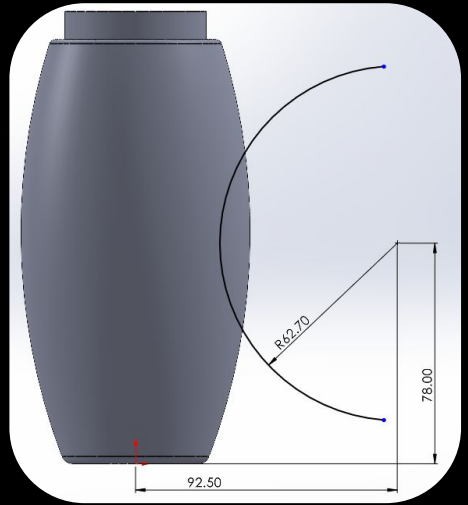
On Front Plane:

3 point arc on right side:

Radius: 62.7

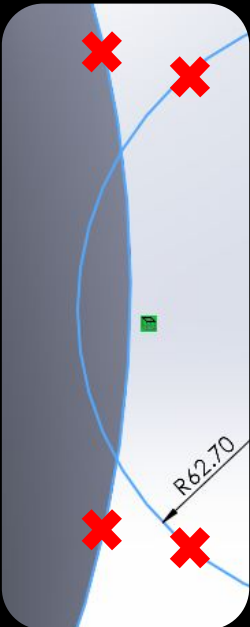
X dist: 92.5

Y dist: 78



Convert Entities Tool:

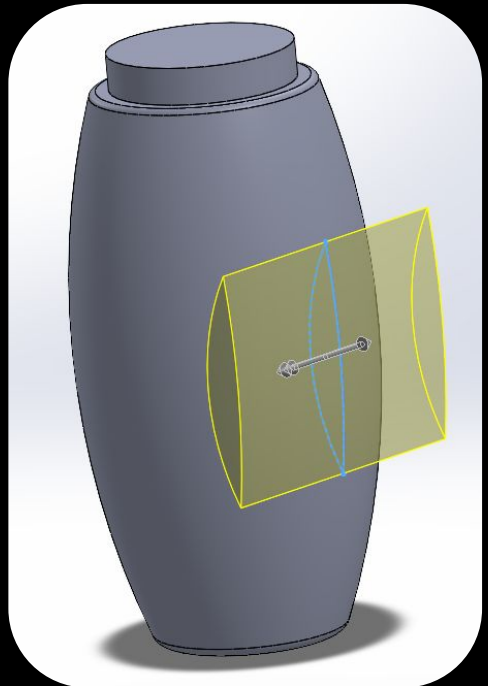
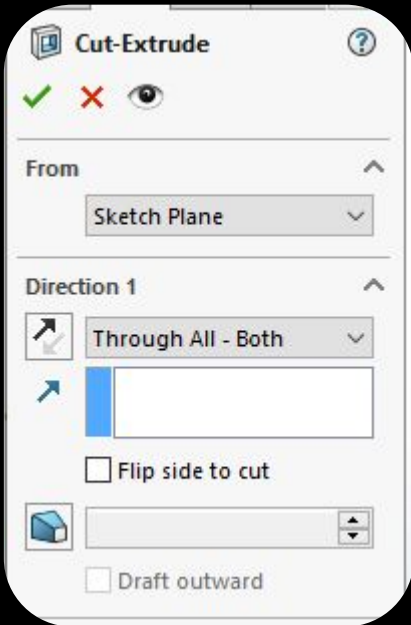
Convert the right most 3D curve of our bottle into a 2D sketch entity we can manipulate



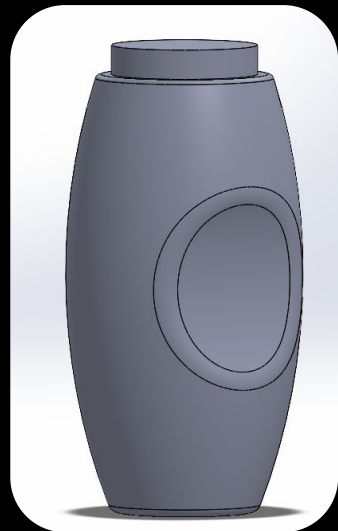
We can now trim the lines we don't want away

# Recessed Contour Continued

Extruded Cut  
"Through All - Both"



10 mm fillet in order to remove sharp edges



# Spline Contour – Construction Geometry

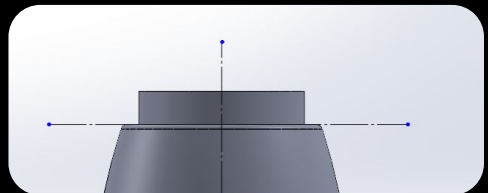
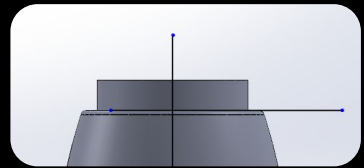
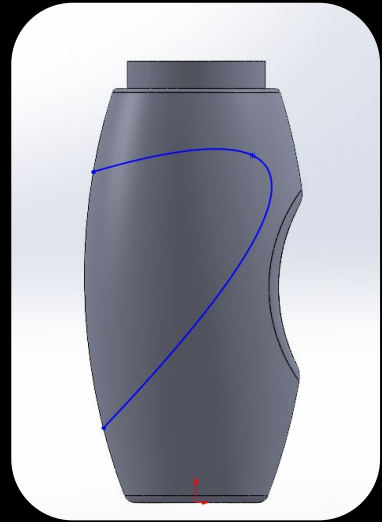
On Front Plane:

Use the Spline tool to create a spline that is coincident twice on the left edge of the bottle

## Construction Geometry for Spline:

To convert a standard line to a construction (reference) line use the convert to construction geometry feature

Make a line on the vertical center, and another one horizontal to the top lip of the bottle, you'll have to create the first part of the line on the horizontal lip, and then extend it

