



HOW TO MODEL A
DEODORANT ROLLER
IN SOLIDWORKS?

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HOW TO MODEL A **DEODORANT ROLLER** IN SOLIDWORKS?

“A step by step SolidWorks Tutorial”

About myself

My name is Jan-Willem Zuyderduyn and I am the founder and owner of LearnSolidWorks.com.

I've been working over twelve years with 3d CAD software and over eight years with SolidWorks. In that time I've learnt a lot about all the possibilities with SolidWorks.

I am graduated in 2008 with a Bachelor Degree in Product Design & Engineering. I've worked for [Sinot Yacht Design](#) as yacht designer. I am currently working as an Industrial Designer for the [TSG Group](#) in Eindhoven, the design city of the Netherlands. I am also working as freelance designer and SolidWorks teacher of "Advanced Surface Modeling 3". I am specialized in concept design, 3d modeling and visualizations. If you need a freelance designer for a project, don't hesitate to send an e-mail to info@learnsolidworks.com

In 2007 and 2008 I ended in the top 3 of the international SolidWorks Car Design Contest of the Benelux (2007) and Europe (2008). It took me six years to learn everything about SolidWorks what I know now. In that time I have been asked many times how to model and render 3D models using SolidWorks. The last few years I've written multiple e-books and tutorials about SolidWorks. My mission is to assist as many people as I can with learning SolidWorks. That's why I've created this website, LearnSolidWorks.com. By the way, I am not related or affiliated with SolidWorks in any way.

I hope you will learn something from this tutorial. Feel free to leave a [Comment](#) on the LearnSolidWorks website or to [Share](#) it with your friends. This free SolidWorks tutorial is a complimentary resource. You may distribute (I encourage you to share) this tutorial as a free gift, or post it on your website as long as the content is not changed and it is delivered via this PDF file. It's not allowed to sell this E-book because I offer it for free on my website. I assume no responsibility or liability whatsoever on this materials.

Check the www.LearnSolidWorks.com website for more free SolidWorks tutorials like this one.

Enjoy this SolidWorks tutorial!

Jan-Willem Zuyderduyn

P.S. Add me on Twitter, and stay up to date with all my newest SolidWorks tutorials:

<http://twitter.com/LearnSW>



How to Model a Deodorant Roller in SolidWorks?

In this tutorial you will learn how to model a Deodorant Roller in SolidWorks. In this lesson I'll show you the following features:

- Draw a 2d sketch
- Insert a blueprint
- Surface Revolve
- Surface Sweep
- Surface Loft
- Surface Fill
- Surface Knit
- Fillet
- New Axis
- Revolved Cut

Render of the model you will create (made in PhotoView360)




Open a new part with model units set to millimeters

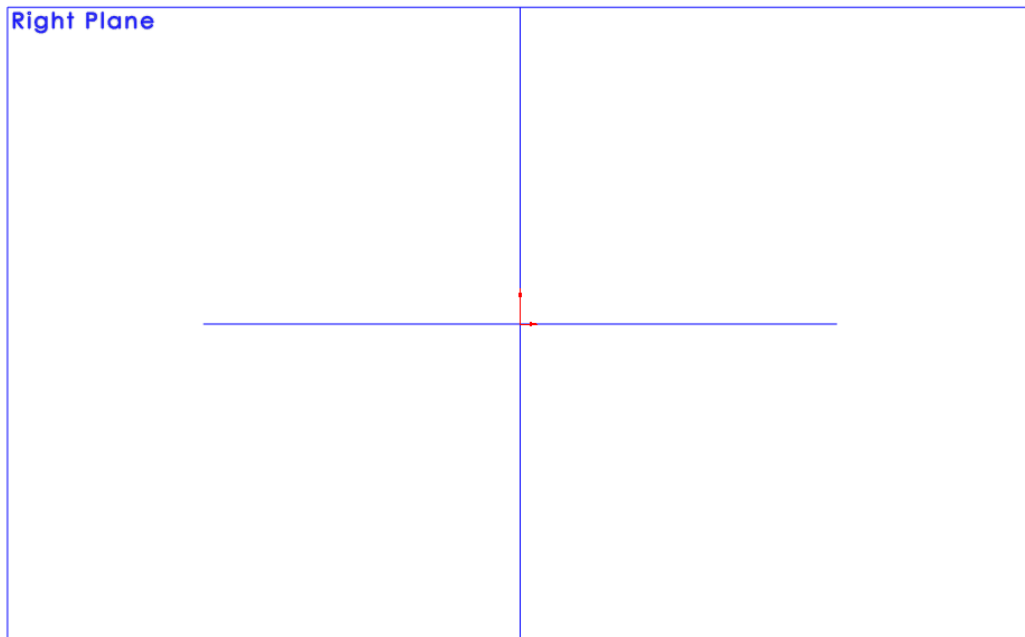
Go to: **File > New > Part**



Create a 2D sketch

Select the Right Plane in the feature tree (menu at the left side) and create a sketch by clicking on the 2D Sketch icon 

The display changes so the Right plane faces you.



Insert a reference picture

For this tutorial we use a blueprint of the Deoroller to approach the organic shape as good as possible.

Download the picture [here](#) and save it into your SolidWorks folder

Go to: **Tools > Sketch Tools > Sketch Picture** 

Go to your SolidWorks folder and select the picture “**SIDVIEW_DEOROLLER.Jpg**”

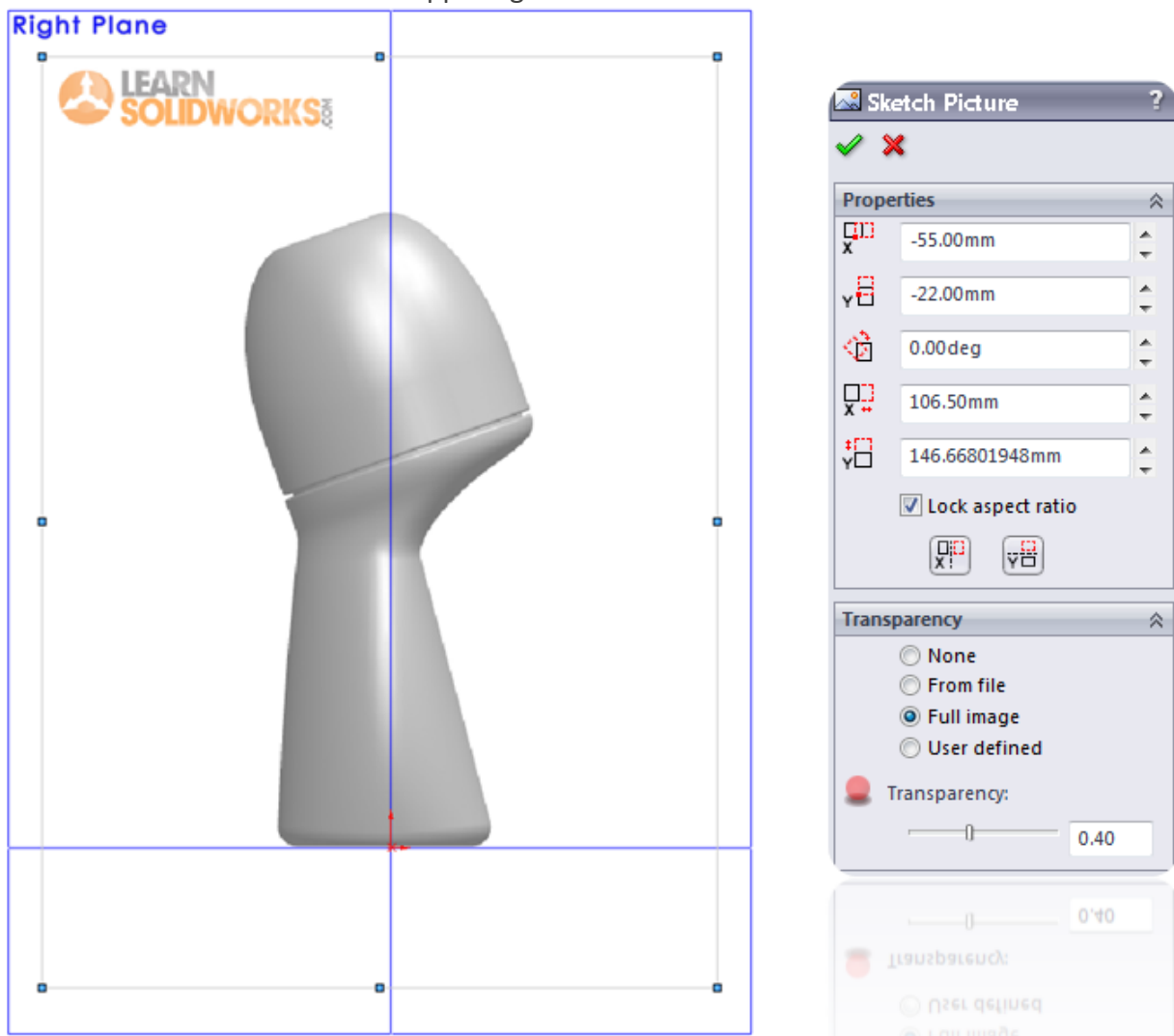
Click: Open

Change the dimensions and position of the picture with the menu as shown in the picture.


Select “Full image” in the Transparency tab and change the transparency into 0.40

Click OK 

Click at the Sketch button in the upper right corner close the 2D Sketch 



Insert a second reference picture

Select the Front Plane in the feature tree (menu at the left side) and create a sketch by clicking on the 2D Sketch icon 

Download the picture [here](#) and save it into your SolidWorks folder

Go to: **Tools > Sketch Tools > Sketch Picture** 

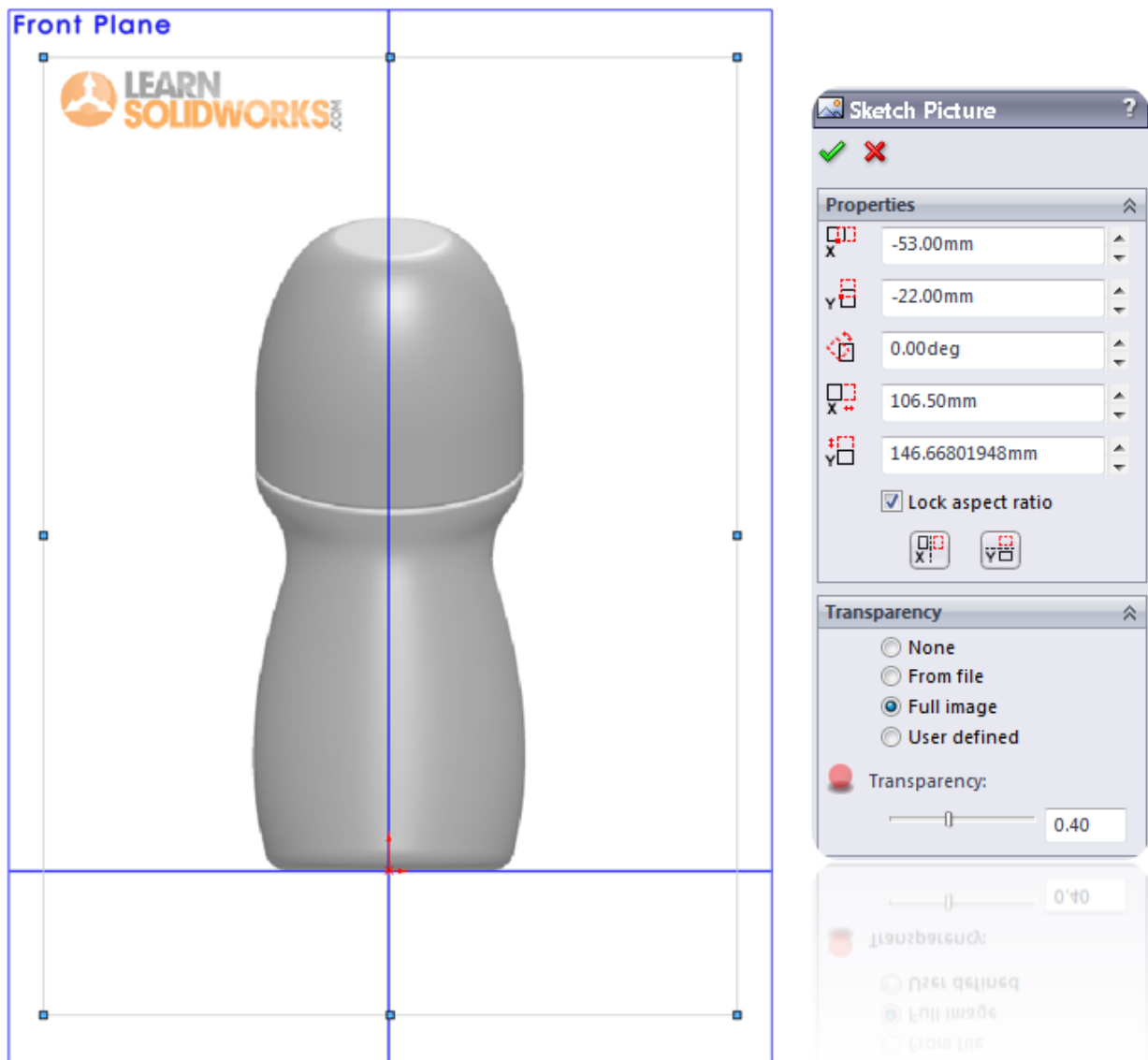
Go to your SolidWorks folder and select the picture “FRONTVIEW_DEROLLER.Jpg”


