SOLIDWORKS
CITRUS SQUEEZER EBOOK

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Jan-Willem **Zuyderduyn**

About LearnSolidWorks.com

Hi, my name is Jan-Willem Zuyderduyn and I live in the Netherlands. I am a Lead Product Designer and founder of LearnSolidWorks.com.

Since 2009 I help SolidWorkers to improve their SolidWorks modeling skills by developing practical, step-by-step SolidWorks tutorials.

I am the author of dozens of SolidWorks eBooks and videos (125.000+ downloads) and 4 premium SolidWorks self-study packages all available on LearnSolidWorks.com.

In these practical SolidWorks tutorial packages, I will show you exactly how to model a stunning Aston Martin One-77 sports car, an incredible American Chopper, an enormous 108 ft. Superyacht and even an entire (76 meter!) long Boeing 747-8:

http://learnsolidworks.com/pricing

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Happy modeling!

Jan-Willem Zuyderduyn

Founder LearnSolidWorks.com

How to Model a Citrus Squeezer in SolidWorks?

In this SolidWorks tutorial I will show you how to model the famous Citrus Squeezer of Philippe Starck in SolidWorks.

Philippe Starck is a famous French designer. He's one of the best known designers in the New Design style. The designs of Starck range from a luxury private yacht to mass produced consumer products such as chairs, dinnerware and even complete houses. This citrus squeezer is also called ''Juicy Salif'' It may look like a spider at a first glance. While this juice squeezer does not serve as a practical tool, it is eye-catching and provocative. I thought it would be awesome to make a SolidWorks tutorial about this famous design squeezer.

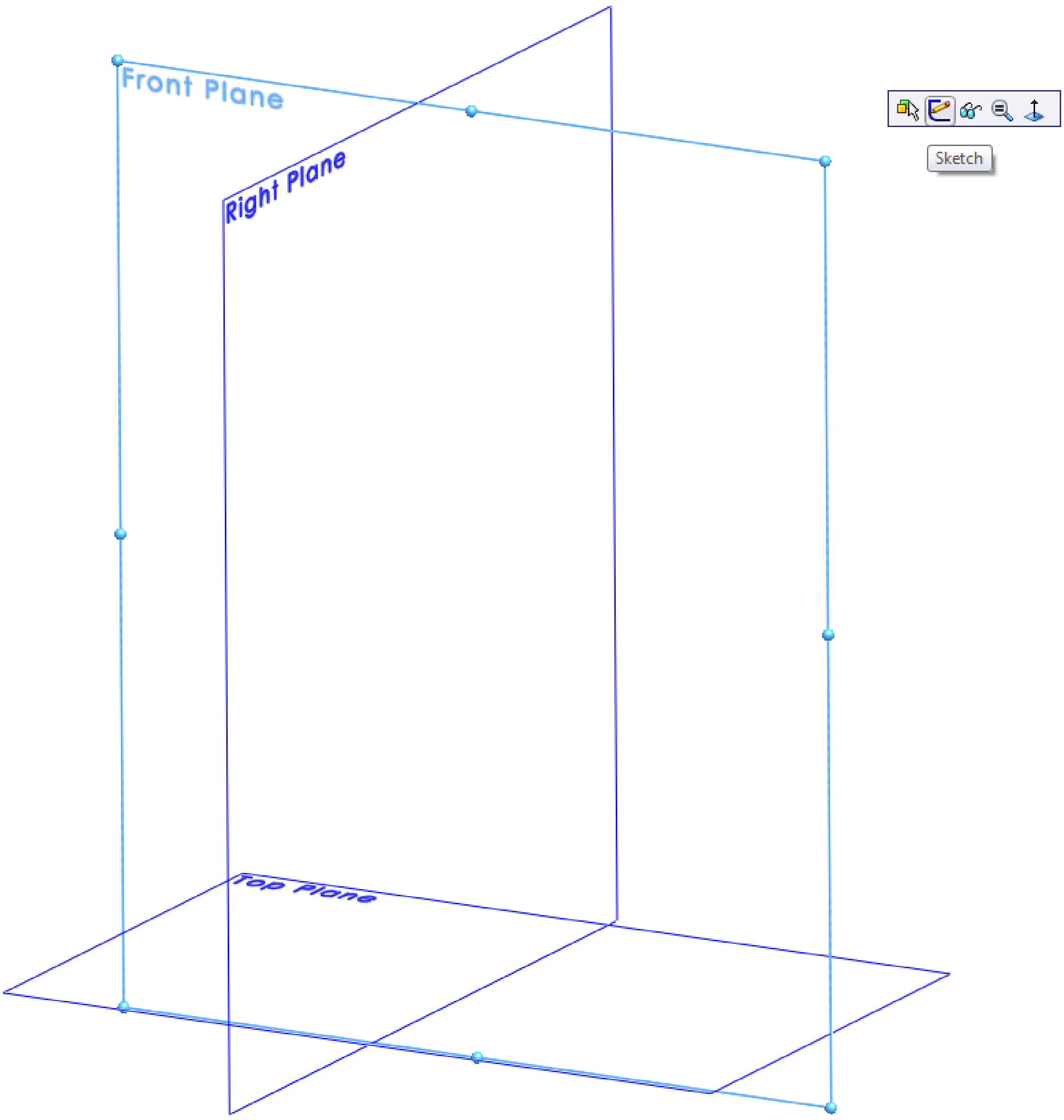
Disclaimer: The intellectual property depicted in this 3D model, is not affiliated with or endorsed by Philippe Starck or Alessi. This 3D model may not be used for any commercial, promotional, advertising or merchandising use.

Open a new part with model units set to millimeters

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

Create a 2D sketch

Select the Front 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 Front plane faces you.



Insert a blueprint

For this tutorial we use a blueprint of the citrus squeezer to create the organic shape.