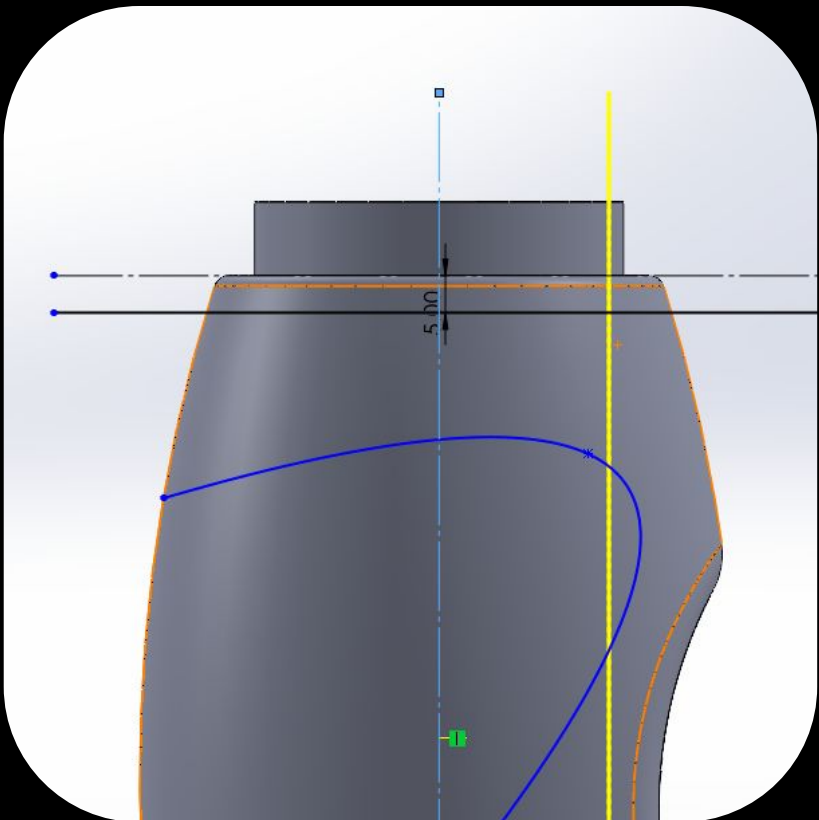


# Spline Contour – Offset Entities

The offset entities tool allows us to create new entities offset by a certain amount from an axis or already existing line



Offset the top line by 5 mm and vertical line by 23mm like so

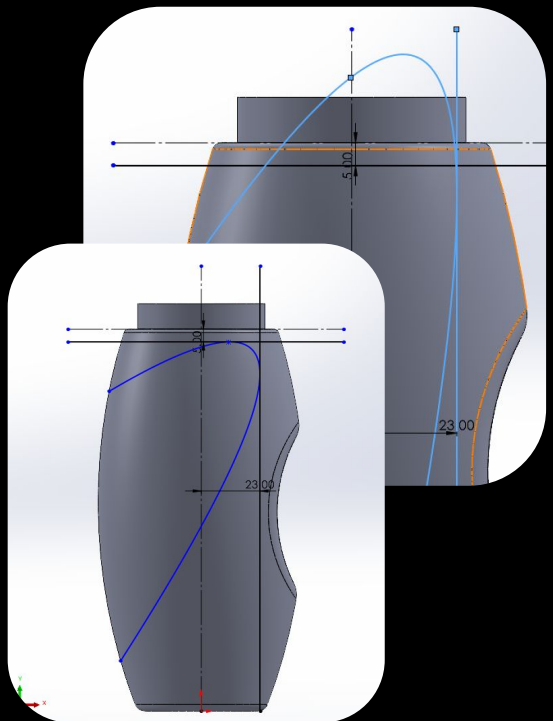
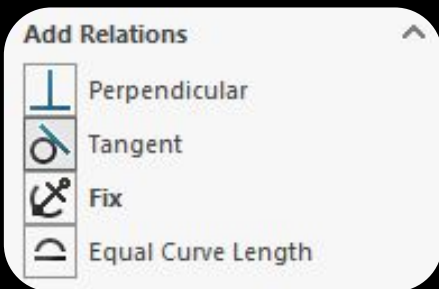


# Spline Contour – Relations (2D “Mates”)

Now that we have two lines that represent the bounds of our spine, we can use them to restrict the spline itself by using relations

Start by clicking on the sketch contour and while holding down control or shift click on the keyboard, click on the reference line

In the menu that pops up, select **tangent**

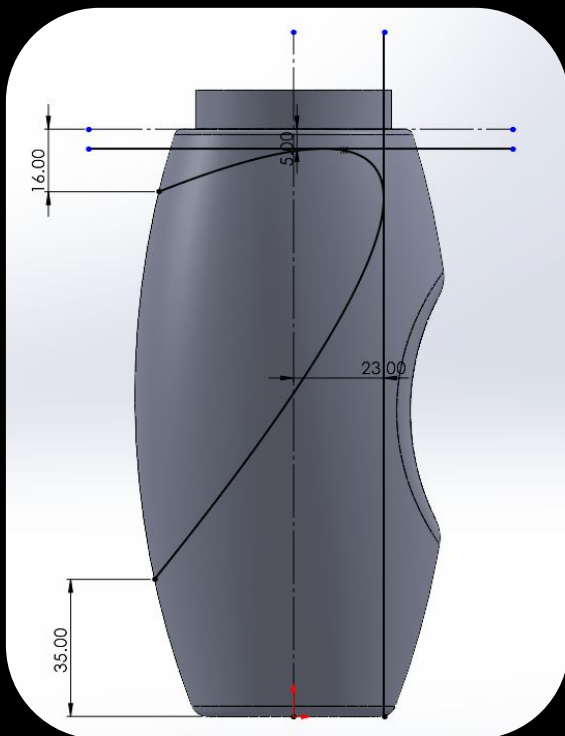


# Spline Contour – Loose ends

Smart dimension the upper point 16mm from the top reference line, the lower point 35mm from the lower face of the bottle

Remember to set the unneeded lines as reference geometry!

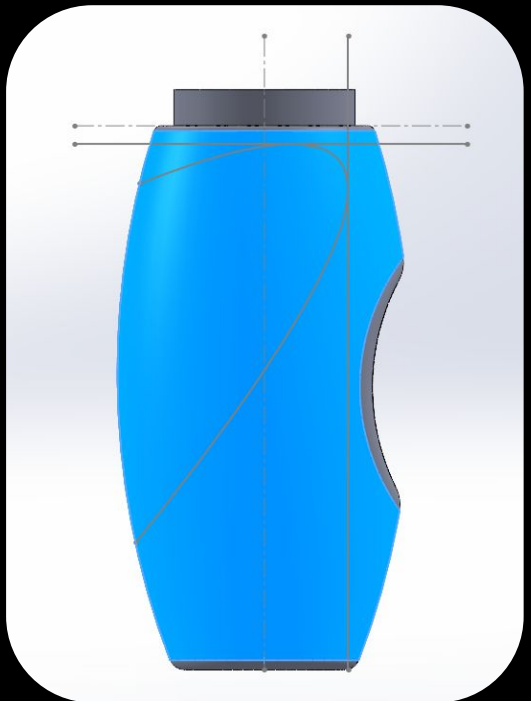
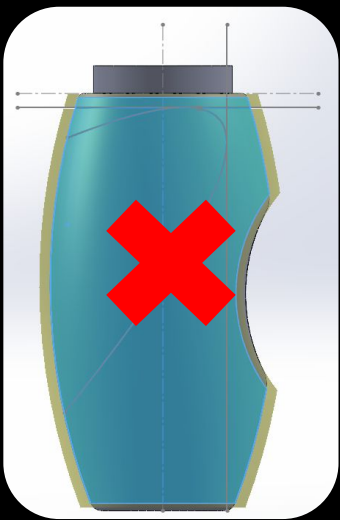
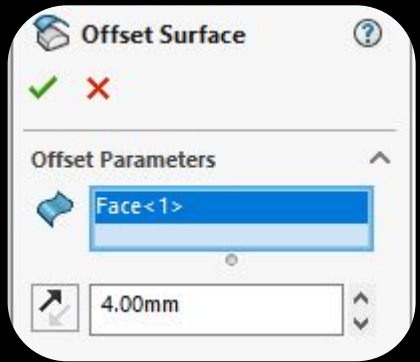
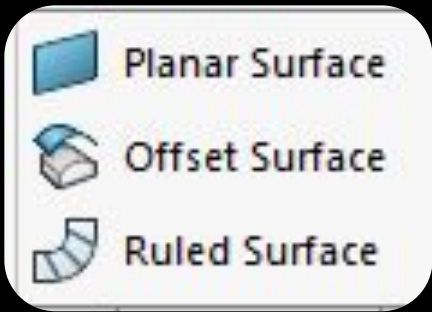
The spline should now be **fully defined** (Denoted by black color instead of blue)