Click: Open

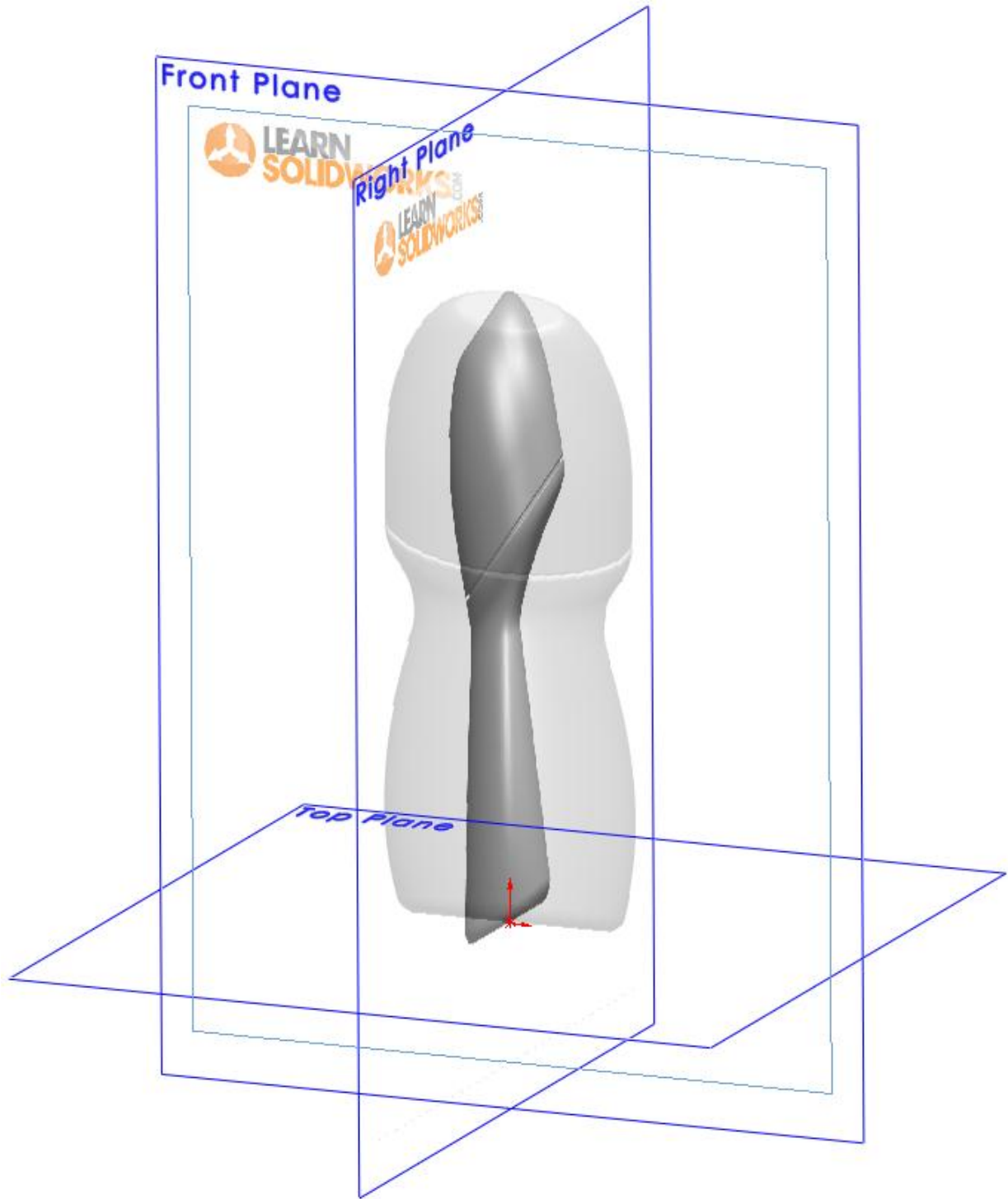
Change the dimensions and position of the picture with the menu as shown in the picture.

Select “Full image” in the Transparency tab and change the transparency into 0.40


Click OK 




Click at the Sketch button in the upper right corner close the 2D Sketch 




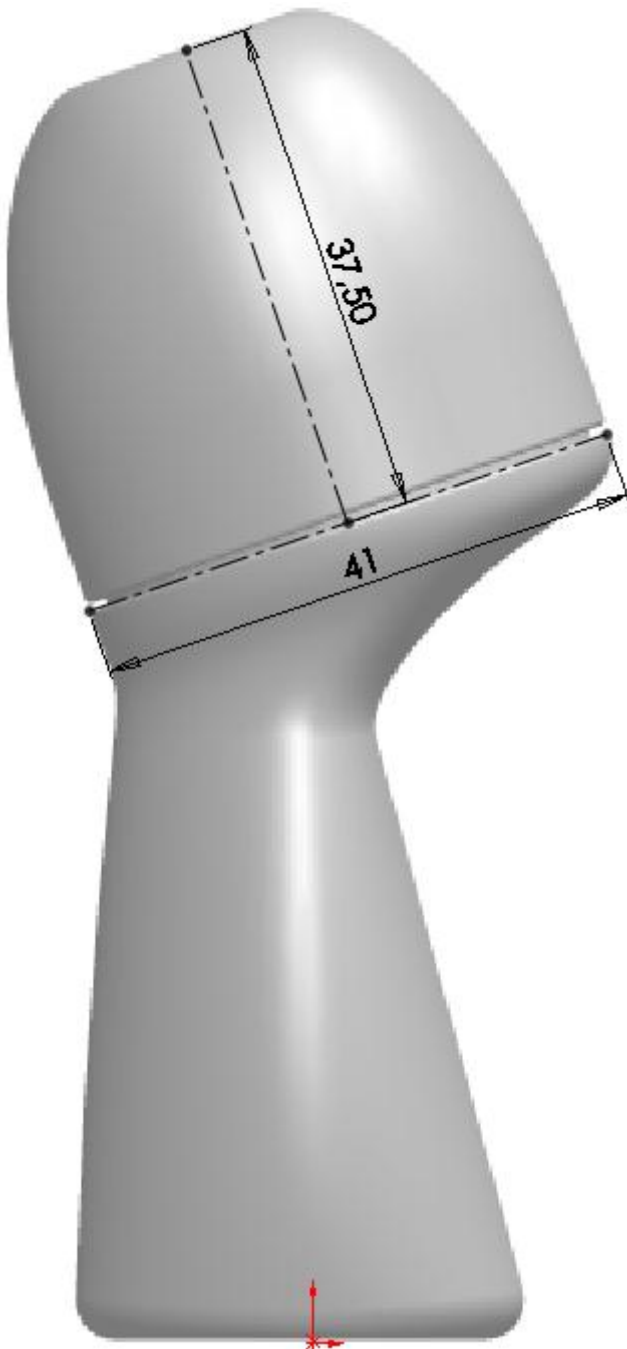
Create a 2D sketch


Select the Right Plane in the feature tree and create a sketch by clicking on the 2D Sketch icon 

Draw the two centerlines as shown in the picture 

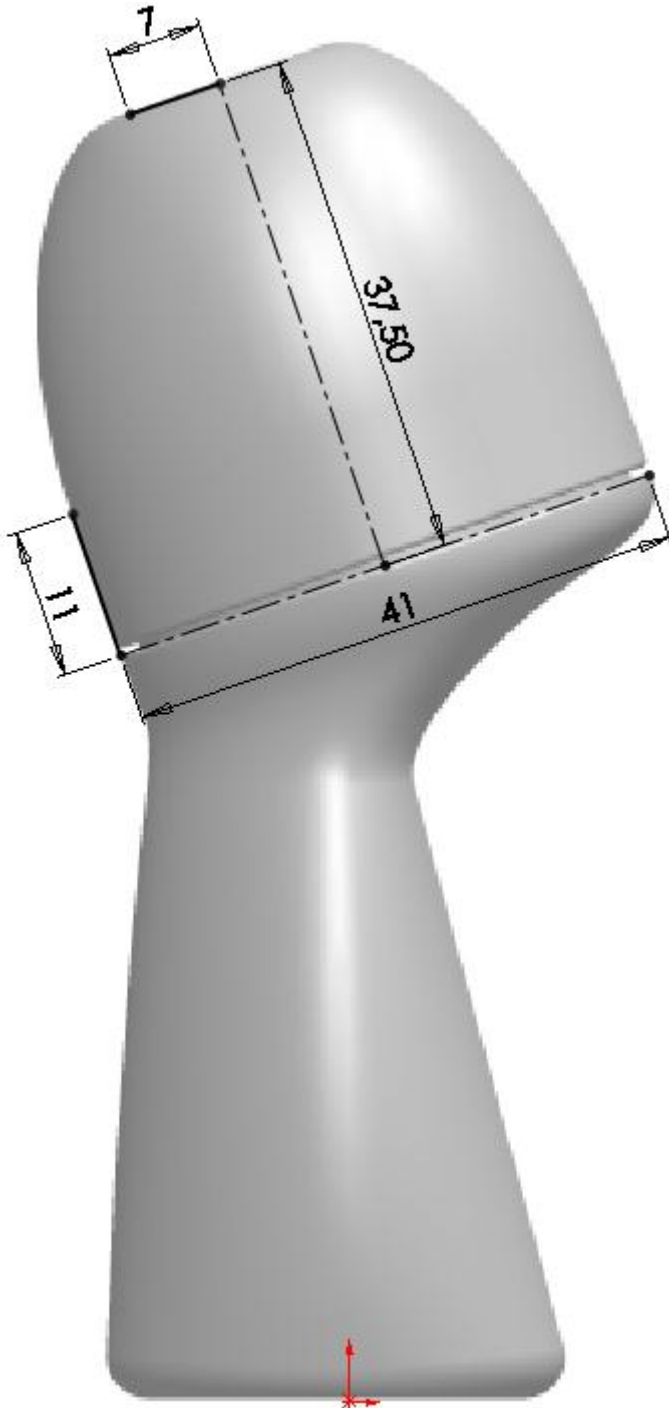
Change the dimensions by clicking at the dimension button 