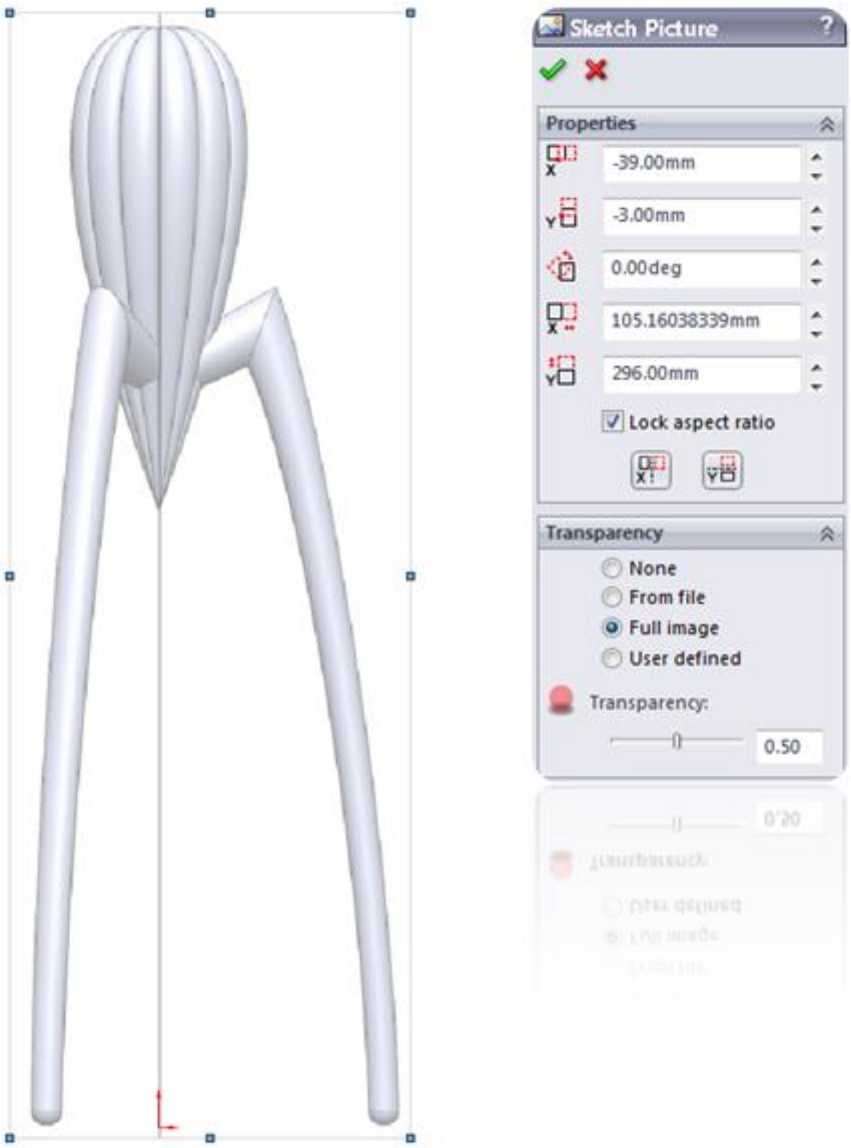
Download the blueprint **here** and save it into your SolidWorks folder
Go to: **Tools > Sketch Tools > Sketch Picture**
Go to your SolidWorks folder and select the blueprint “SIDEVIEW_CITRUS_SQUEEZER.Jpg”

Click: Open

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

Select ‘’Full image’’ in the Transparency tab and change the transparency into 0.50

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



Create another 2D sketch

Select the Front Plane again and create another sketch by clicking on the 2D Sketch icon

Draw two centerlines

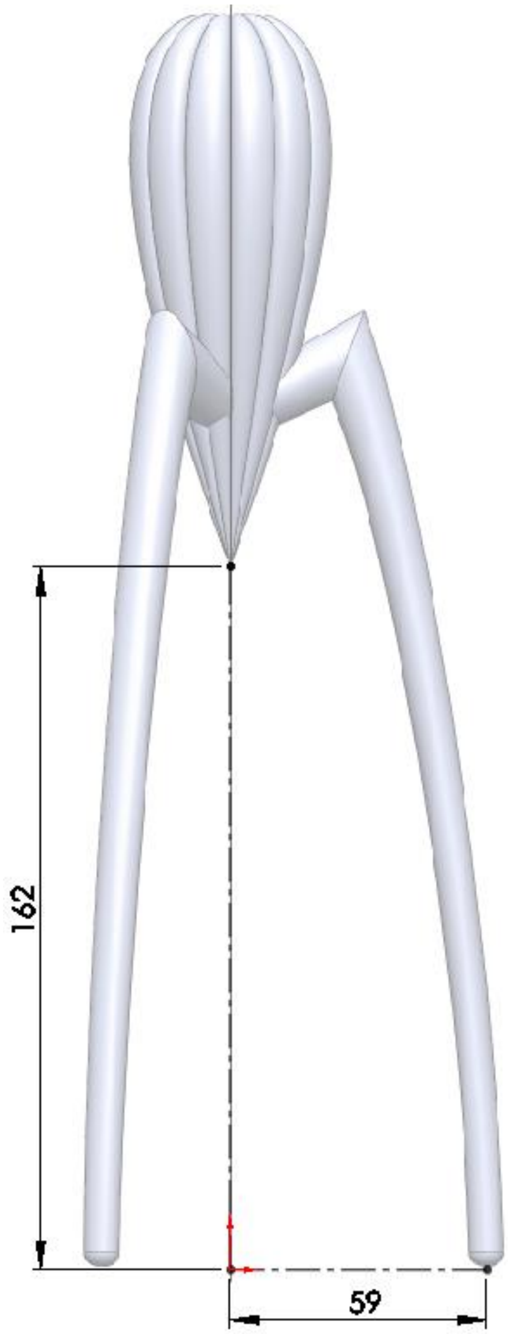
Go to **Tools > Sketch Entities > Centerline** or click at the Centerline icon

Draw a vertical centerline that starts at the origin.

Change the length of the line into 162 mm by clicking at the dimension button

Draw a horizontal centerline that starts at the origin.