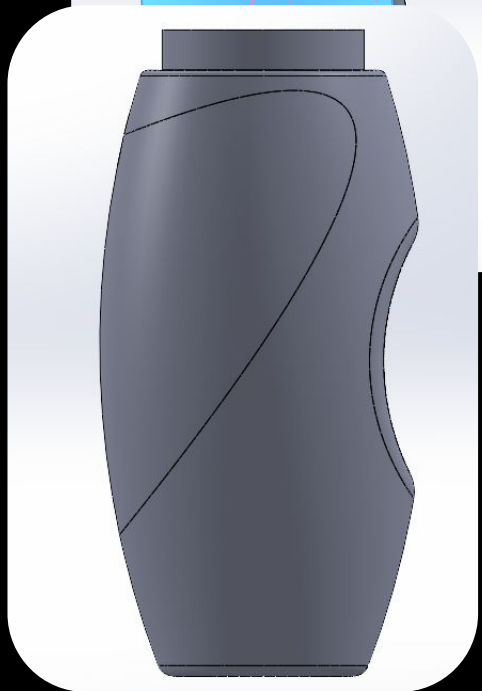
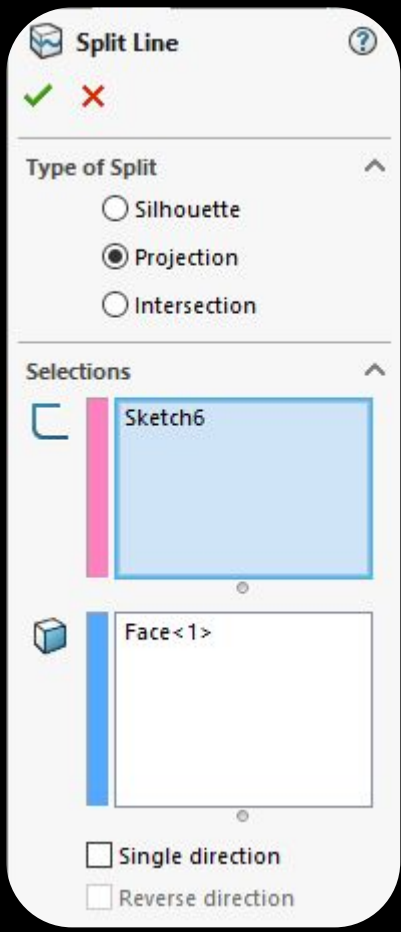
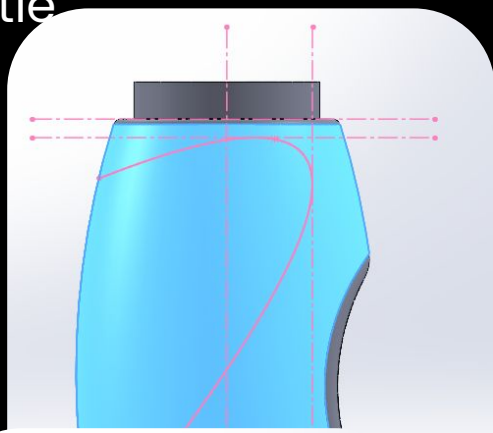
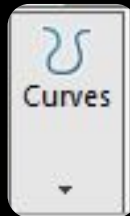
# Offset Surface

Create an Offset Surface of the Revolved Base with a dimension of 4mm **inwards**



# Split Line using Spline

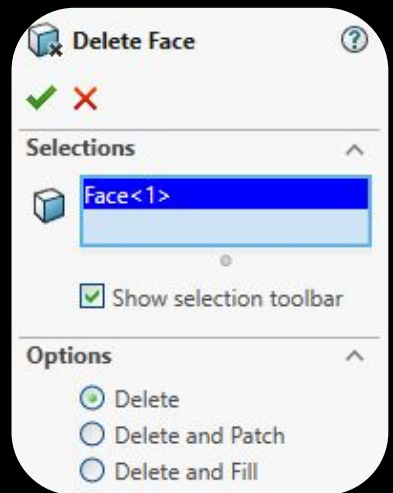
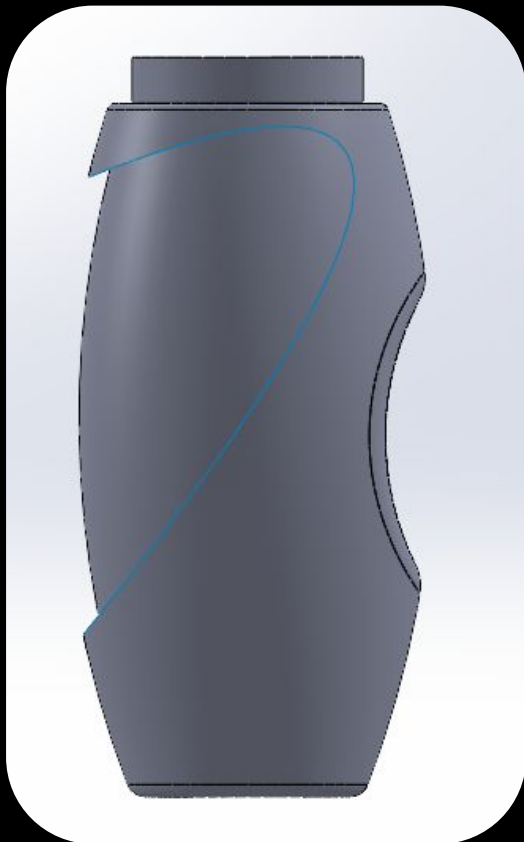
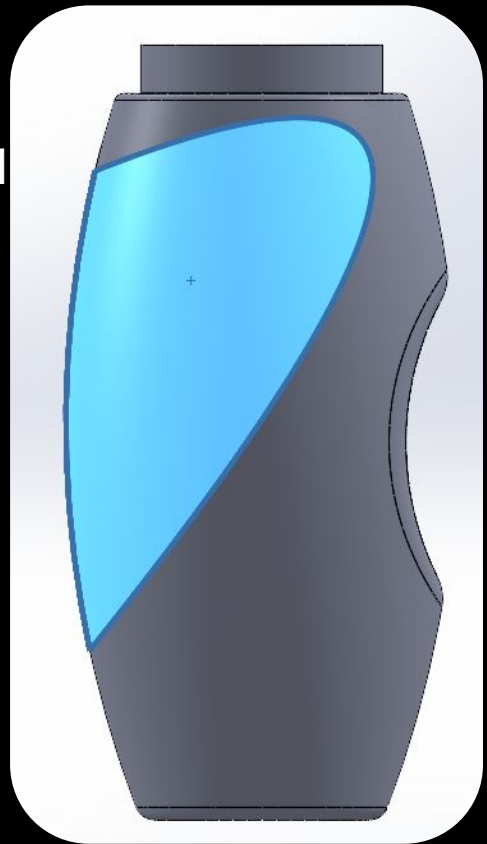
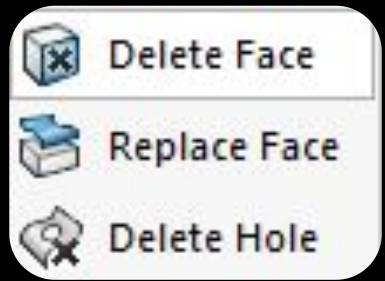
Using the Split Line tool, select the spline we made earlier and the outer face of our bottle like so:



# Delete Face

Now that we split the faces in two by our spline split line feature, we can remove the unwanted face easily with the delete face tool

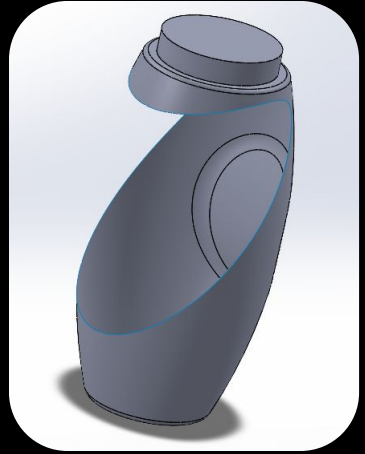
Make sure to get options right



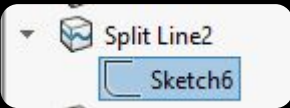
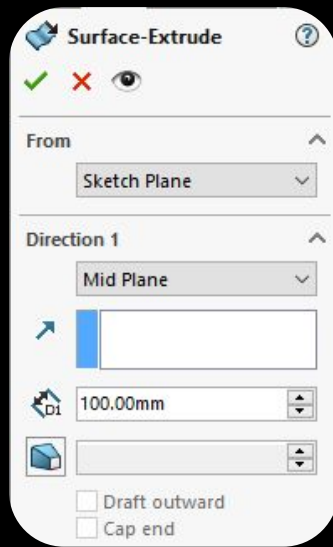
# Repairing the hole - Pt 1

## Surface Extrude

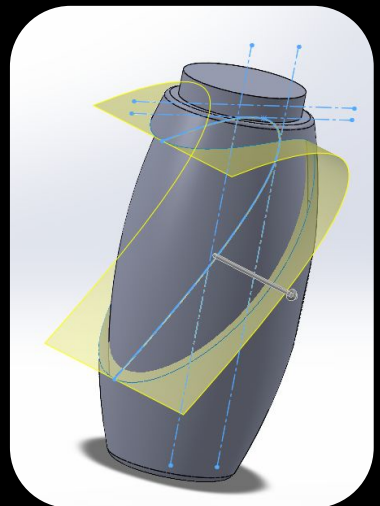
Because we deleted a face we now have a hole between the two surfaces (the original and the offset)



To correct for this we will use Surface Extrude on our original Spline sketch



With the Spline Sketch Selected use the Extruded Surface tool



# Repairing the hole - Pt 2

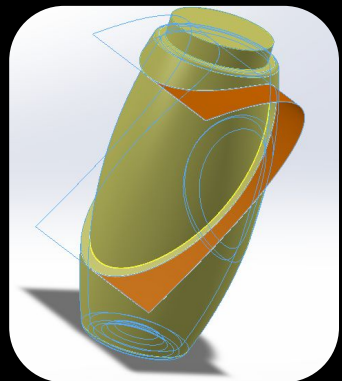
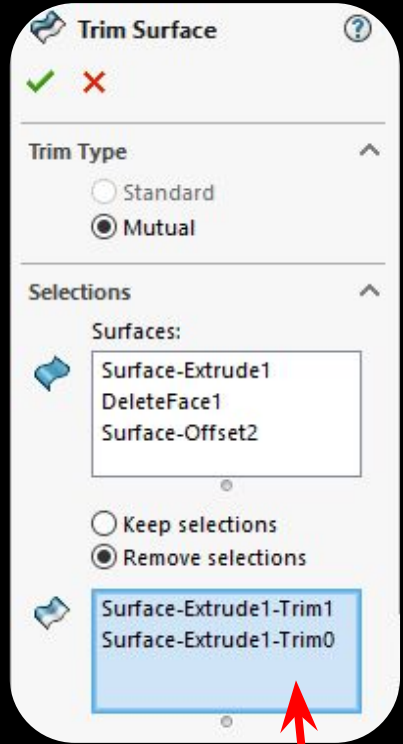
## Trim Surface

Now we have too much excess material, to remove this we can use the Trim Surface tool:



For the Surface Selection Select all of the surfaces (Control+A or individually)

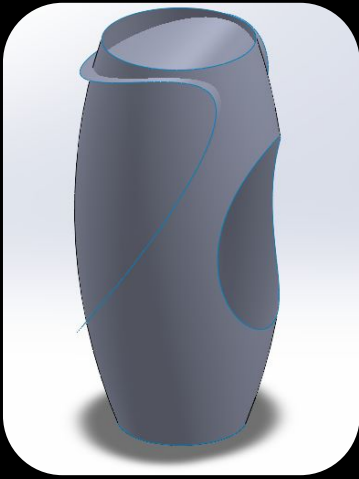
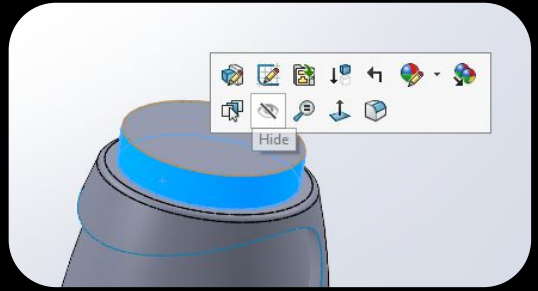
Then Left click on the next selection box and select the excess