Fix the two lines with the Fix icon 

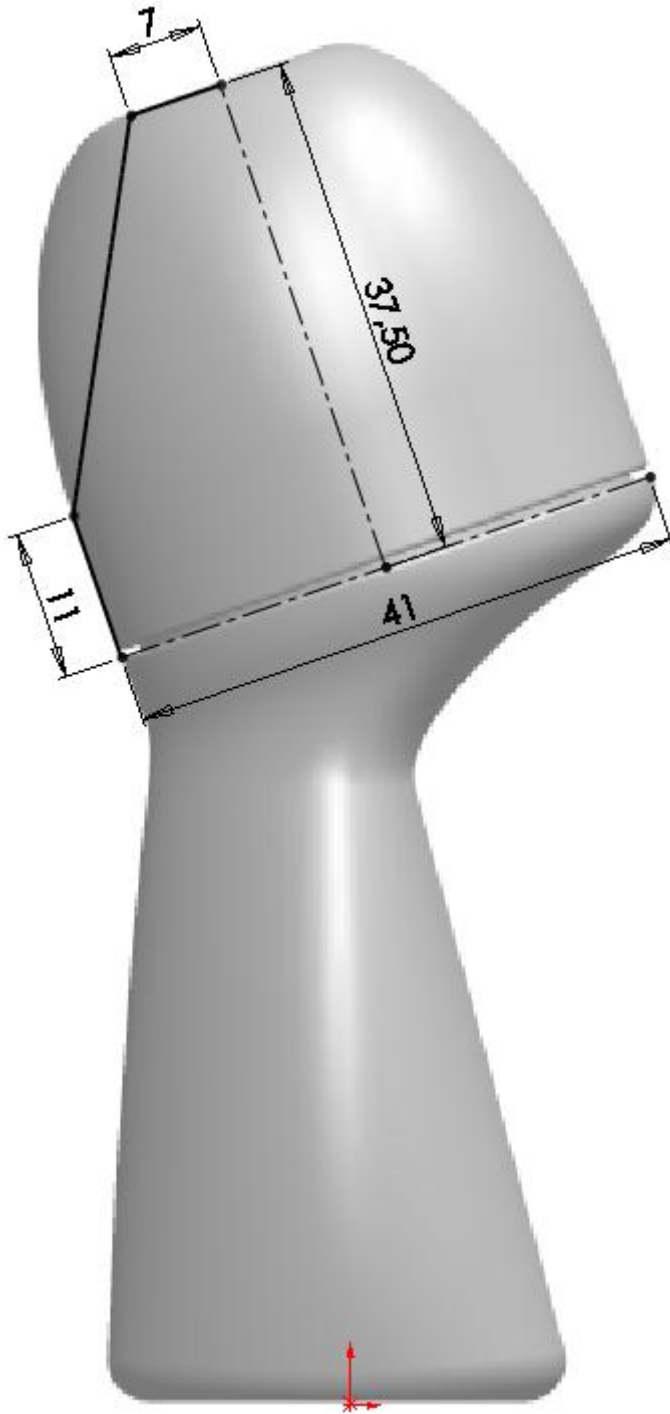


Draw the two lines as shown in the picture 


Change the dimensions by clicking at the dimension button 



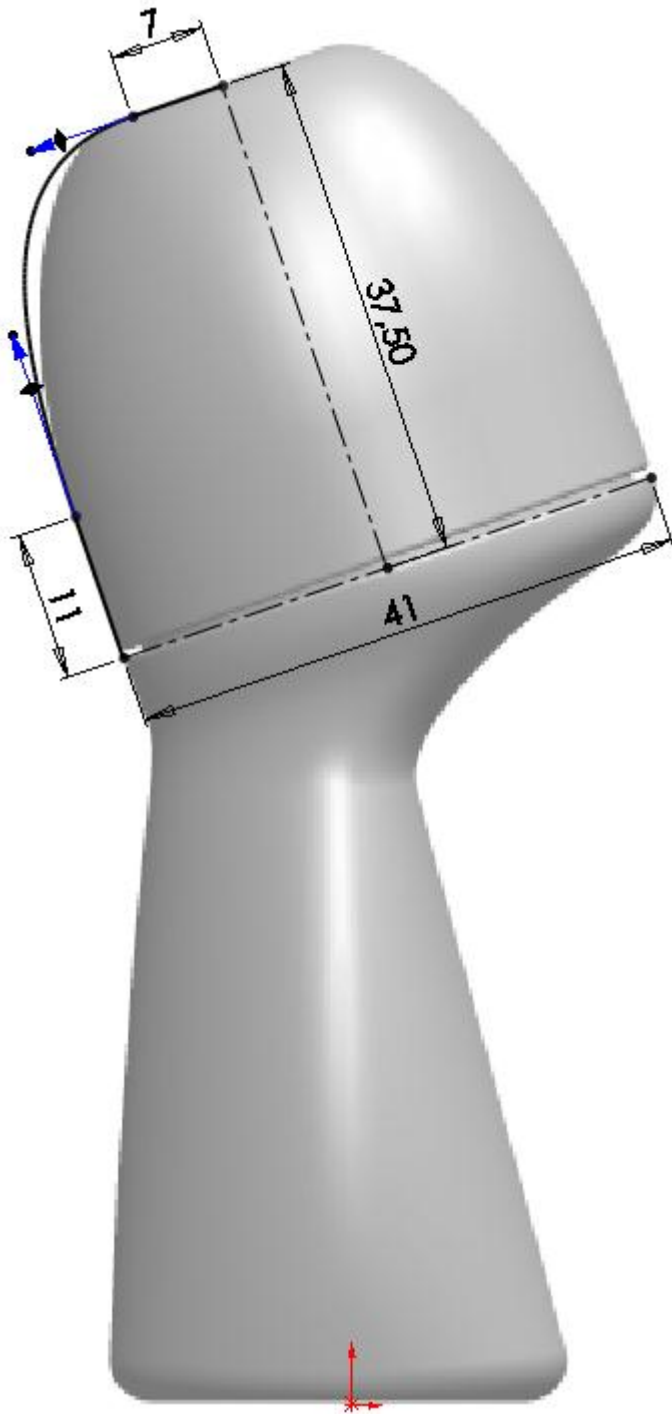
Draw a spline without any midpoints 



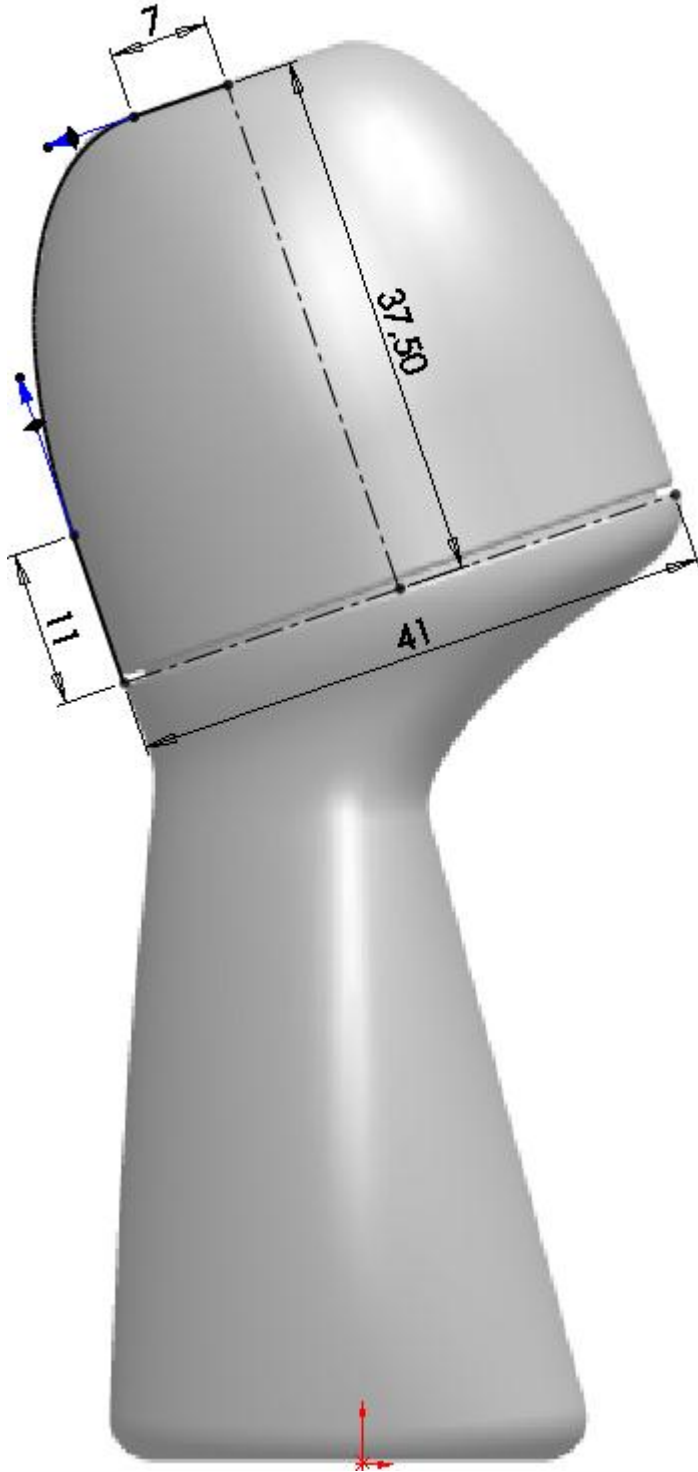
Select one of the straight lines, hold down the control key and select the spline


Click at the Tangent icon as shown in the picture 

Repeat this action for the other side of the spline




Change the length of the arrows to approach the curve of the blueprint




Click at the Sketch button in the upper right corner close the 2D Sketch 

Create a Surface Revolve

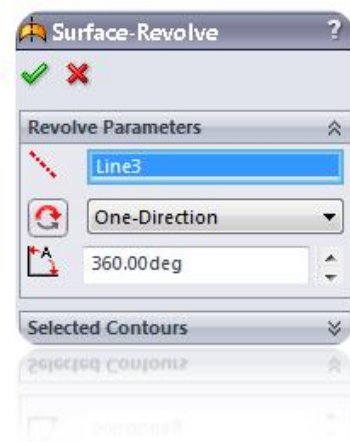
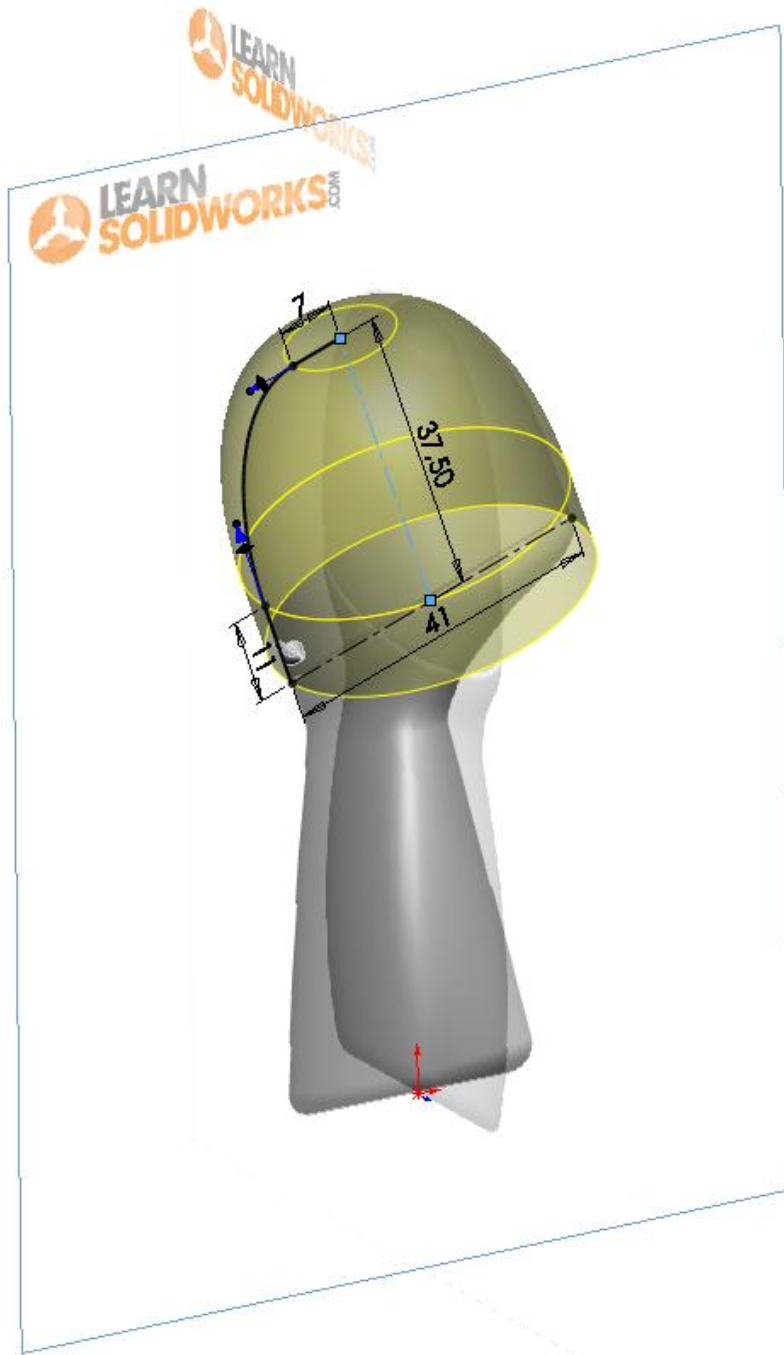
Go to **Insert > Surface > Revolve** or click at the Revolve icon 

Click at the blue Centerline to define the Axis of Revolution 


Use the One-Direction option 


Set the Revolution Angle to 360 degrees 

Click OK 




Create a 2D sketch

Select the Right Plane in the feature tree and create a sketch by clicking on the 2D Sketch icon 

Draw the line as shown in the picture 

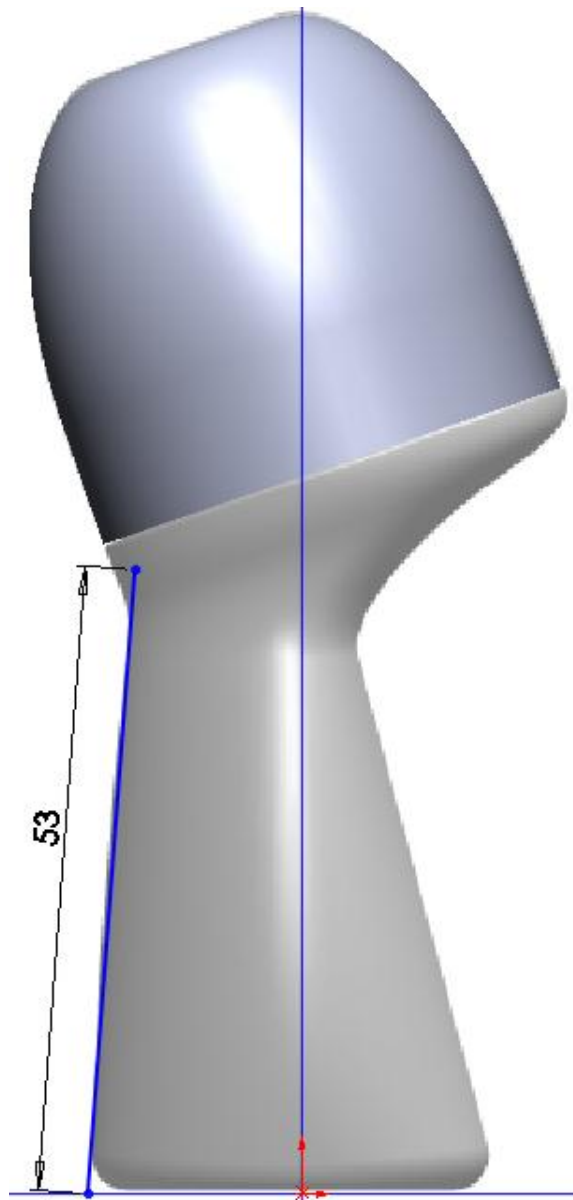
Change the dimensions by clicking at the dimension button 

Make sure that the angle of the line is equal to the front curve of the deoroller


Click at the Sketch button in the upper right corner close the 2D Sketch 


Rename the Sketch4

Double click at Sketch4 in the feature tree and rename it to **GUIDELINE_FRONT**




Create another 2D sketch

Select the Right Plane in the feature tree and create a sketch by clicking on the 2D Sketch icon 