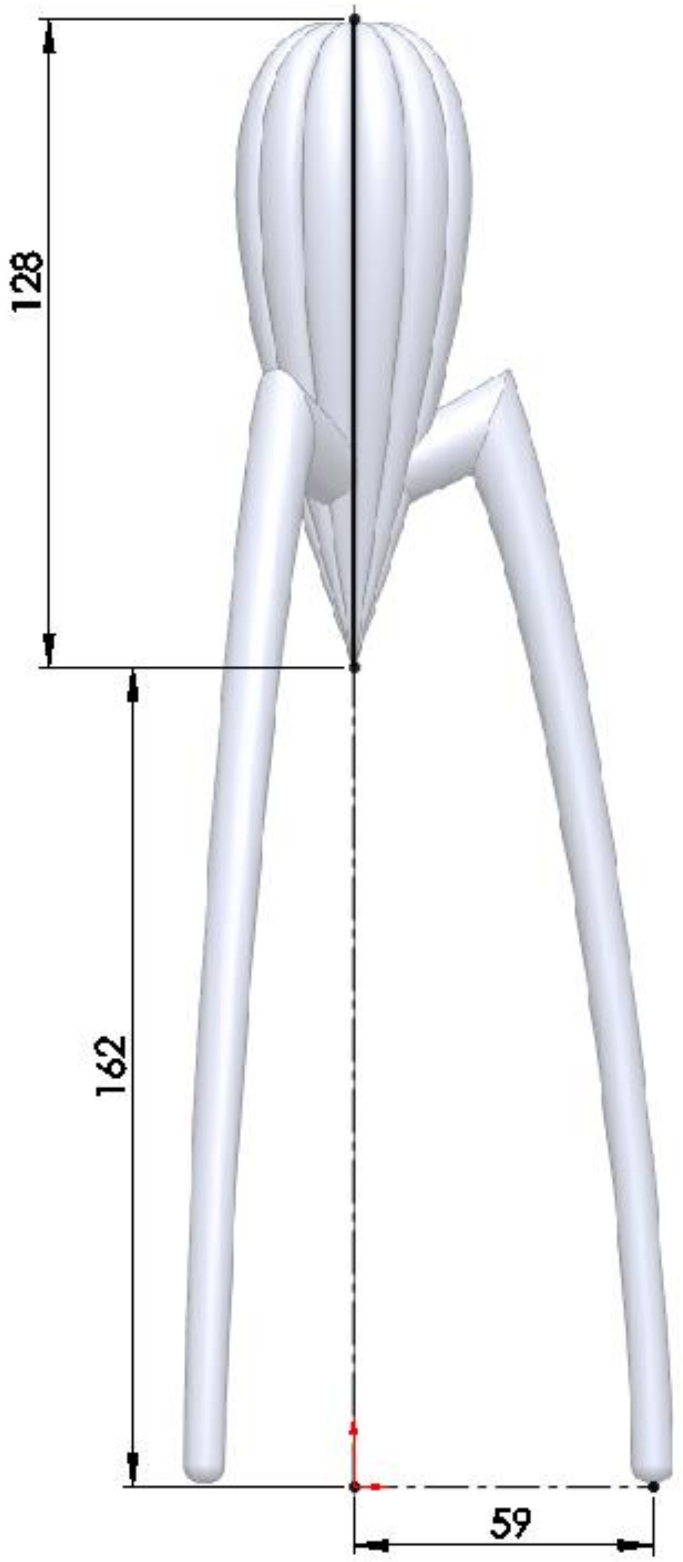
Change the length of the line into 59 mm by clicking at the dimension button



Draw a vertical line

Go to **Tools > Sketch Entities > Line** or click at the Line icon
Draw a vertical line that starts at the top of the vertical construction line