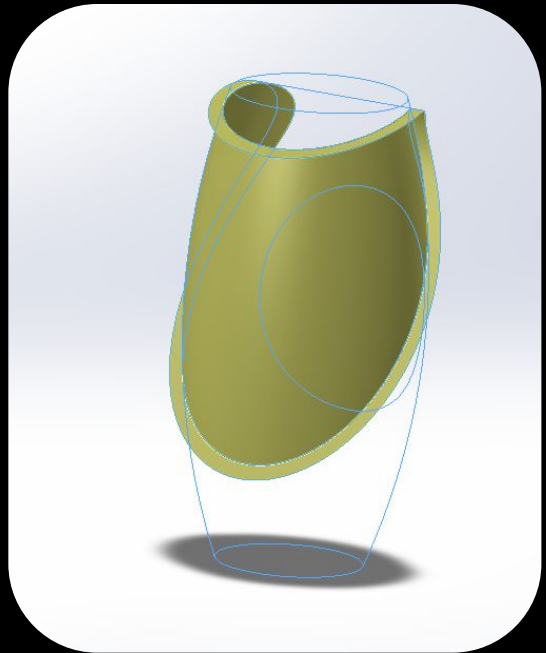
# Repairing the hole - Pt 3

## More Trim Surface

Hide the outer surface to reveal this:



The Result:

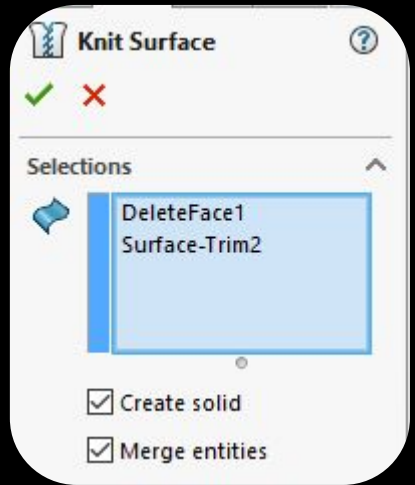
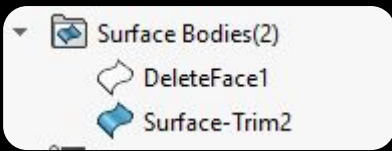


Using Trim Surface as before, select all visible surfaces and trim the parts of the bottle we don't want

# Repairing the hole - Pt 4

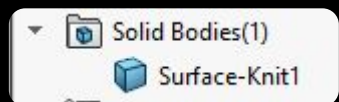
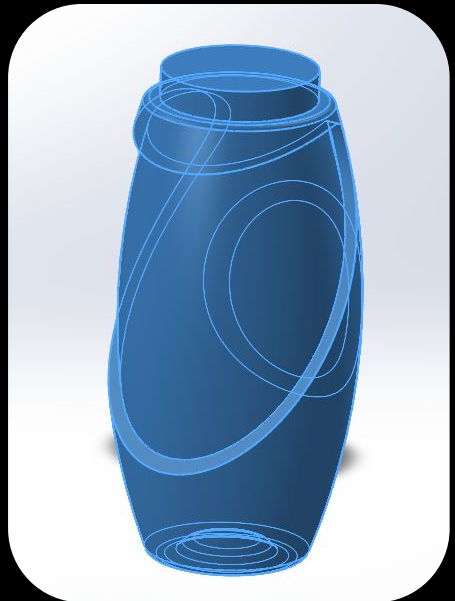
## Knit Surface

To reveal our hidden surface, go to the feature manager and unhide it from Surface Bodies



The Knit Surface tool combines our surfaces into

a solid that we can then modify with the normal SolidWorks features



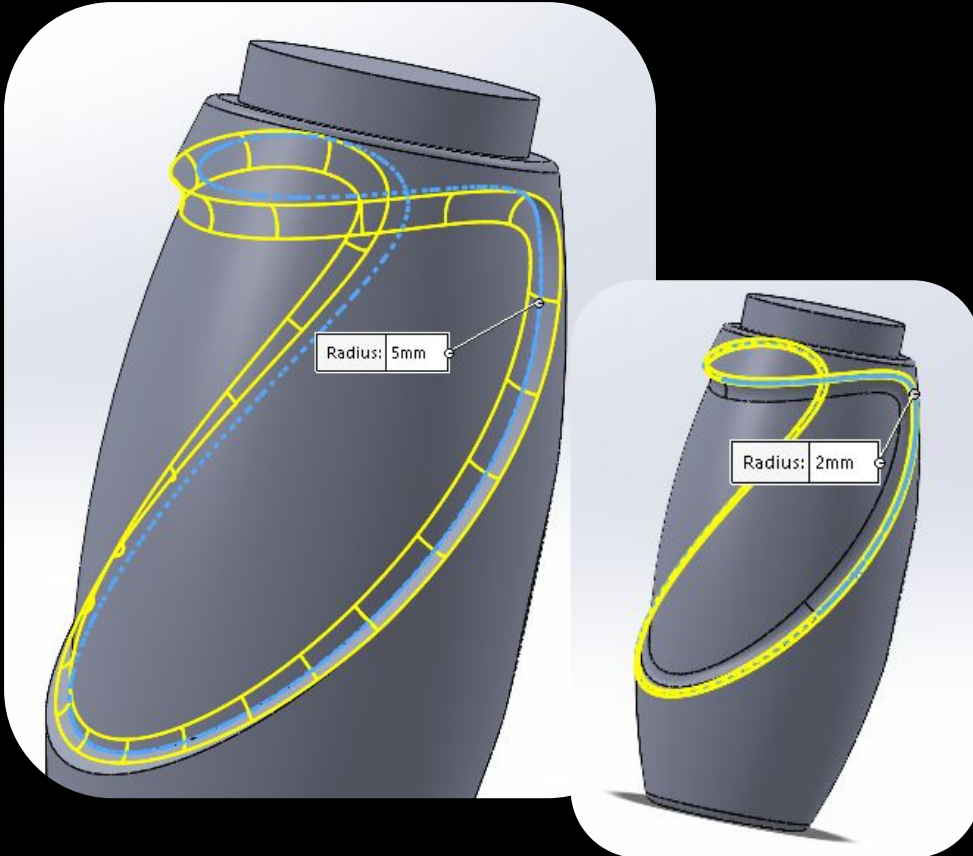
Select all and check the two boxes to create a merged solid bottle

# Friendly Fillets

To make this easy to hold:

Add a 5mm fillet to the inner edge

Add a 2mm fillet to the outer edge



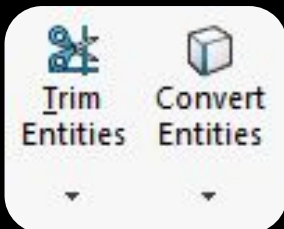
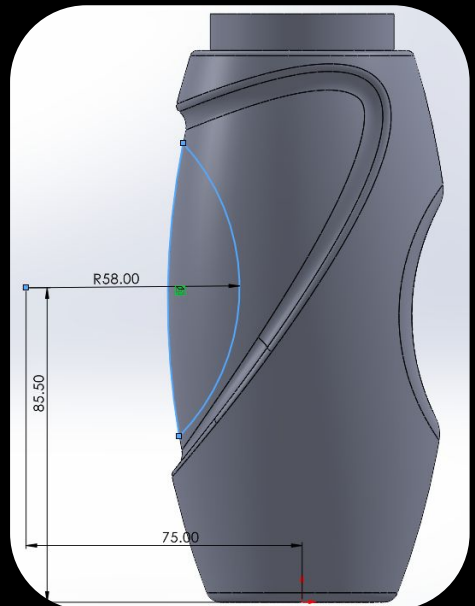
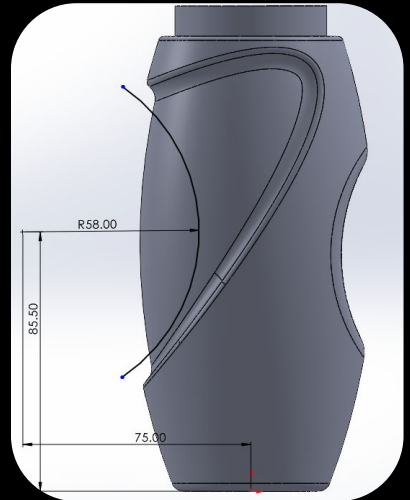


# Second Recessed Contour

On the Front Plane make a three point arc with dimensions:

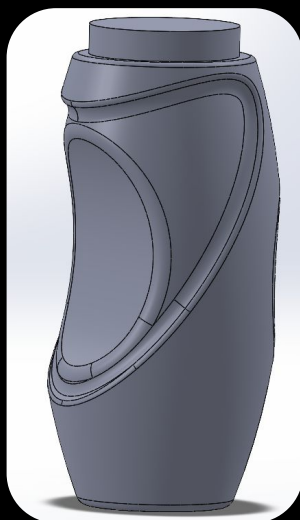
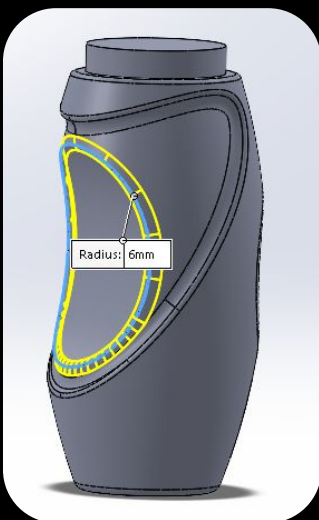
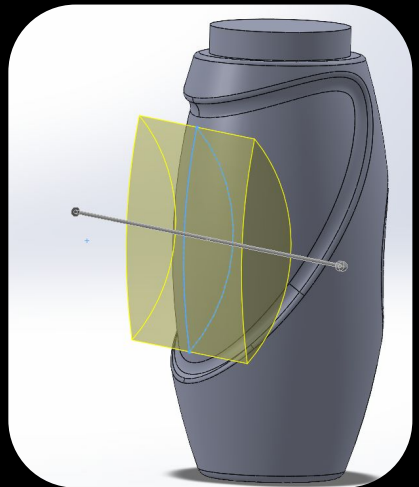
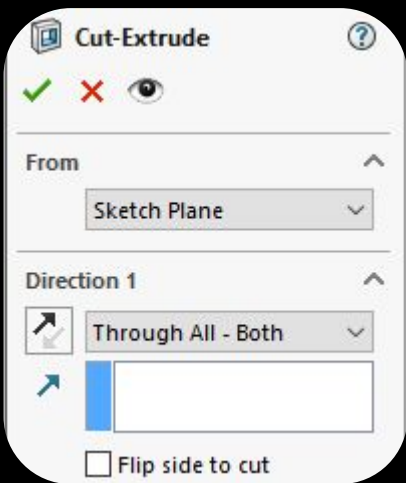
- Radius: 58mm
- Center point 85.5 up from origin
- Center point 75 left from origin

Convert the leftmost edge entity and trim out the unnecessary pieces to make a moon crescent shape



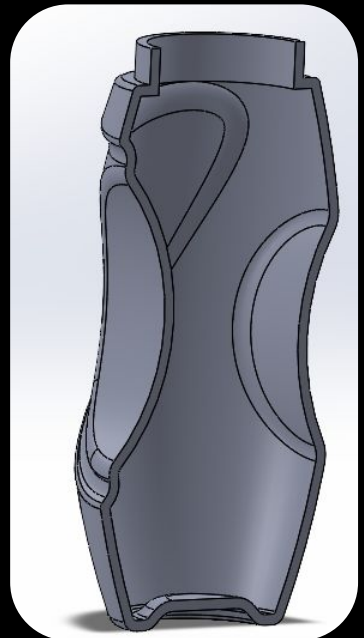
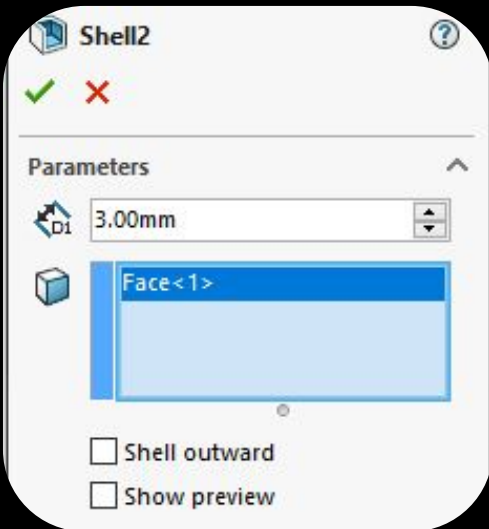
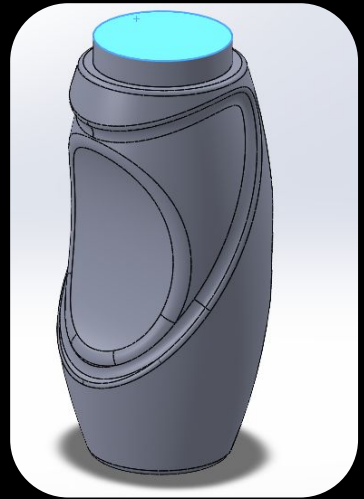
# Second Recessed Contour Continued