Draw the line as shown in the picture 

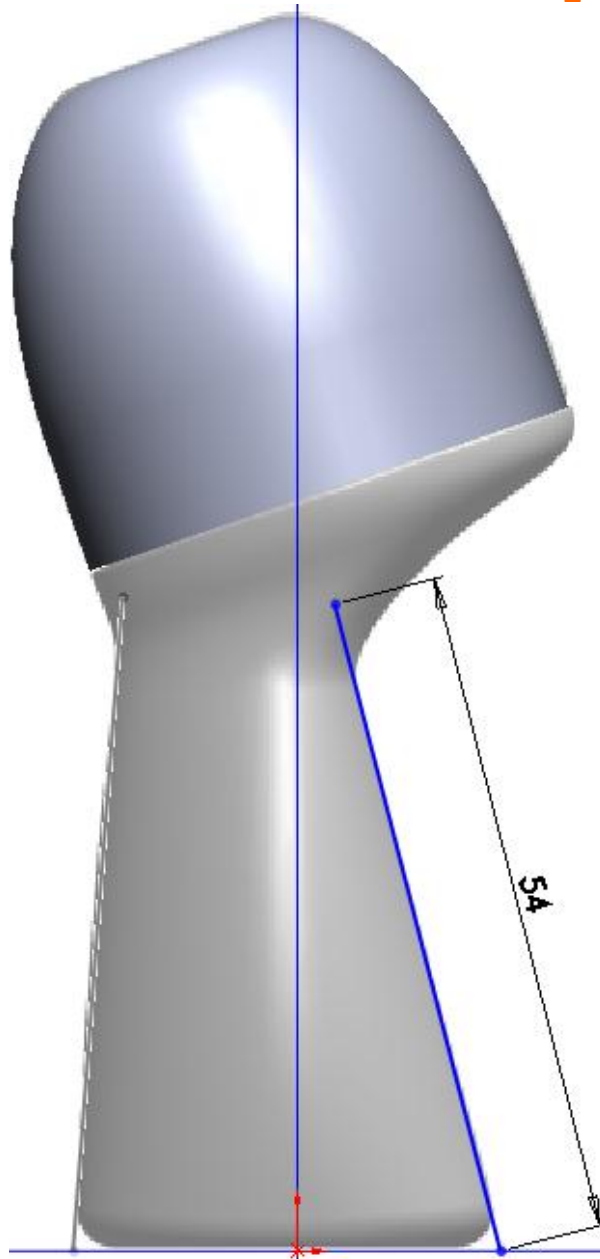
Change the dimensions by clicking at the dimension button 

Make sure that the angle of the line is equal to the back curve of the deoroller


Click at the Sketch button in the upper right corner close the 2D Sketch 


Rename the Sketch5

Double click at Sketch5 in the feature tree and rename it to **GUIDELINE_BACK**




Create a 2D sketch

Select the Front Plane in the feature tree and create a sketch by clicking on the 2D Sketch icon 

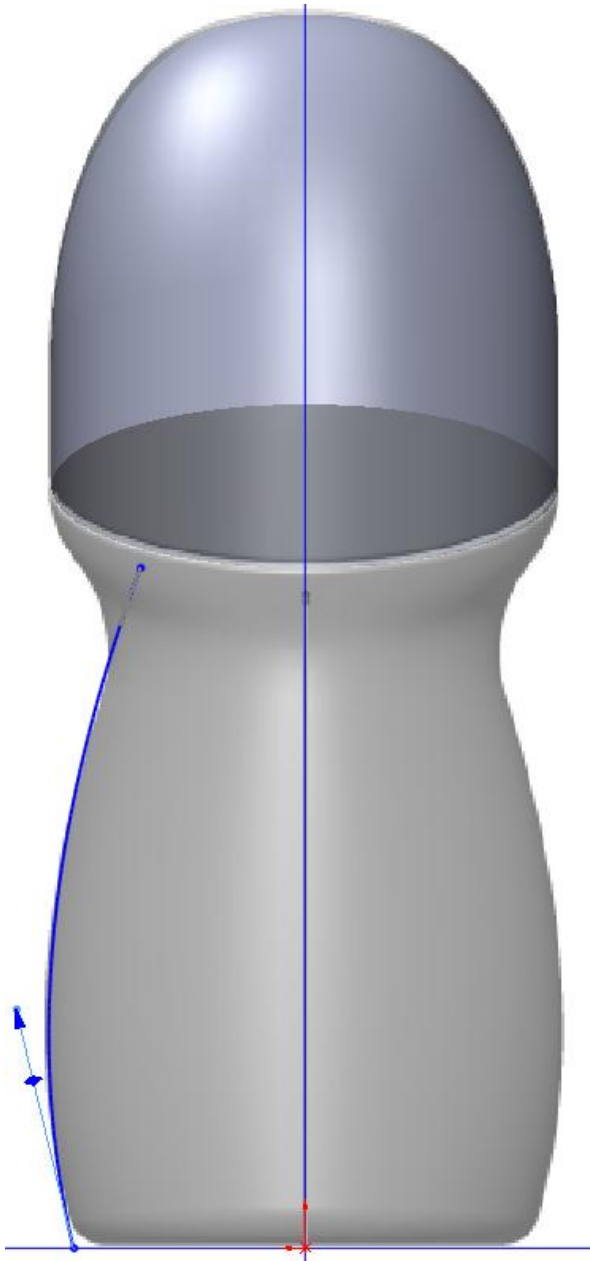
Draw a spline without midpoints as shown in the picture 

Change the length of the lower arrow to approach the curve of the blueprint


Click at the Sketch button in the upper right corner close the 2D Sketch 


Rename the Sketch6

Double click at Sketch6 in the feature tree and rename it to **GUIDELINE_SIDE**

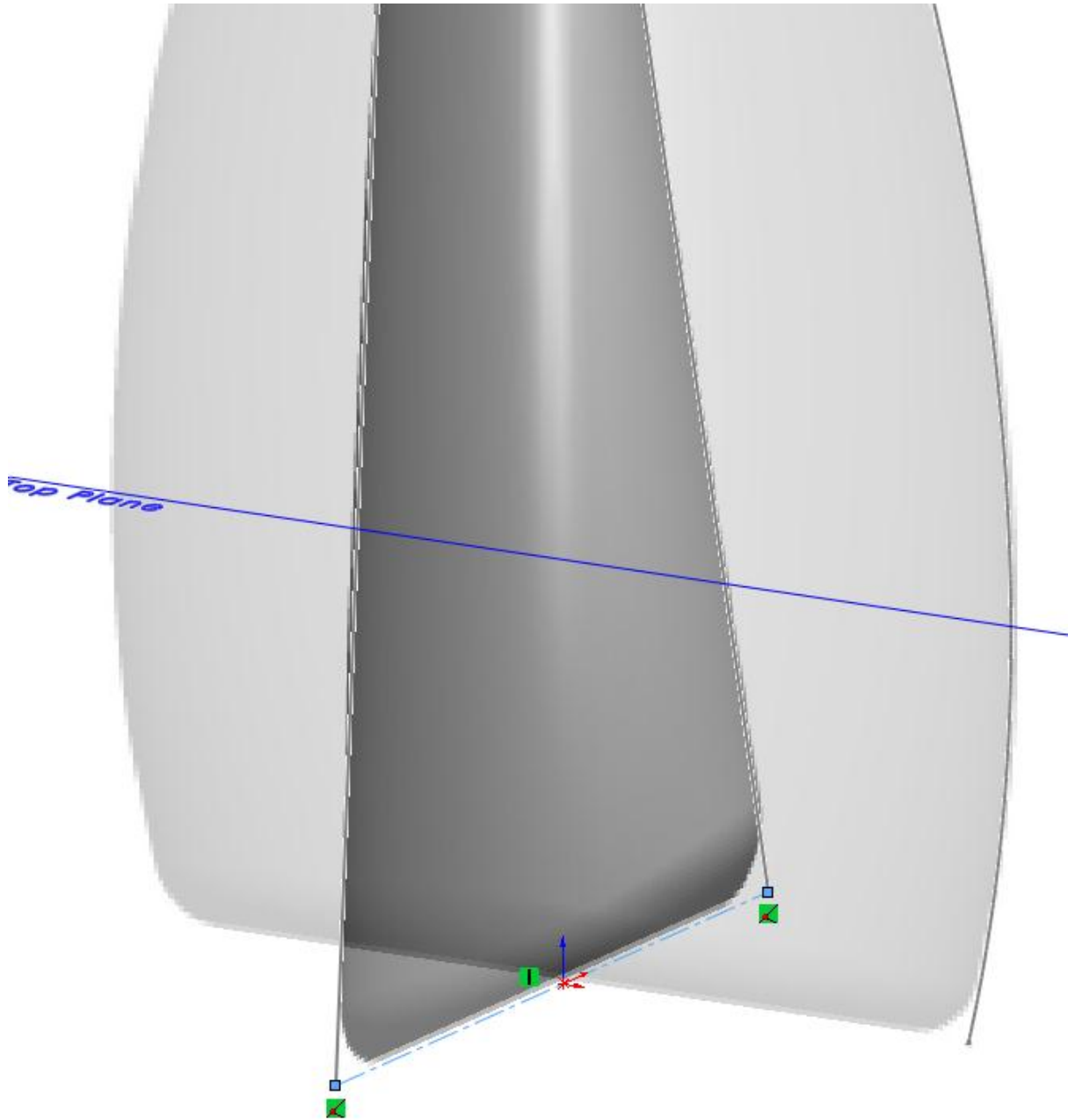



Create a 2D sketch

Select the Top Plane in the feature tree and create a sketch by clicking on the 2D Sketch icon 

Draw the centerline as shown in the picture 

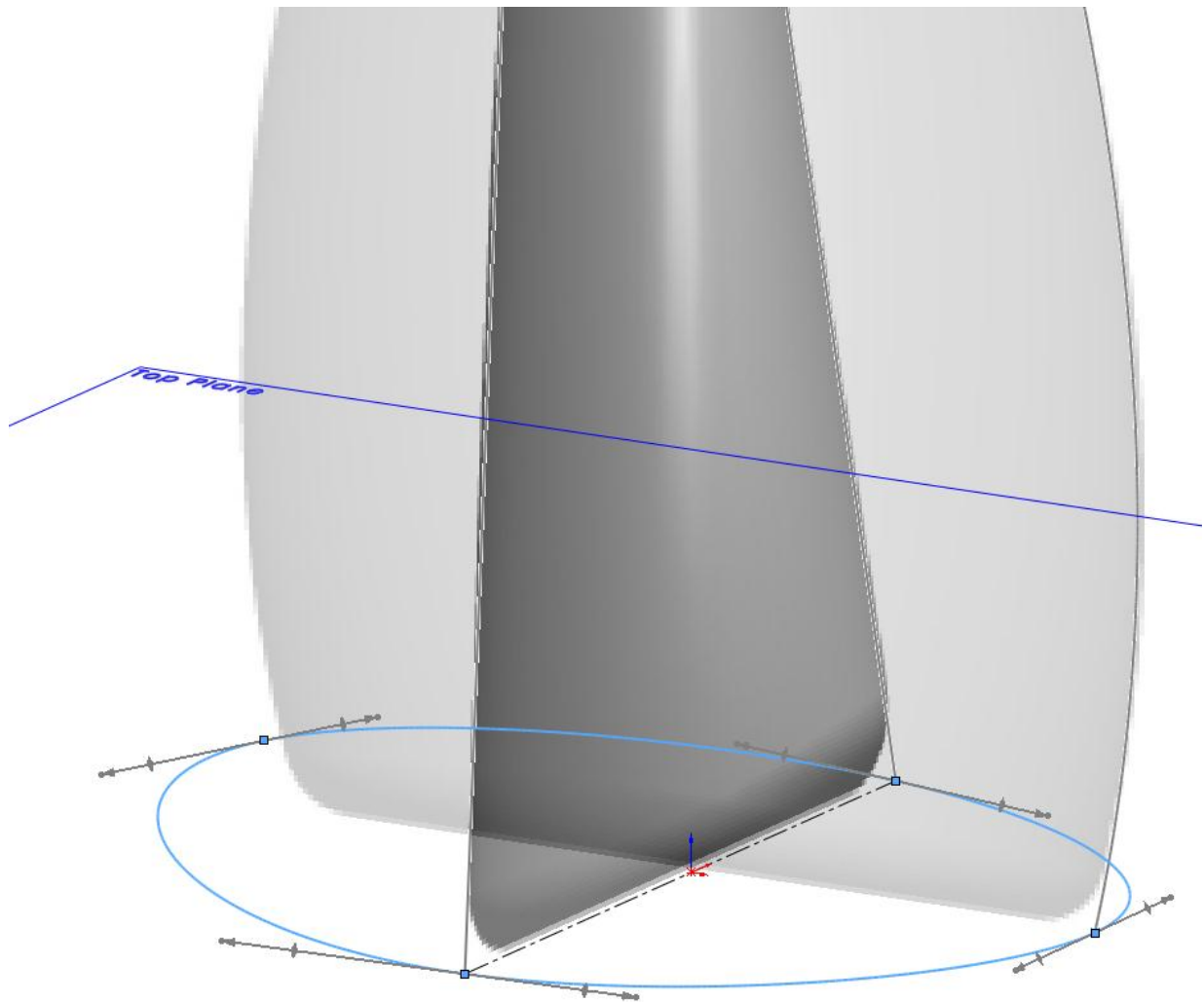
Connect the centerline with the endpoints of **GUIDELINE_FRONT** and **GUIDELINE_BACK**



Draw a spline with four points as shown in the picture 


Connect three points with **GUIDELINE_FRONT**, **GUIDELINE_BACK** and **GUIDELINE_SIDE**