Change the length of the line into 128 mm by clicking at the dimension button

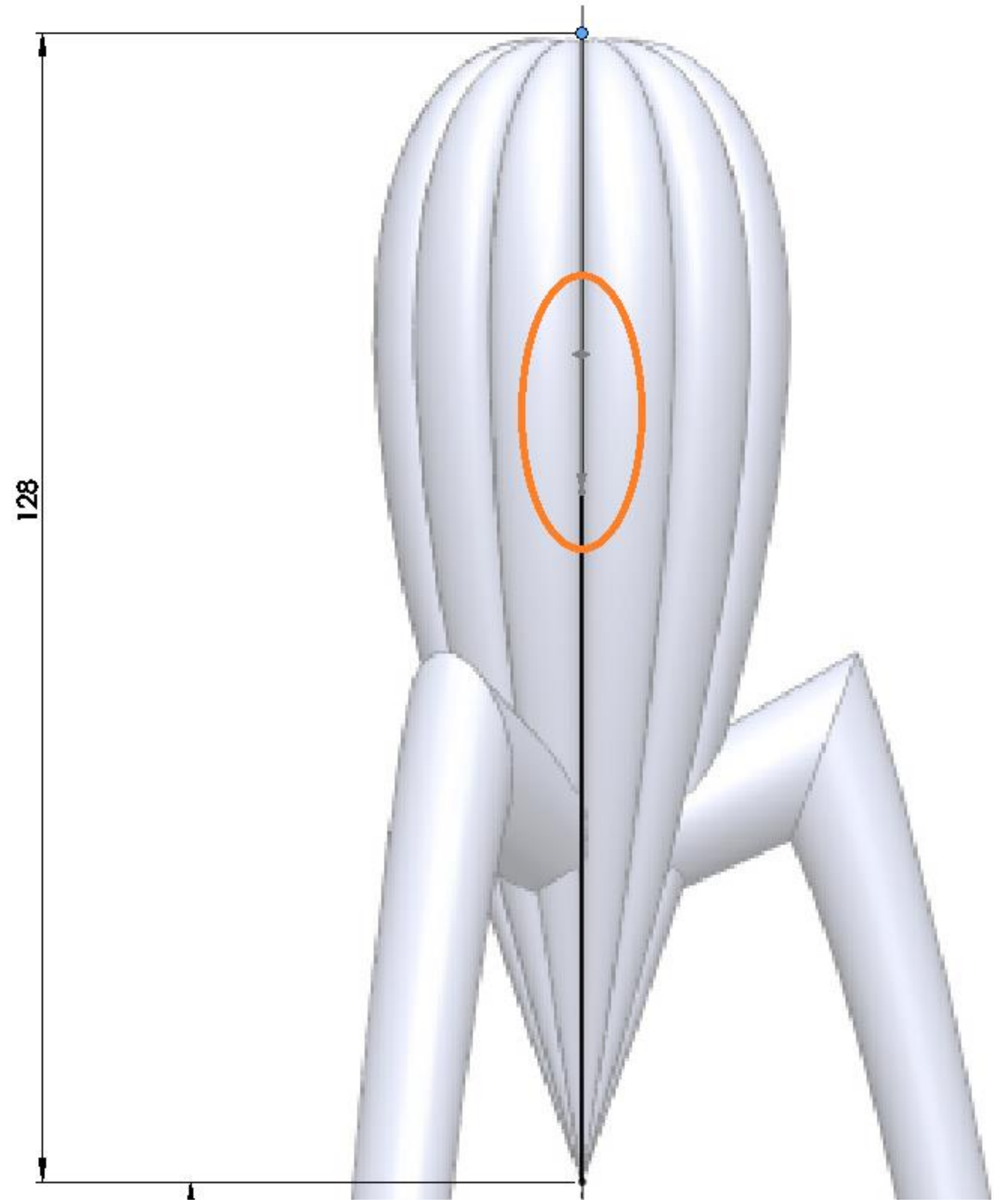


Draw a spline without midpoints

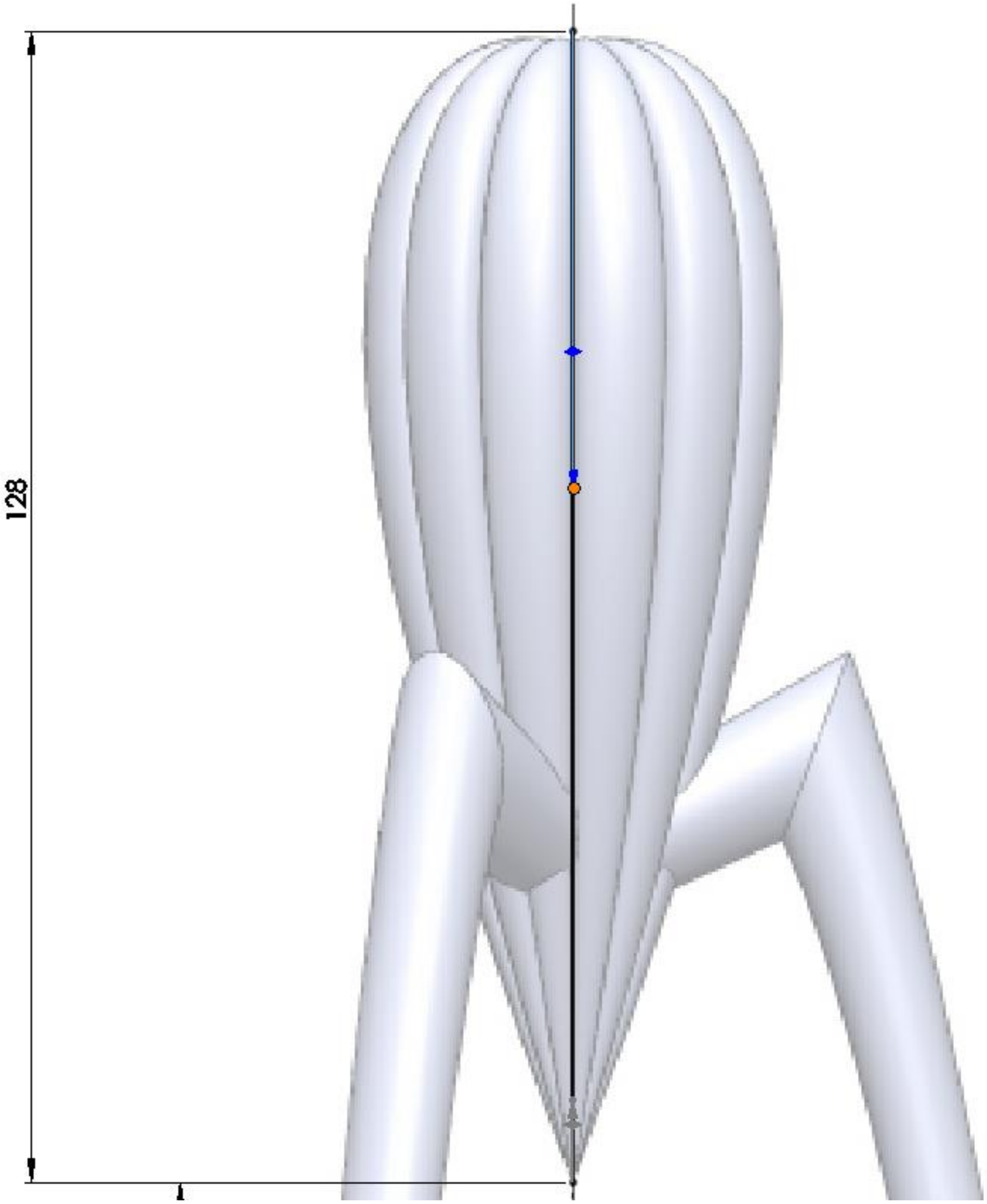
Go to **Tools > Sketch Entities > Spline** or click at the Spline icon 
Draw a spline, starting at the top point of the solid line and ending at the bottom point of the solid line.