Just the same as before with the first recessed contour, we will use a Through All - Both Extruded Cut in order to create a clean contour all of the way around, and then fillet it to remove the sharp edges (6mm)



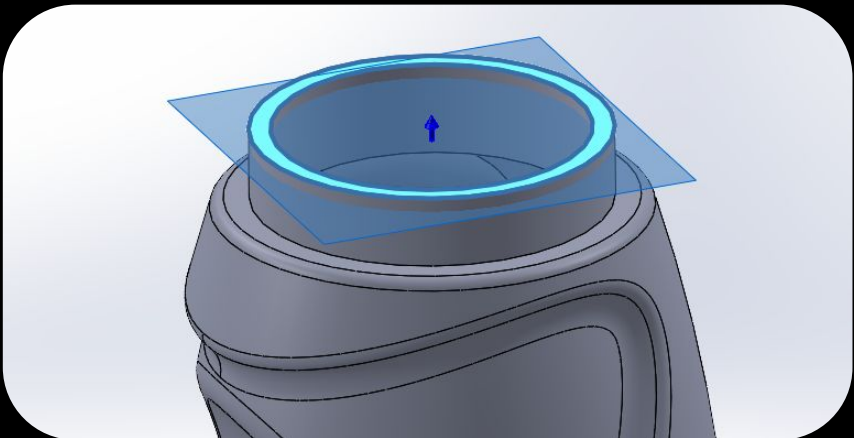
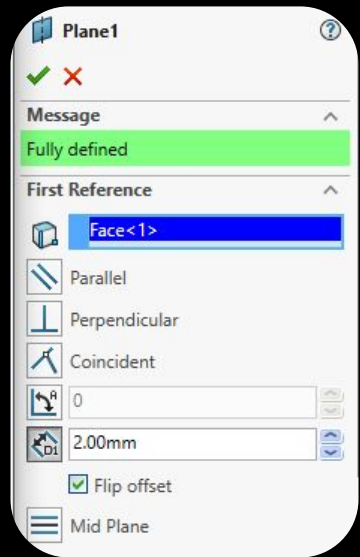
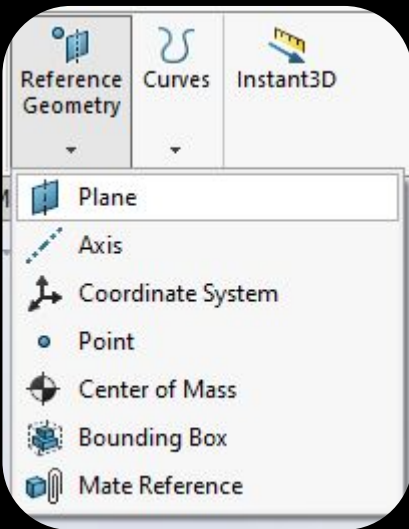
# Bottle Time (Shell Tool)

Time to make a bottle out of our shape,  
Using the Shell tool on the top face apply a thickness of 3 mm



# Helix Spiral (Bottle Cap Screw)

First we need some reference geometry, create a **2mm distanced** reference plane on the top lip of the bottle:



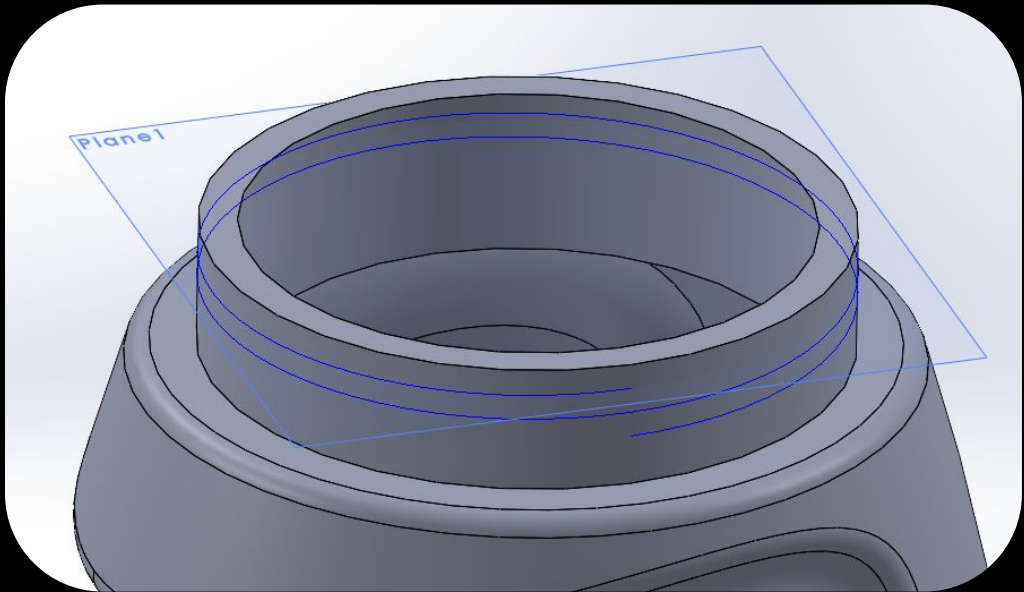
# Helix Spiral (Bottle Cap Screw)



For the sketch on the Plane, Convert the top edge's sketch and exit sketch



Next apply the Helix and Spiral Tool with a 3mm pitch, 2 revolutions, and a starting angle of 0 degrees