Draw the fourth point somewhere in the space as shown in the picture

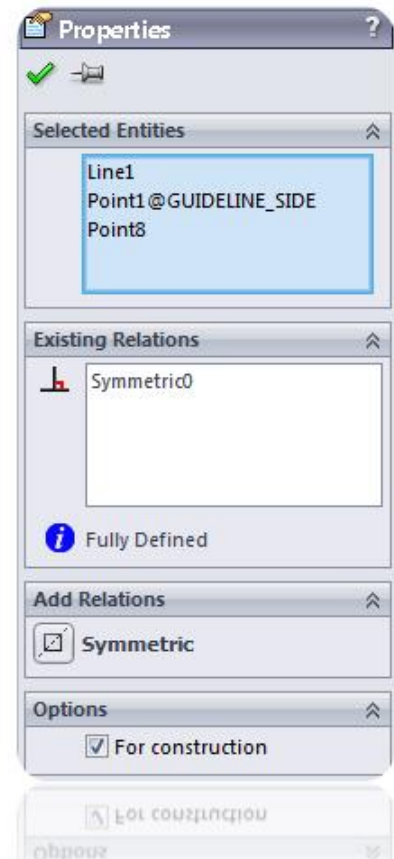
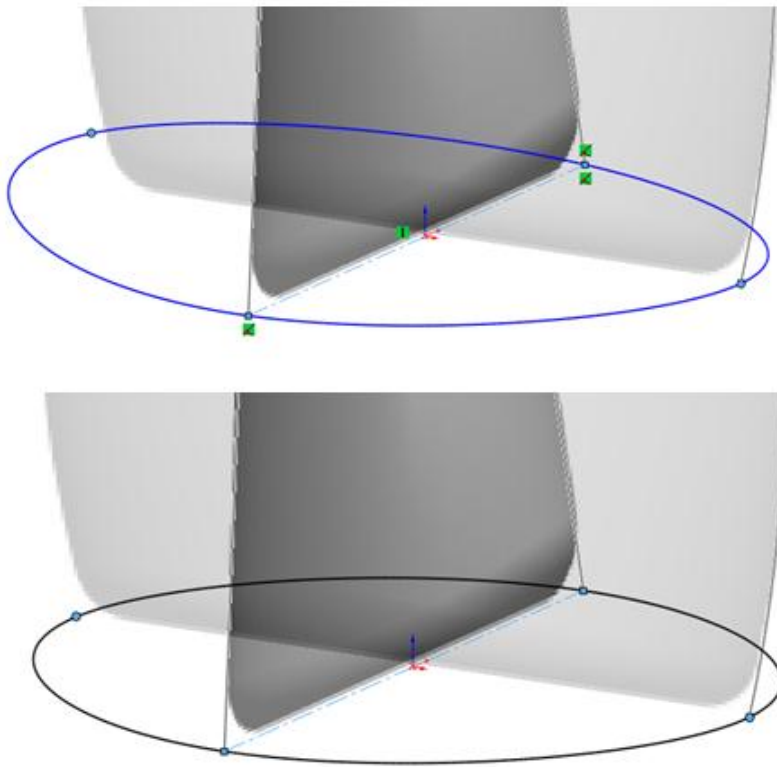



Make the Spline symmetric

Select the right point, hold down the Control key, select the centerline and select the left point.

Click at the Symmetric option in the menu 

Click OK 





Click at the Sketch button in the upper right corner close the 2D Sketch 


Rename the Sketch7


Double click at Sketch7 in the feature tree and rename it to **PROFILE**

Create a 2D sketch

Select the Right Plane in the feature tree and create a sketch by clicking on the 2D Sketch icon 

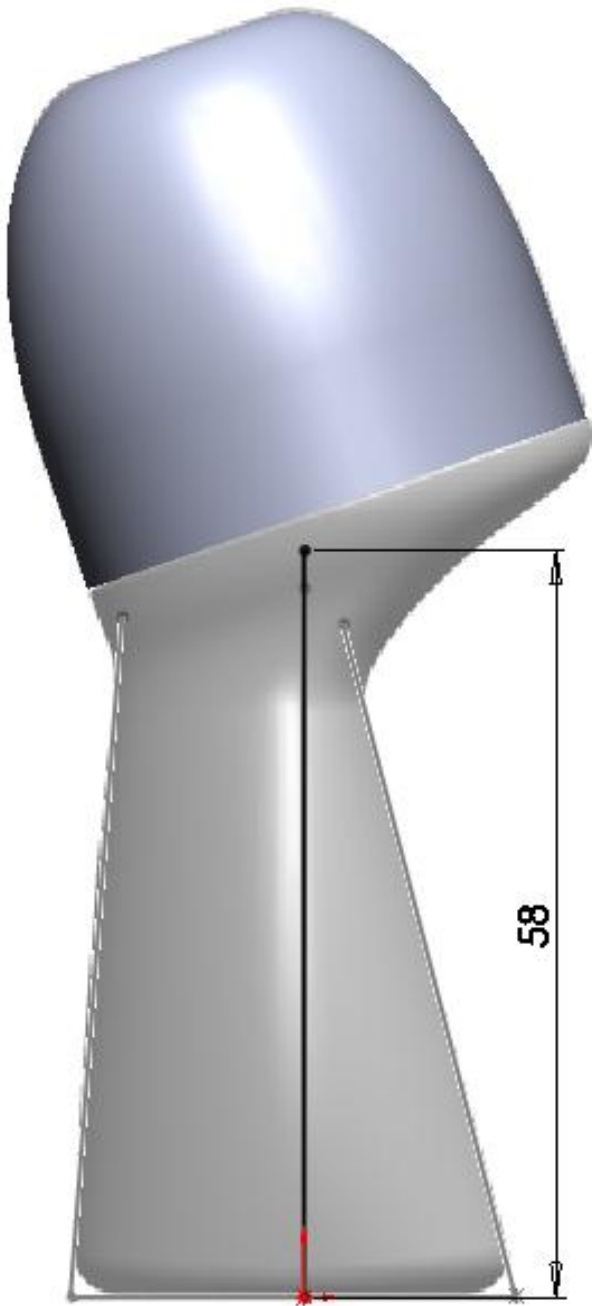
Draw a line, starting at Origin 

Change the length of the line to 58 mm 

Click at the Sketch button in the upper right corner close the 2D Sketch 

Rename the Sketch8

Double click at Sketch8 in the feature tree and rename it to **PATH**

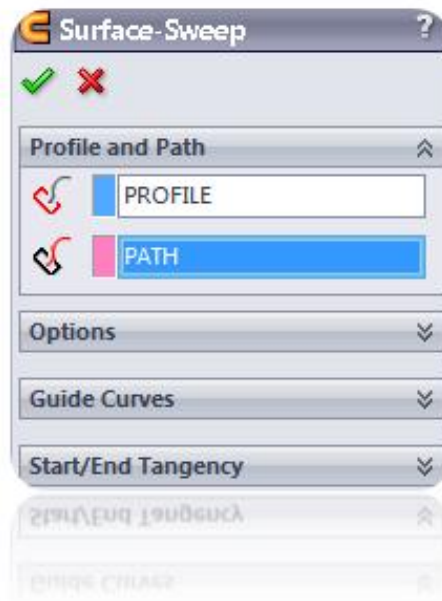
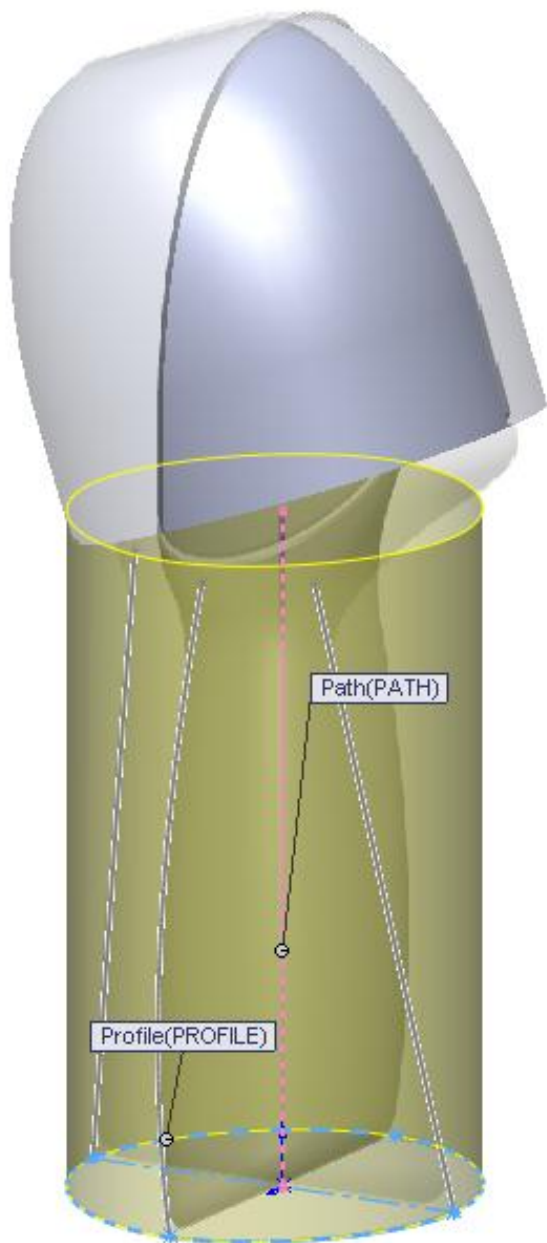


Create a Surface Sweep

Go to **Insert > Surface > Sweep** or click at the Sweep icon 

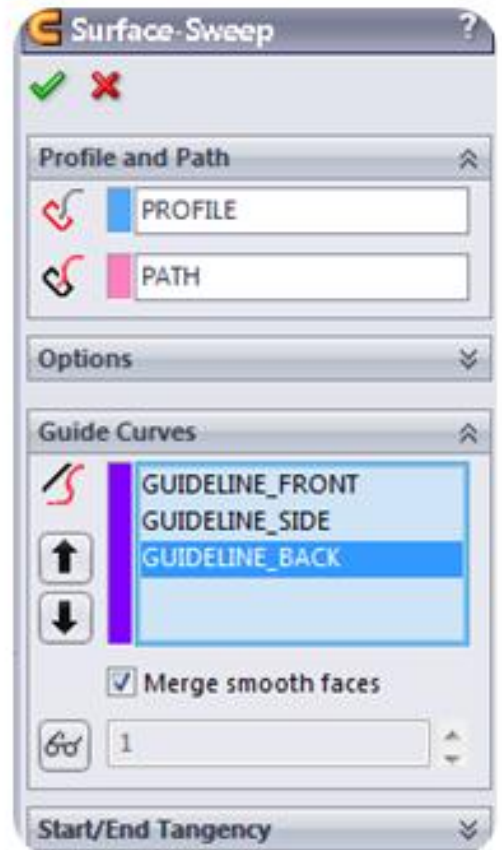
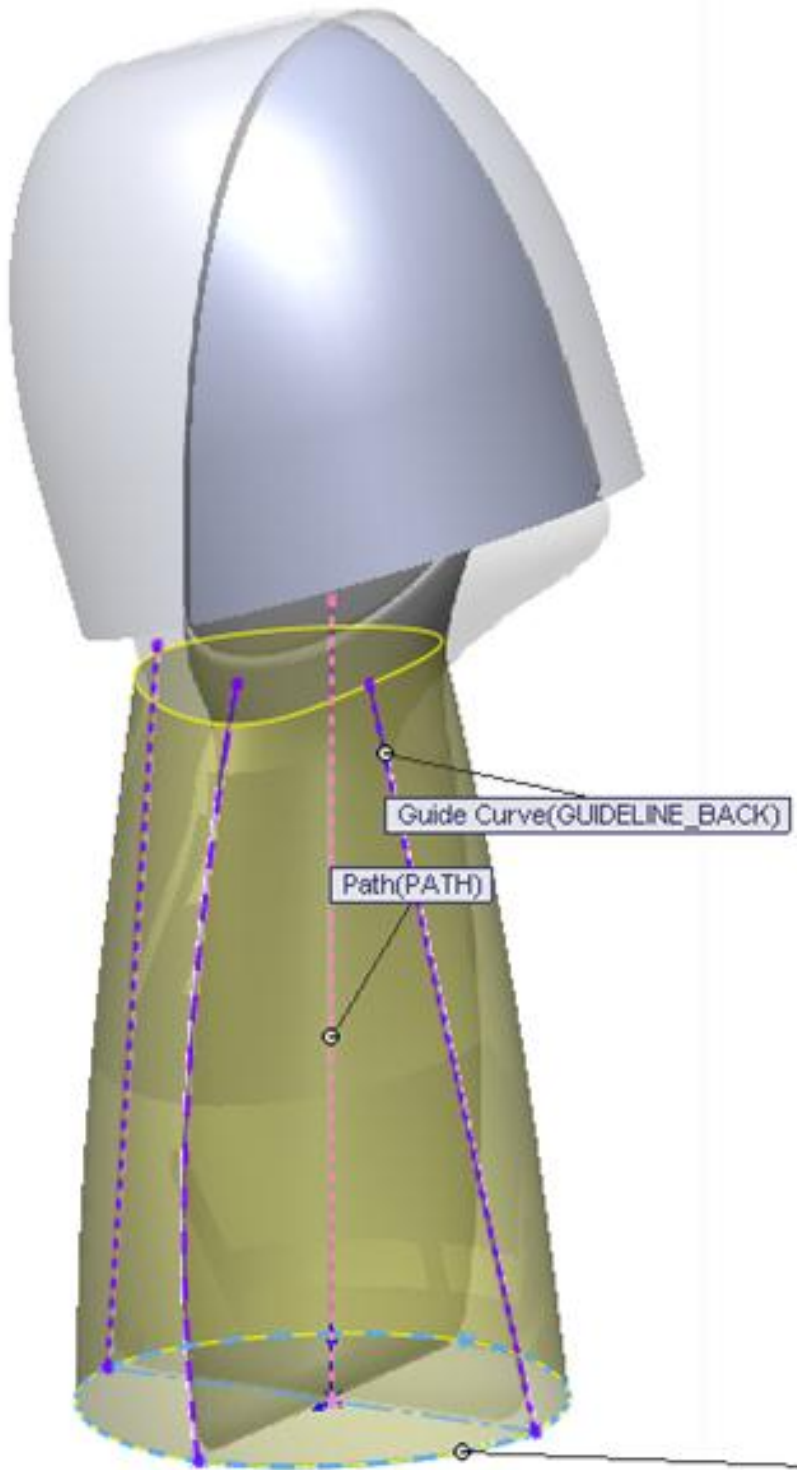
Select the **PROFILE** sketch in the Feature Tree as Sweep Profile