Right mouse button > Select

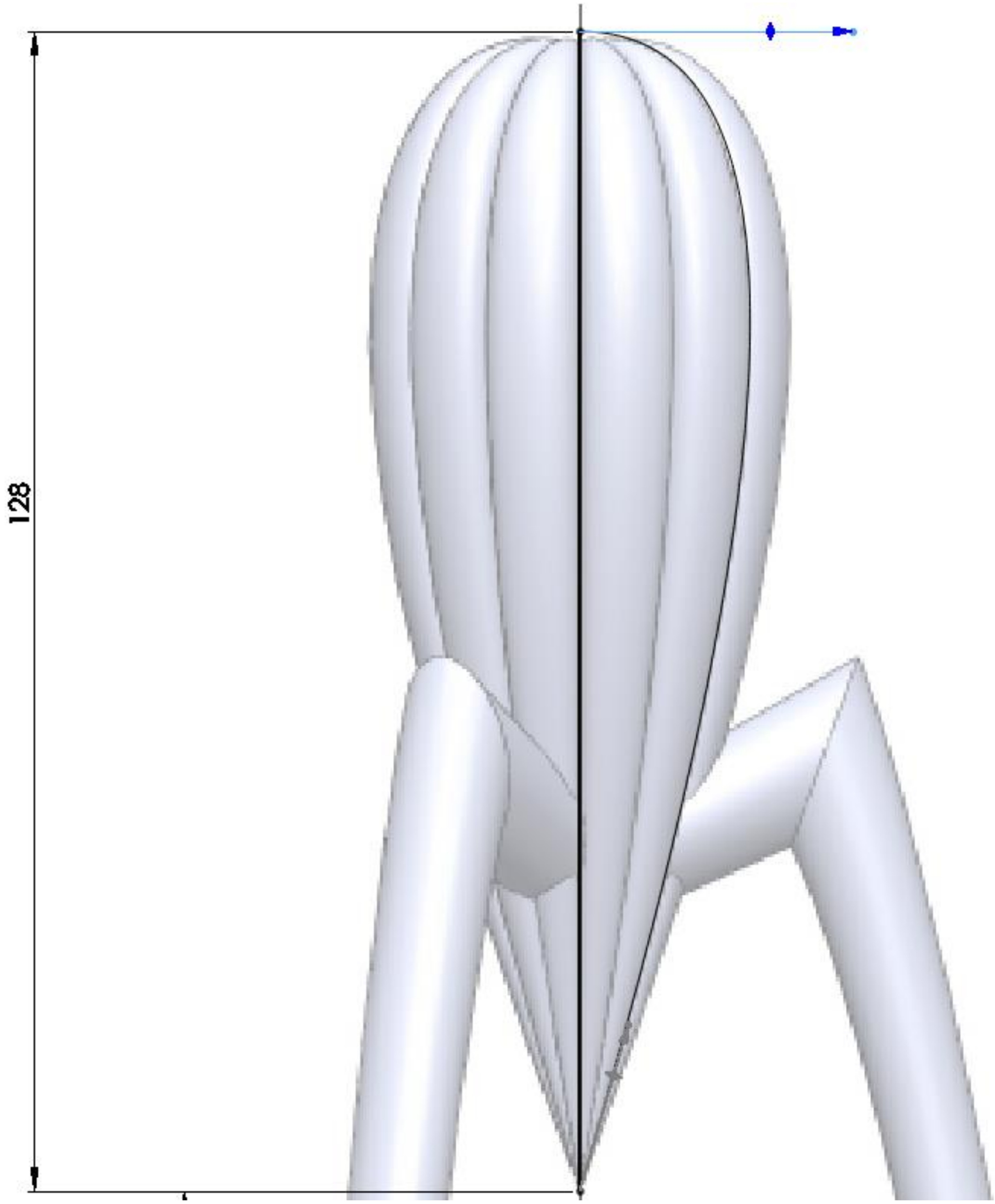
Click at the Top point of the spline > the grey arrows of the Spline appear as shown in the orange circle



Click and drag the round endpoint of the grey arrow as shown in the picture (the orange dot)



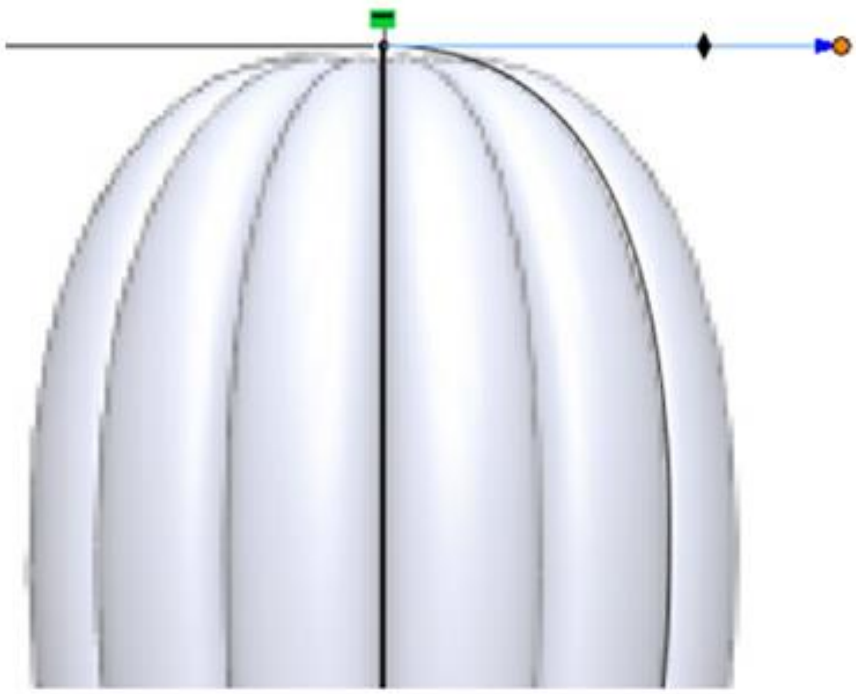
Try to create the inner curve of the blueprint as shown in the picture

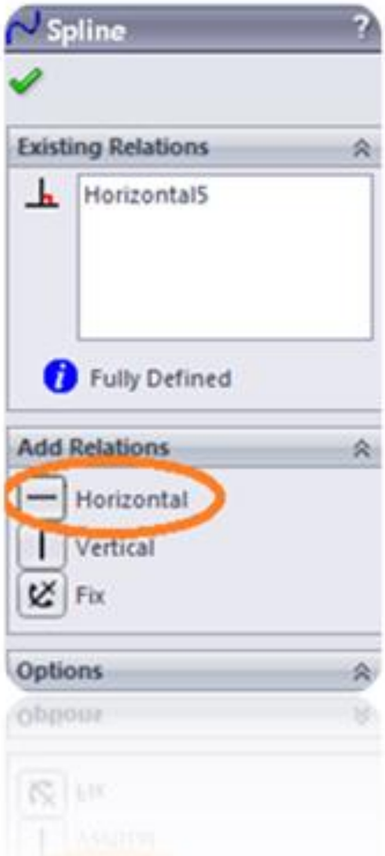


Add a tangency relation add the end of the spline

Click at the orange dot as shown in the picture

Select the Horizontal relation in the Spline menu bar at the left side
The endpoint of the spline is fully tangent now