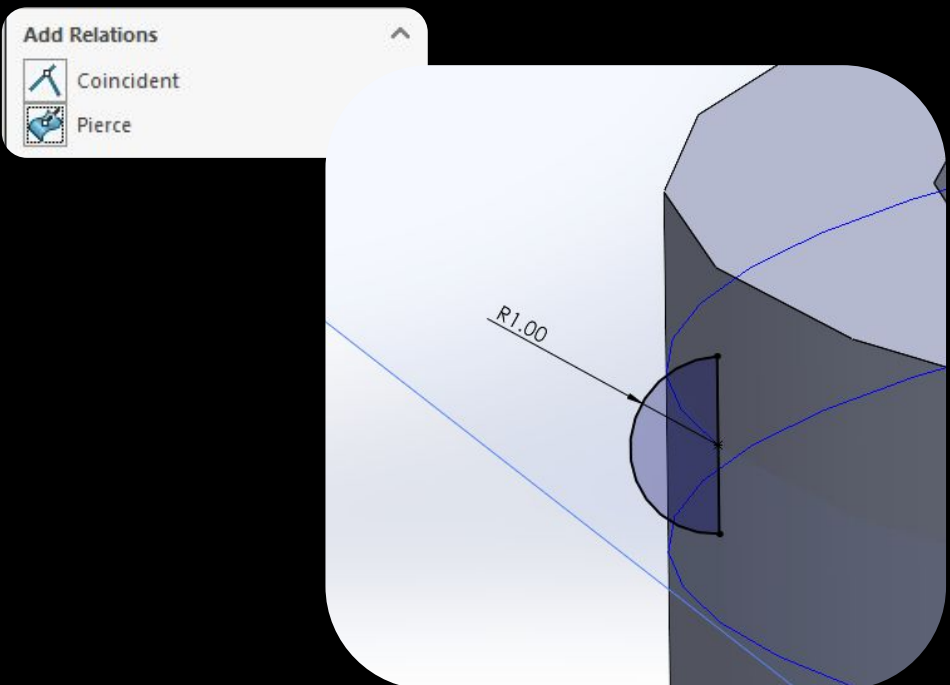


# Helix Spiral (Bottle Cap Screw)

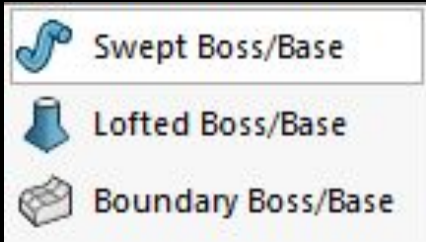
On the Right Plane make a small circle with a radius of 1 mm (Or diameter of 2)

Connect the center of the circle to its top edge with a Line, and trim the right side of the circle to make a semicircle

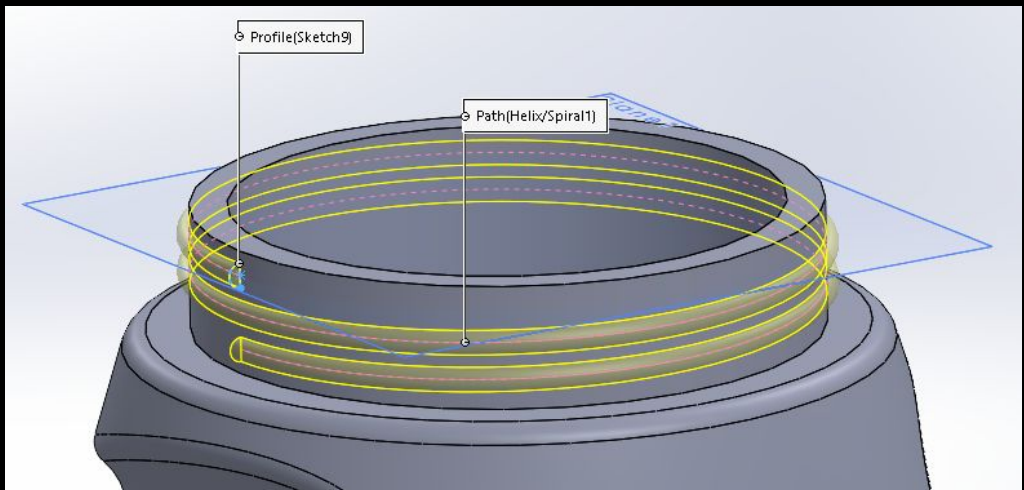
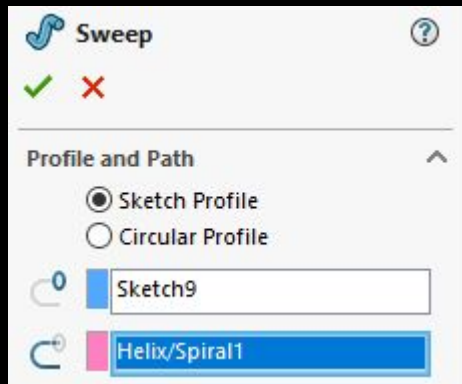
Selecting the mid point of the circle / line hold control and select the helix of the previous step and add a "Pierce" Relation



# Helix Spiral (Bottle Cap Screw)



Using a Swept Boss Base we will be able to rotate our semi circle around the lip of the bottle



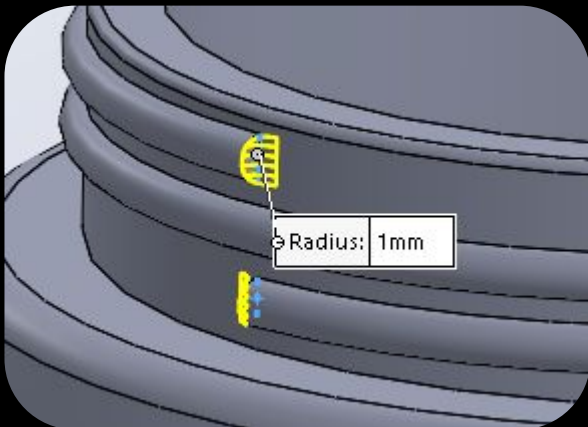
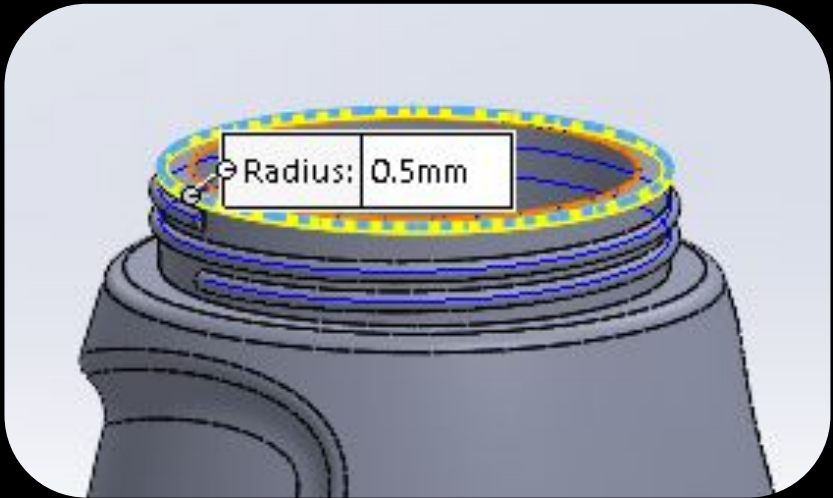
# Clean up!

Last finishing touches:

Hide the Reference plane and Helix Spiral

Fillet remaining features:

- the ends of the spiral (1mm)
- the top lip (0.5mm)





# Bottle Finished!

- Bottle (Default<<Default>>)
- History
- Sensors
- Annotations
- Solid Bodies(1)
  - Fillet11
- Material <not specified>
- Front Plane
- Top Plane
- Right Plane
- Origin
- Revolve1
- Cut-Extrude1
- Fillet3
- Fillet4
- Fillet5
- Cut-Extrude2
- Fillet6
- Surface-Offset2
- Split Line2
  - Sketch6
- DeleteFace1
- Surface-Extrude1
- Surface-Trim1
- Surface-Trim2
- Surface-Knit1
- Fillet7
- Fillet8
- Cut-Extrude3
- Fillet9
- Shell2
- Plane1
- Helix/Spiral1
- Sweep1
- Fillet10
- Fillet11