Select the **PATH** sketch in the Feature Tree as Sweep Path





Select **GUIDELINE_FRONT**, **GUIDELINE_BACK** and **GUIDELINE_SIDE** in the Feature Tree as Sweep Guides

Click OK 




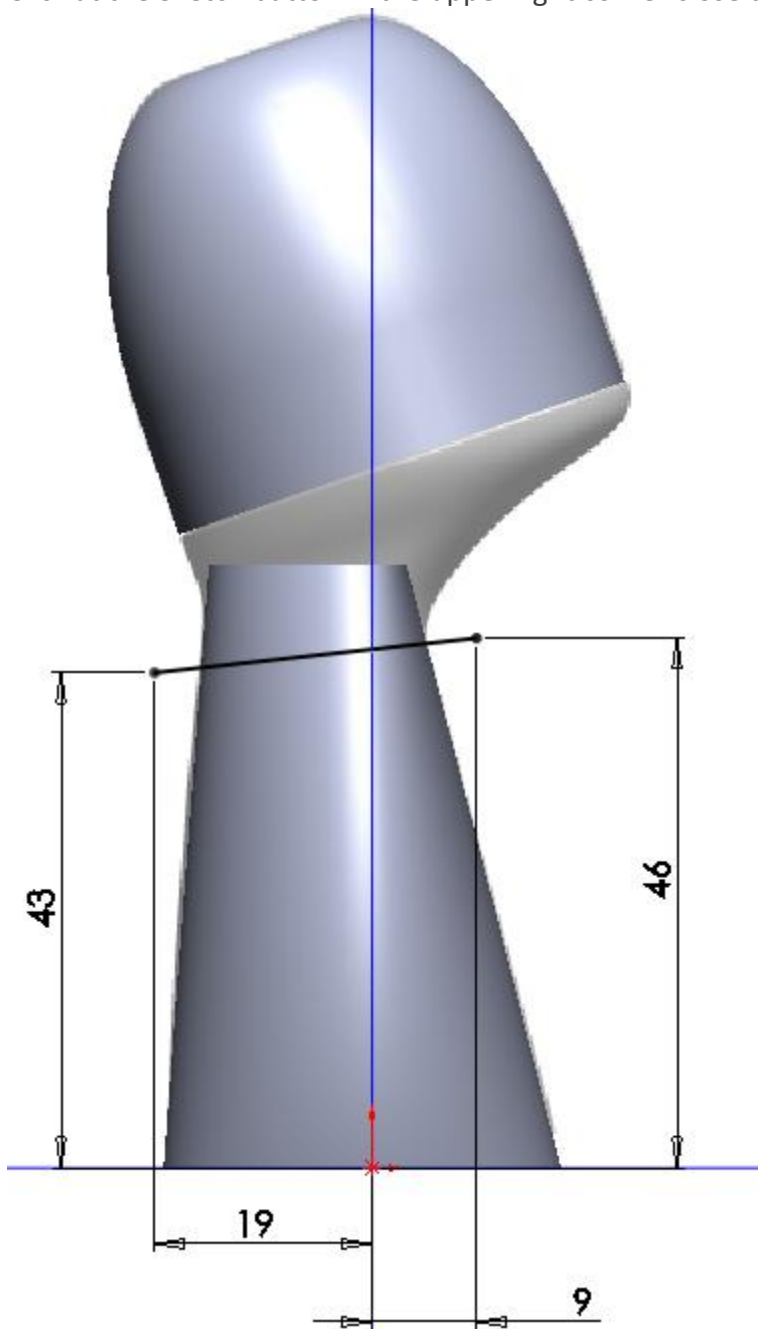
Create a 2D sketch

Select the Right Plane in the feature tree and create a sketch by clicking on the 2D Sketch icon 

Draw the line as shown in the picture 

Change the dimensions by clicking at the dimension button 

Click at the Sketch button in the upper right corner close the 2D Sketch 



Trim the upper side of the Surface Sweep

Go to: **Insert > Surface > Trim** or click at the Trim icon 

Click in the Trim tool box 

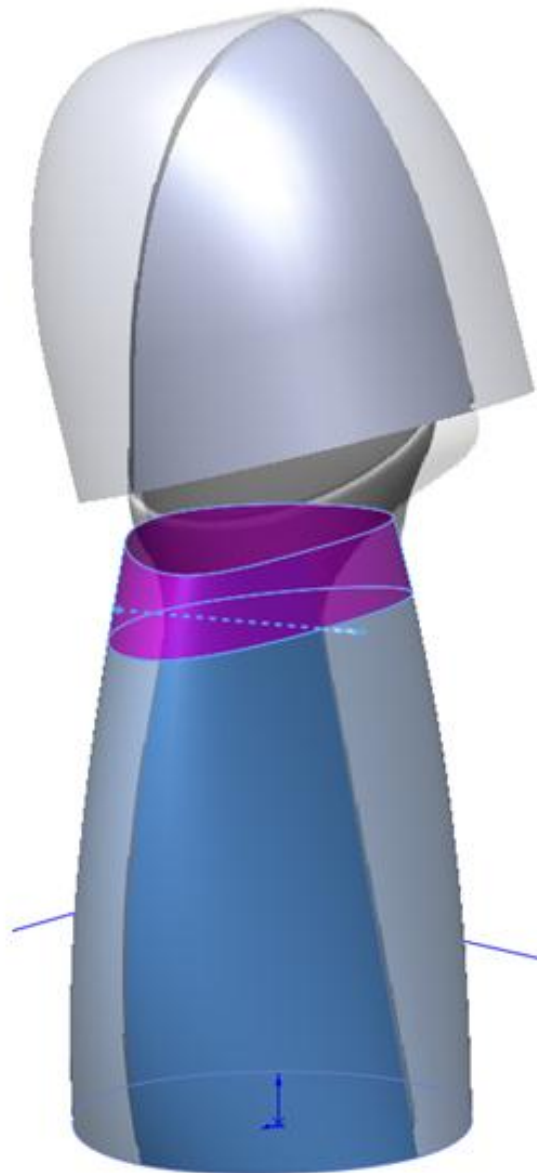
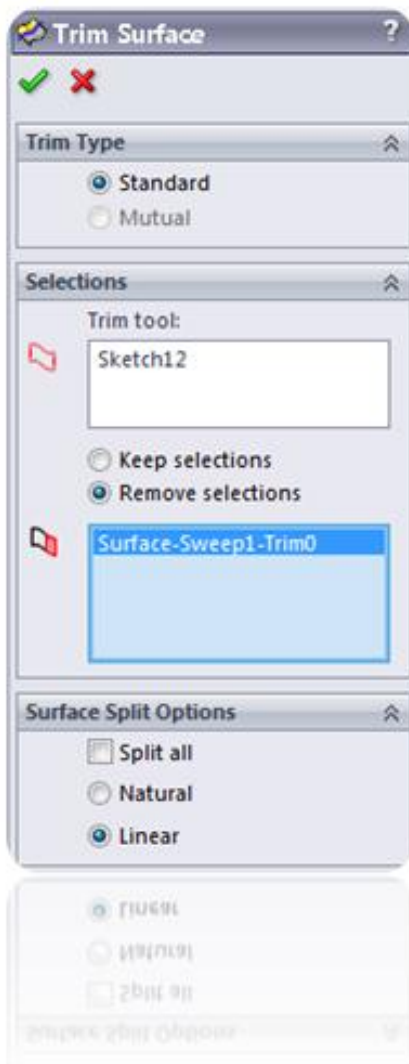
Select the line of the new Sketch9 as shown in the picture.

Select the **“Remove selections”** option.

Select the purple surface above the line as shown in the picture. 

Surface Split Options: Natural

Click OK 



Create a Surface Loft

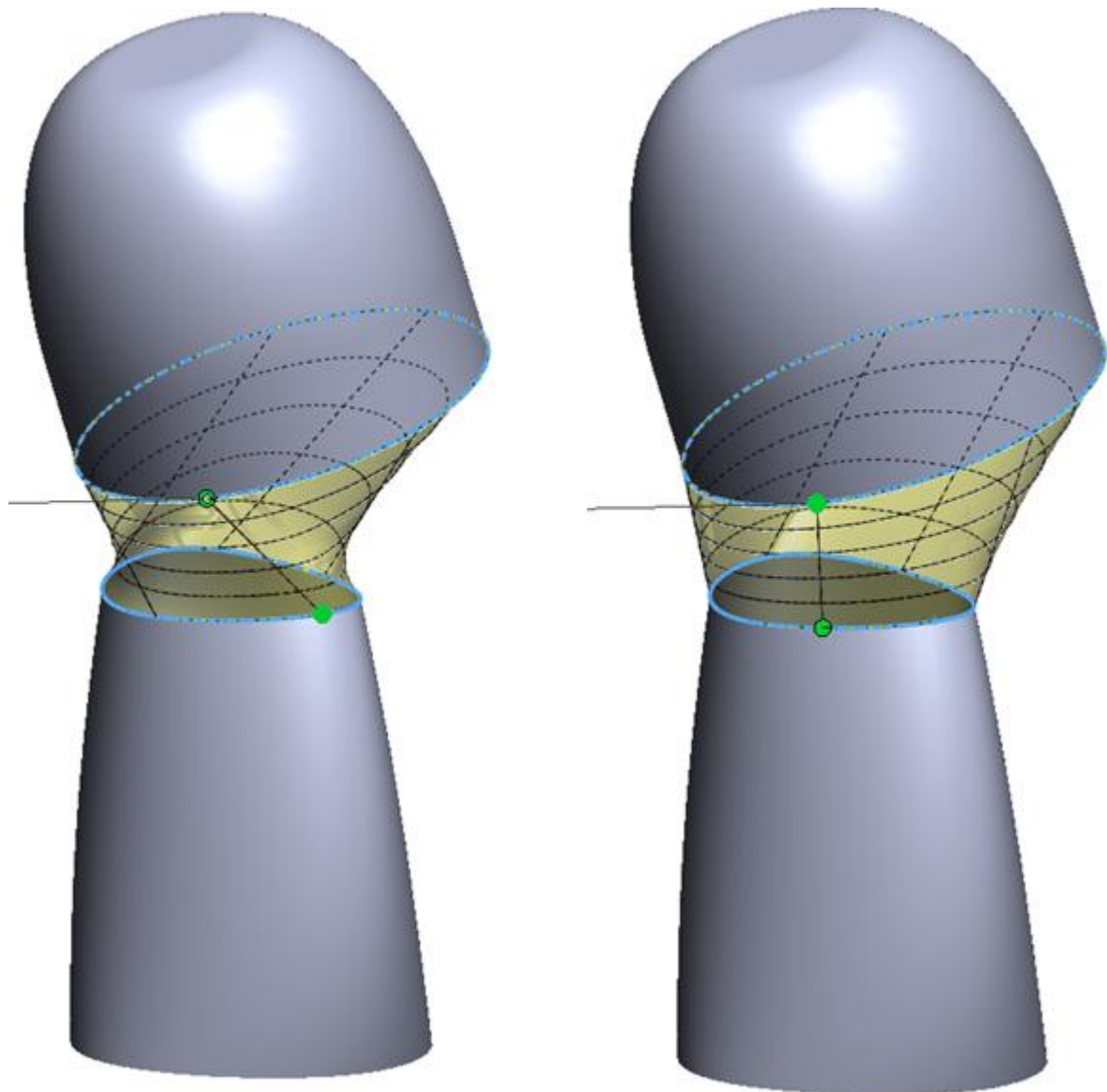
Go to **Insert > Surface > Loft** or click at the Surface icon 

Click in the Profiles box 

Select the two edges as shown in the picture

Make sure that the green balls are both on the same end as shown in the picture

If not, click and drag them to the other side of the sketch



Make the Loft Curvature

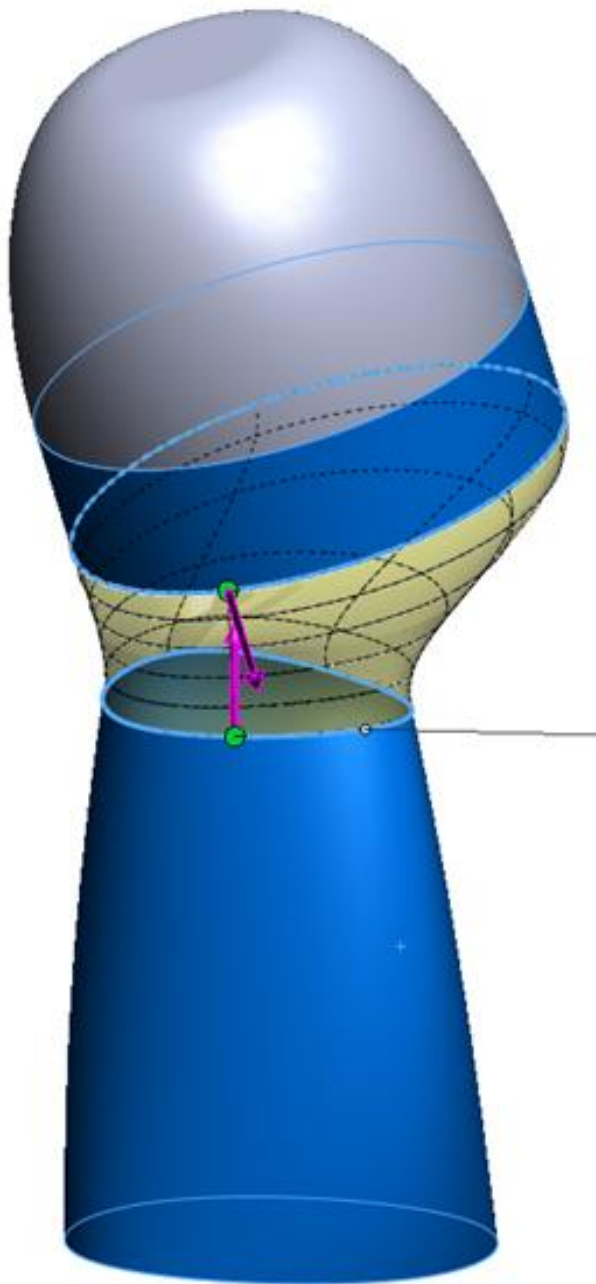
Click at the **Start/End Constraints** box