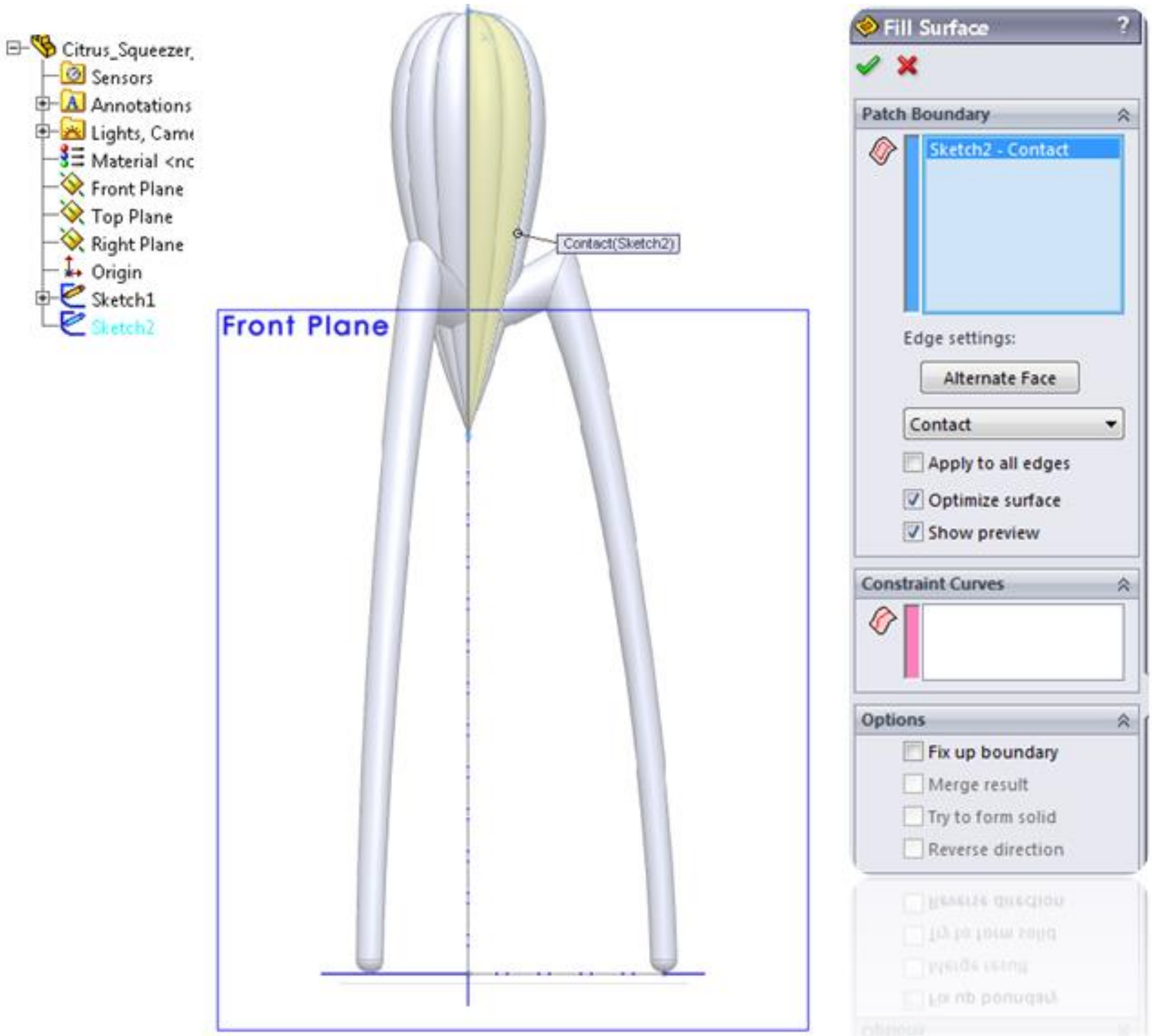
Click at the Sketch button in the upper right corner close the 2D Sketch