Set the Start constraint to **Curvature To Face** as shown in the picture

Set the End constraint to **Curvature To Face** as shown in the picture

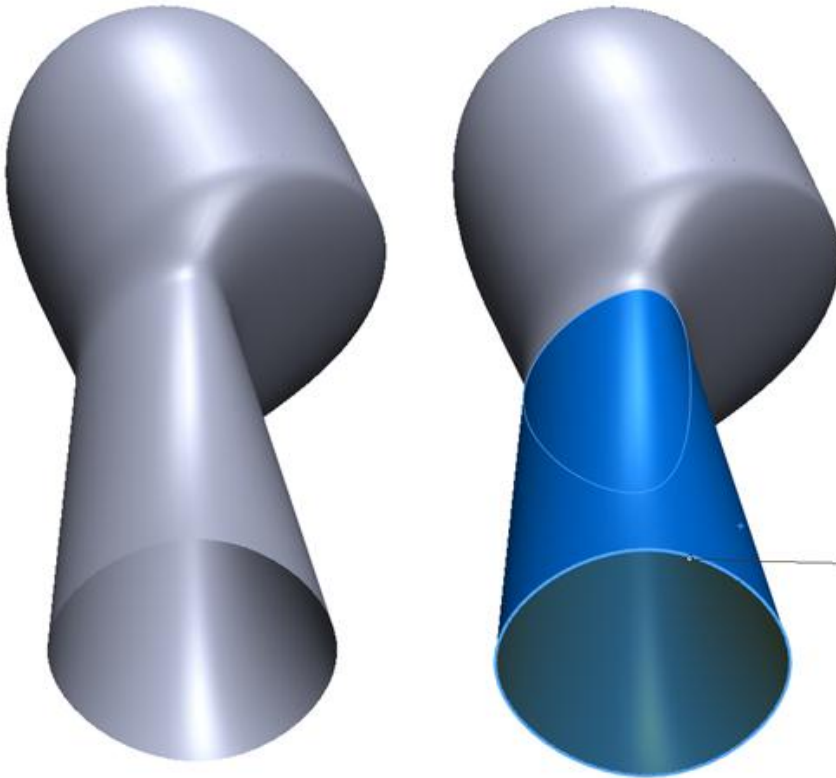
You can optimize the shape of the Loft by changing the Length of the Curvature arrows


Click OK 

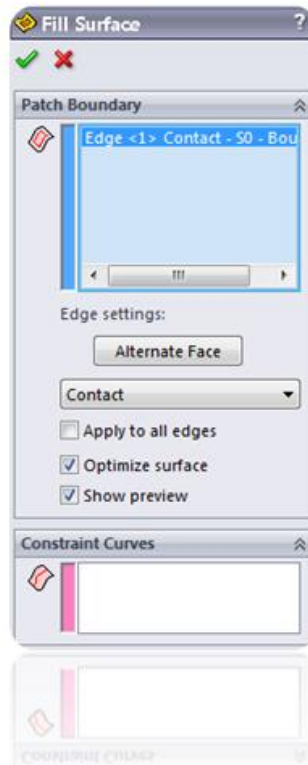


Fill the bottom of the Deoroller


Go to: **Insert > Surface > Fill** or click at the Fill icon 




Click OK 



Knit the surfaces and create a solid body

Go to **Insert > Surface > Knit** or click at the Surface Knit icon 

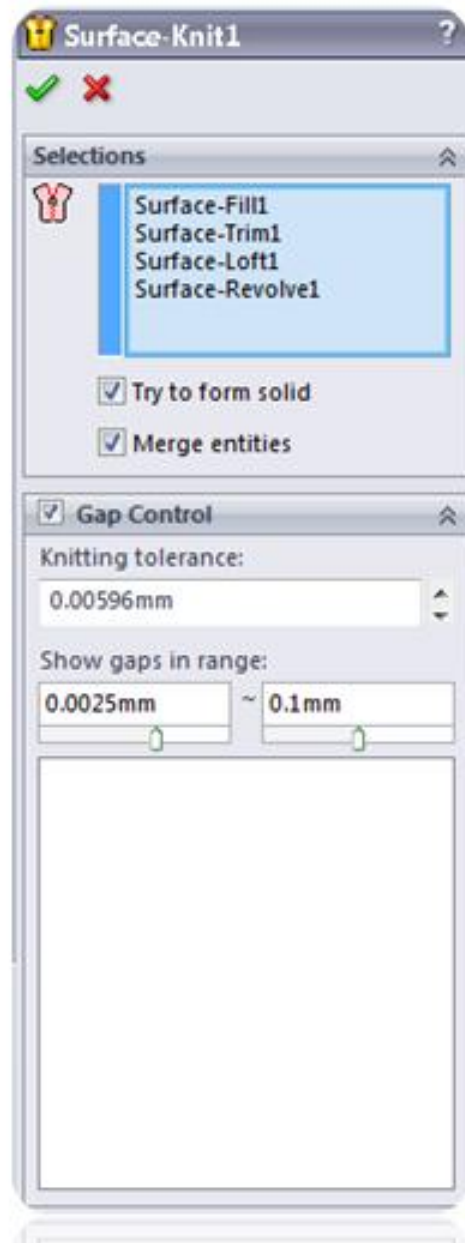
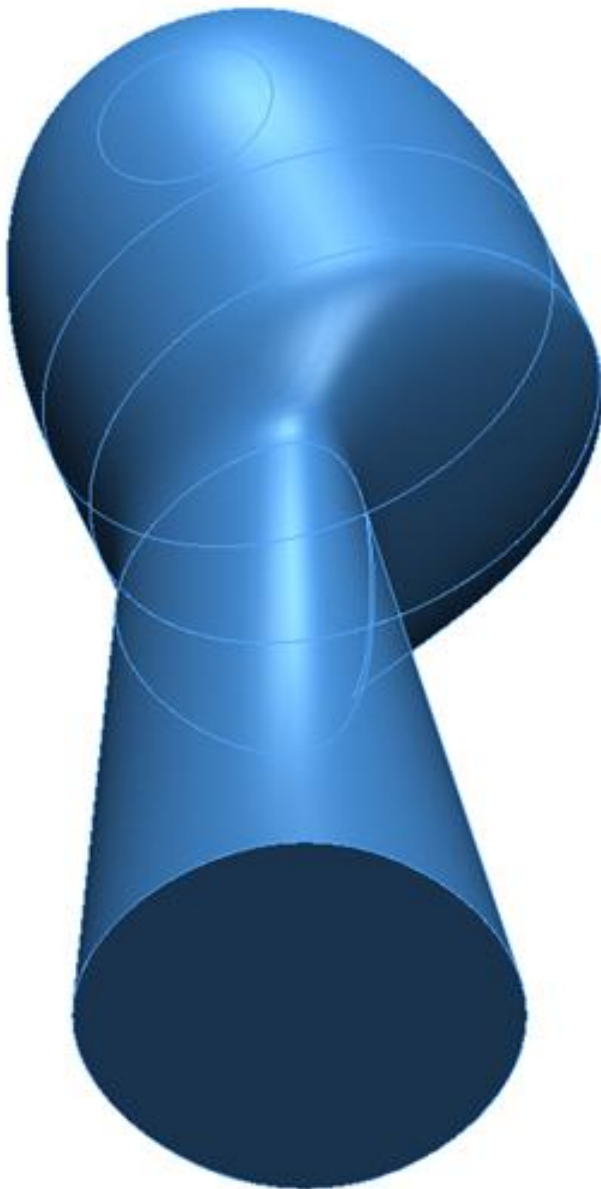
Click in the Selections box and select the 4 blue surfaces 

Select the **“Try to form solid”** option

Select the **“Merge entities”** option

Deselect the **“Gap Control”** option


Click OK 




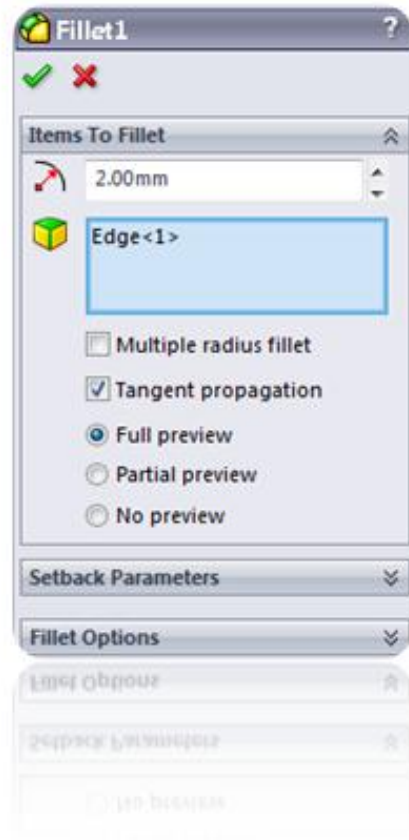
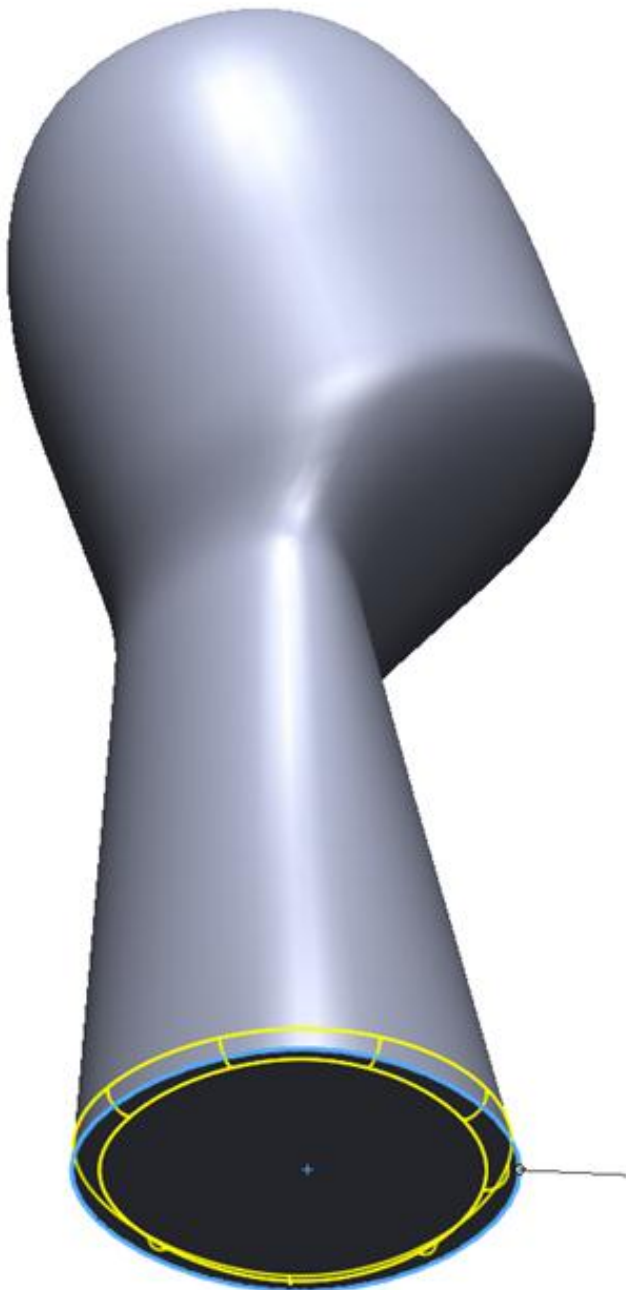
Create a fillet on the edge of the bottom

Go to: **Insert > Features > Fillet/Round** or click at the Fillet icon 

Select the edge as shown in the picture.

Change the Radius into 2 mm 

Click OK 




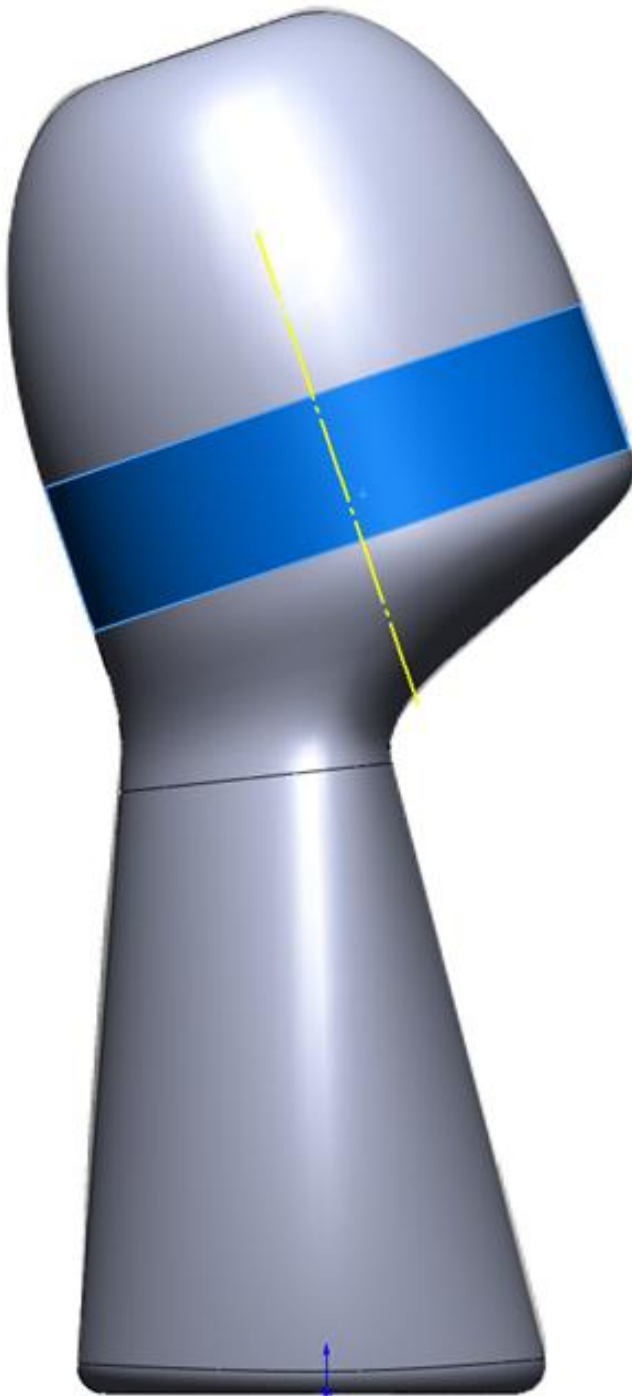
Create a new Axis

Go to: **Insert > Reference Geometry**


Select the **Cylindrical/Conical Face** option


Select the blue surface as shown in the picture


Click OK 

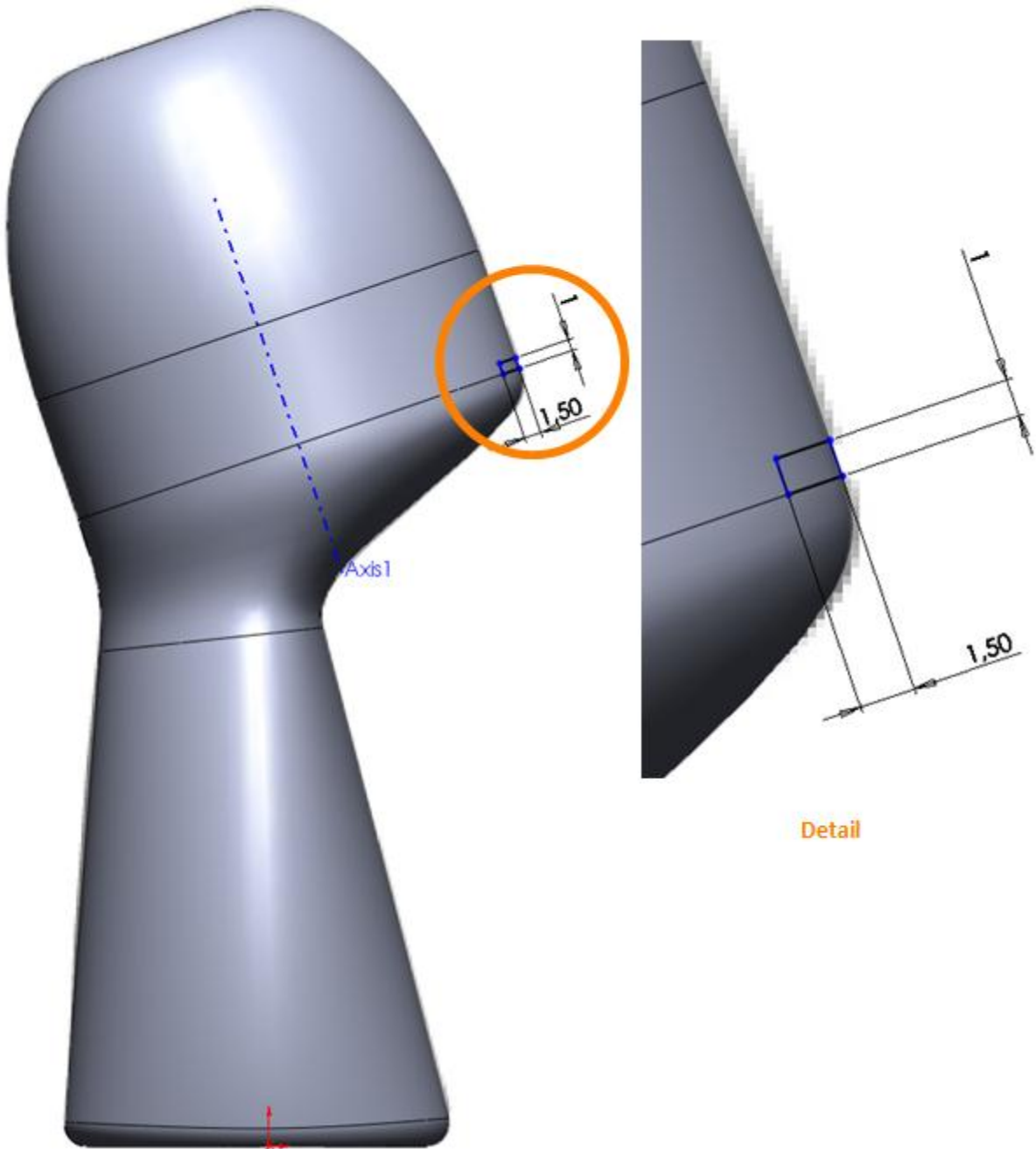


Create a 2D sketch

Select the Right Plane in the feature tree and create a sketch by clicking on the 2D Sketch icon 

Draw the rectangle using the 3 Point Corner Rectangle option as shown in the picture and detail 

Change the dimensions by clicking at the dimension button 

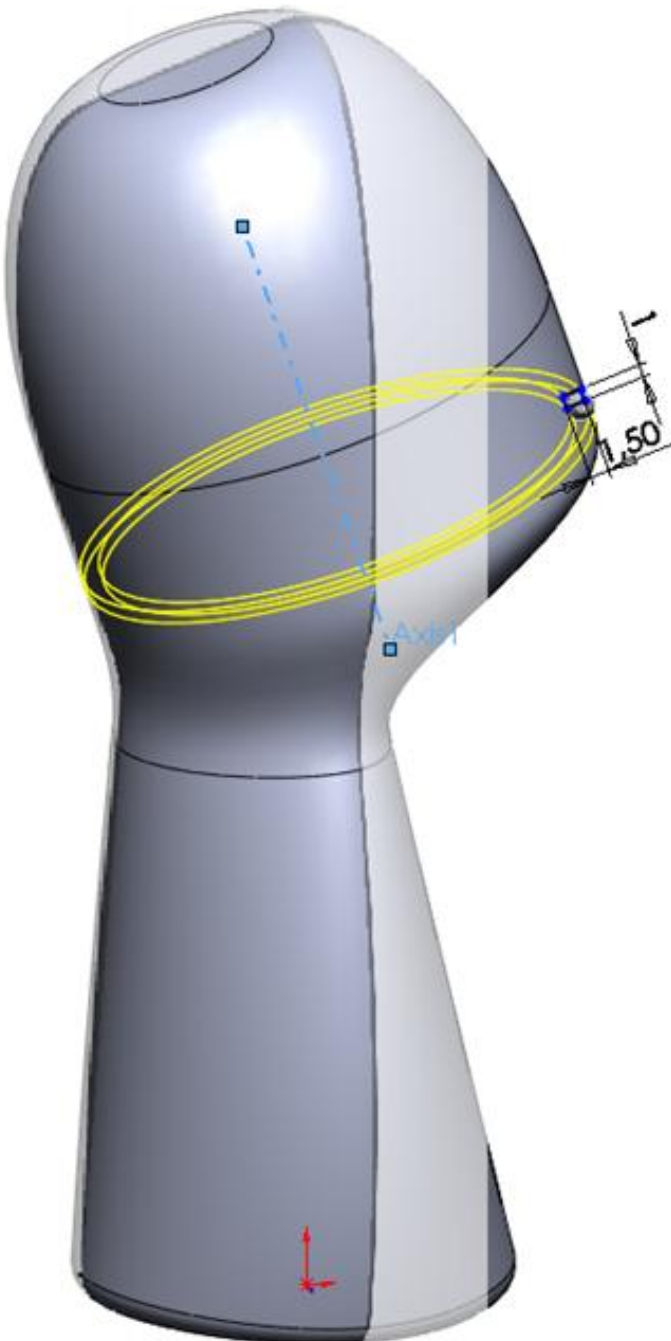


Create a Revolved Cut

Go to: **Insert > Cut > Revolve** or click at the Cut Revolve icon 

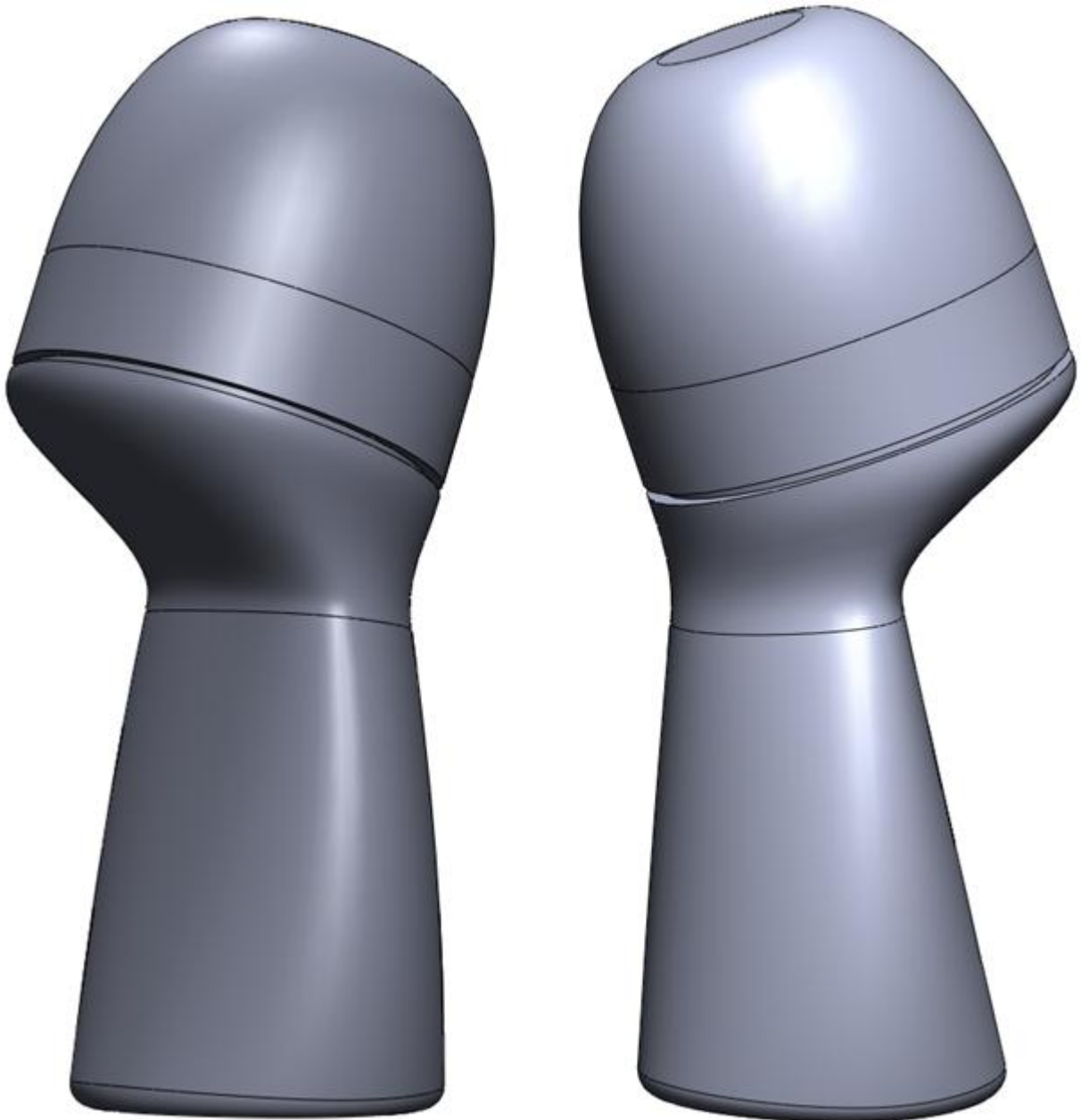
Axis of revolution : Select Axis1.

Click OK 



Save  the file with the following name: **Deoroller.SLDPRT**

Congratulations, you just finished the Deodorant roller!



Did you like this Ebook? Don't hesitate to send it to your friends!



Renders made in PhotoView360

What's next?

Now that you have made the Double Walled Glass in SolidWorks you can immediately start designing your own products in SolidWorks. With a little bit luck you can create your first SolidWorks design in a few days...

I know from experience that it's hard to start without any basic knowledge of SolidWorks. It's not difficult but you have to know all the possibilities and different features of SolidWorks and sometimes it's difficult to keep up the motivation.

New Way To Learn SolidWorks

You can now choose to slow down and gently begin to start with SolidWorks but your changes to become a great SolidWorks designer are negligible.

You Should Immediately Go For It!

I can imagine that you're thinking: what a lot of work ... sigh And that's it. You will have to work very hard to learn all the different features of SolidWorks. The most people learn all the features separately and will later try to bring it all together in one product. Usually this learning process takes several years...

Until Now!

Especially for people like you, I have developed the SolidWorks Chopper Tutorial. In this Step-by-Step SolidWorks course I will teach you how to Model, Assemble and Render a complete Chopper in SolidWorks. Click at the green button below or visit

<http://www.SolidWorksChopper.com>