Fill the sketch with a surface

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

Select ''Sketch 2'' in the feature tree as Patch Boundary sketch

Click OK

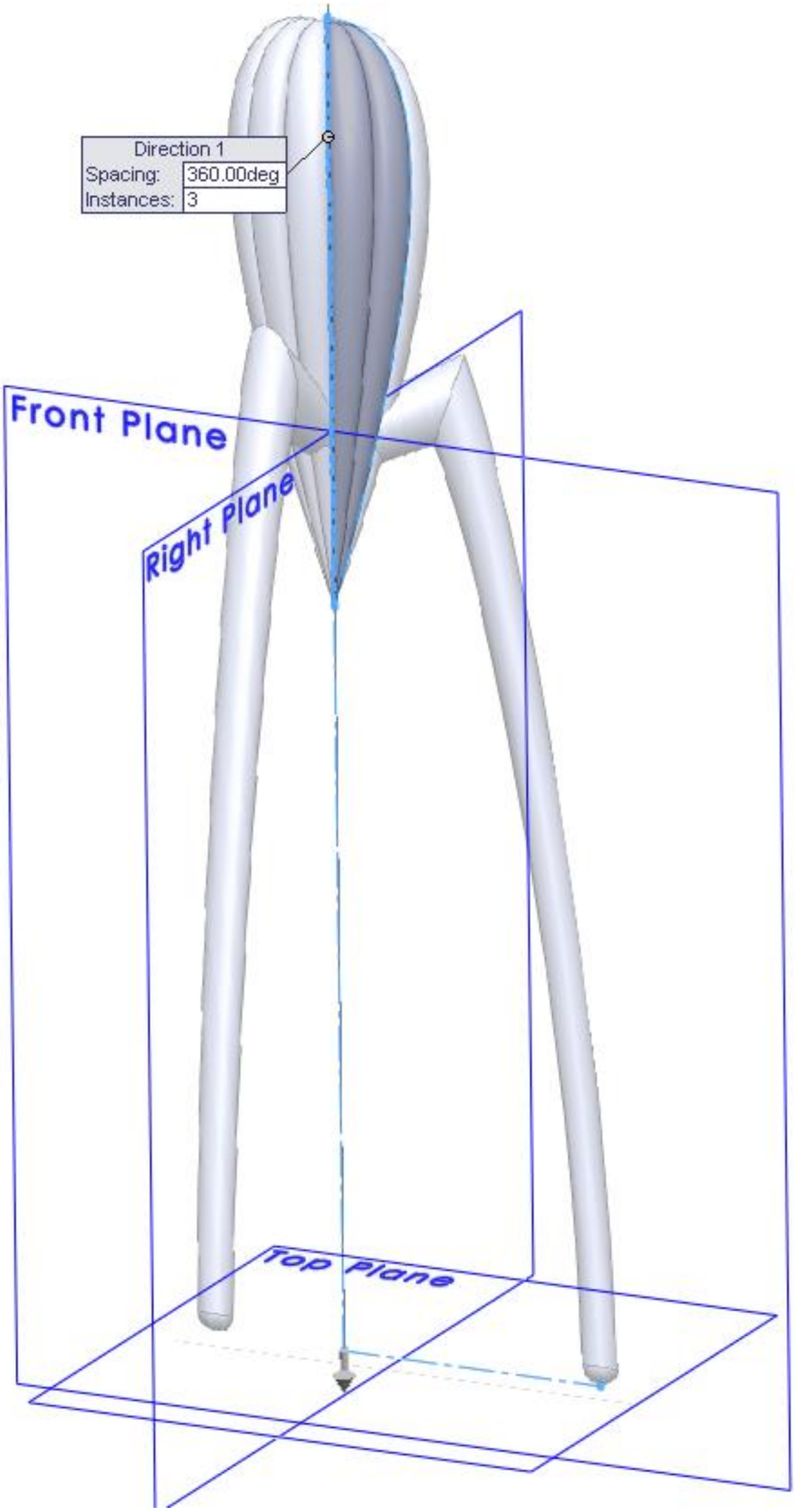


Create a Circular Pattern

Go to **Insert > Pattern/Mirror > Circular Pattern** or click at the Circular Pattern icon

Click in the **''Parameters''** box at the white Pattern-Axis box

Select the vertical edge of the surface fill as shown in the picture



Change the Total Angle into 360 degrees

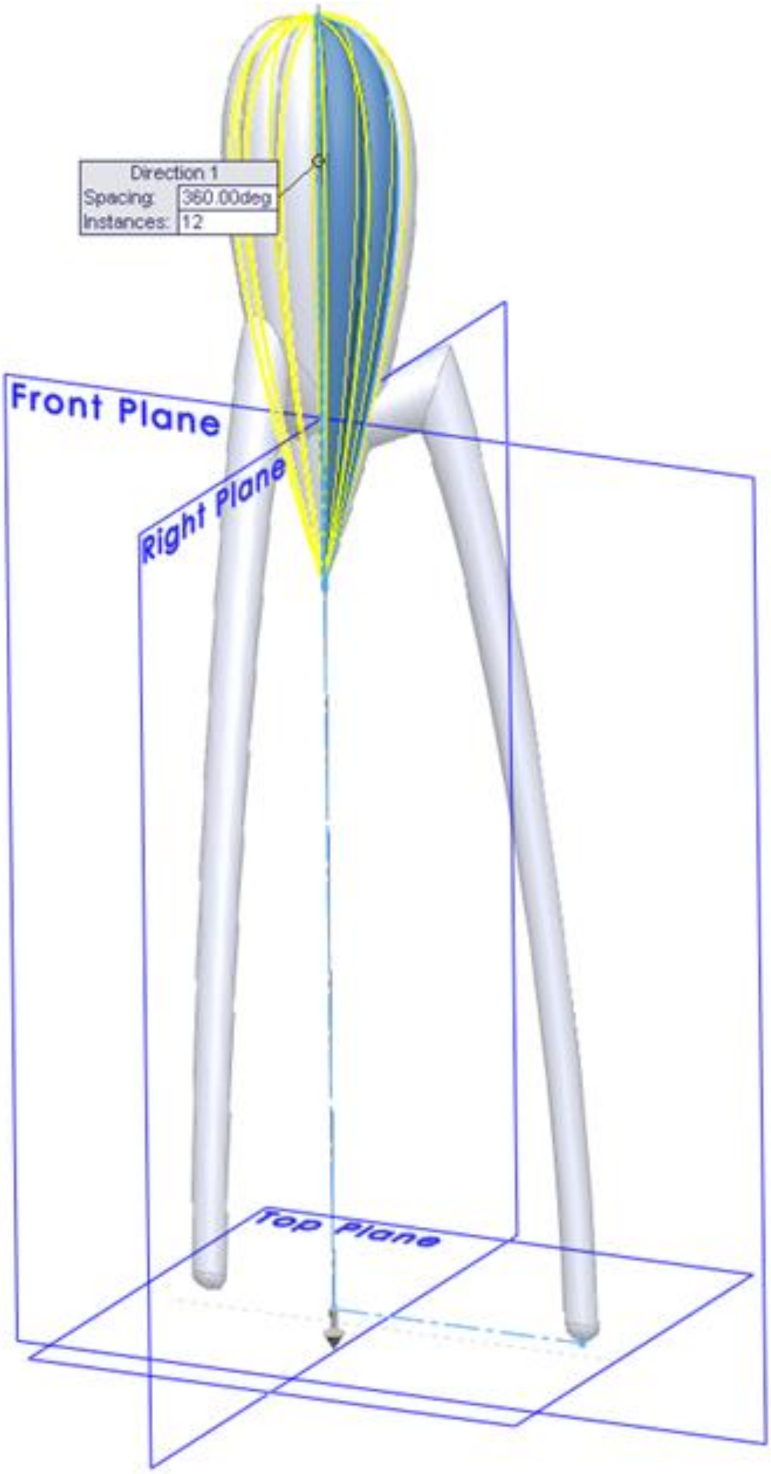
Change the Number of Instances into 12

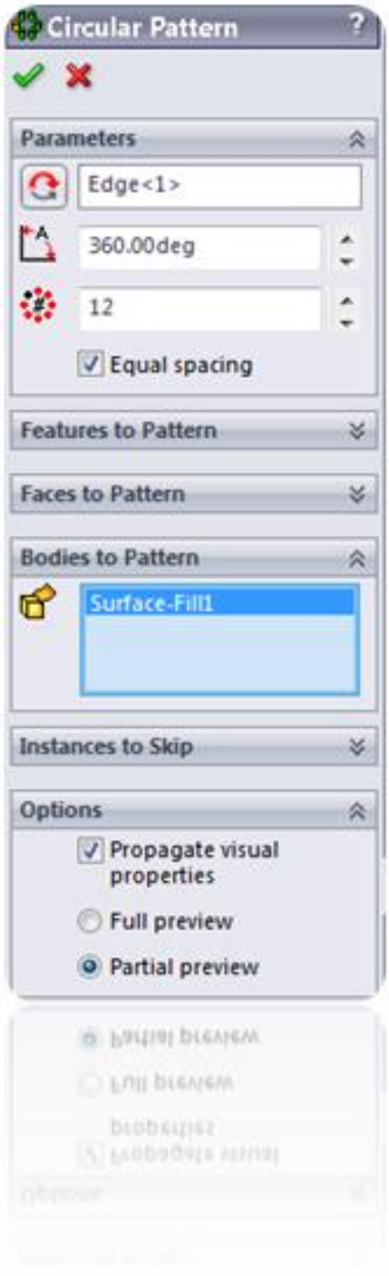
Select the ''Equal Spacing'' option

Click at the **''Bodies to Pattern''** box

Select Surface-Fill1 as ''Surface to Pattern''

Click OK



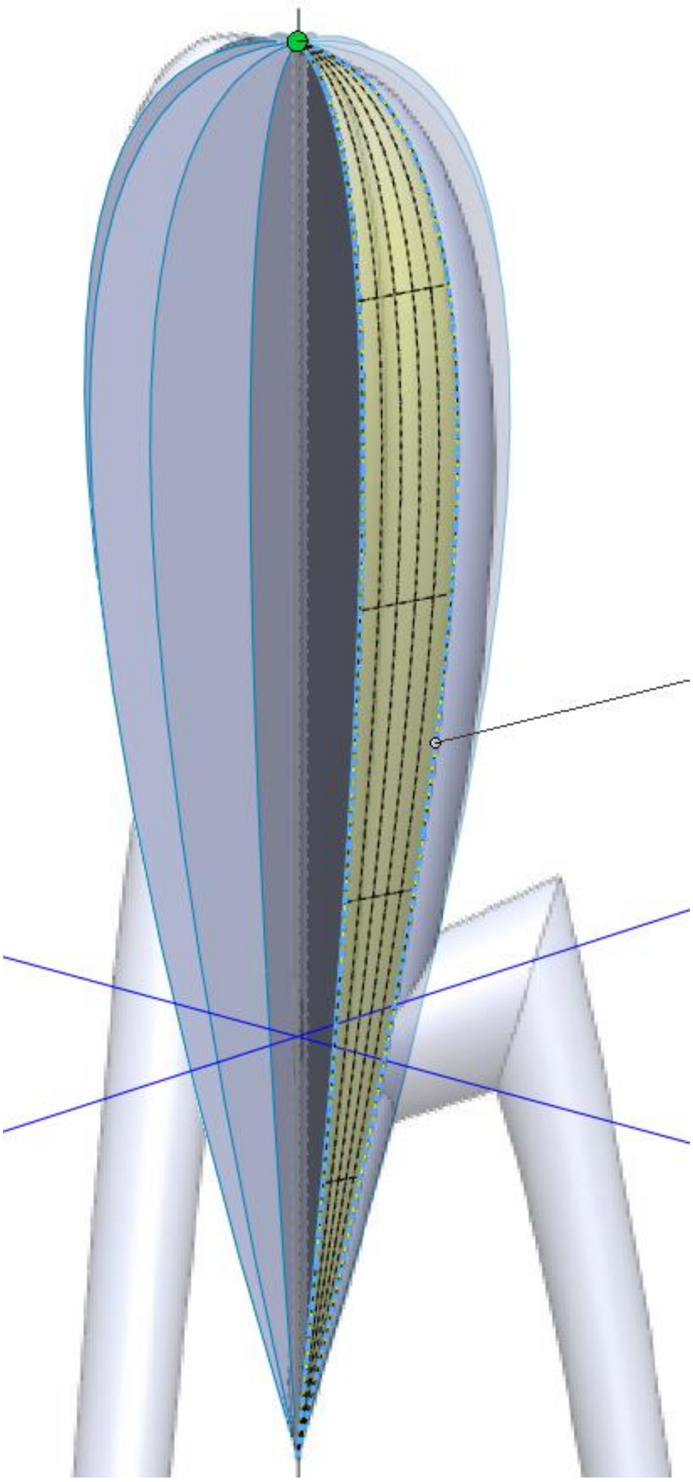


Create a Surface Loft

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

Click in the Profiles box

Select two outer edges of the Circular Pattern as shown in the picture



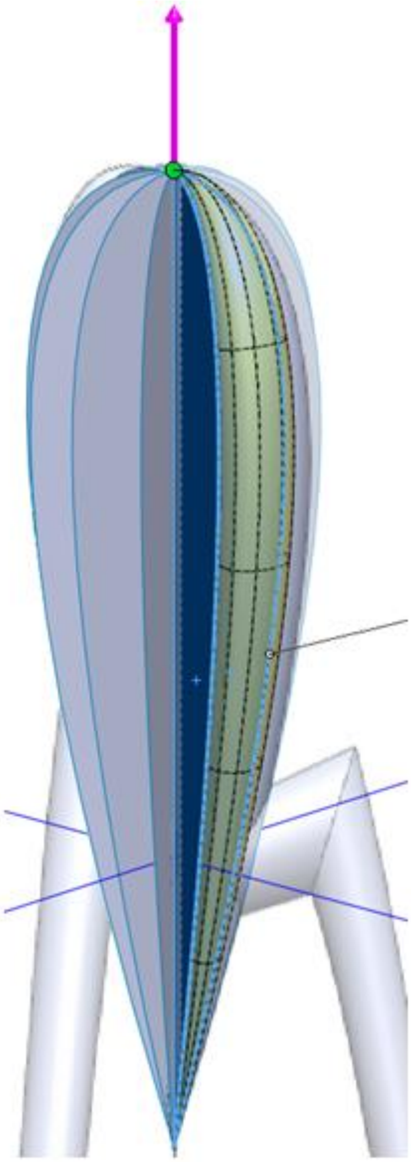
Click at the arrow of the **Start/End Constraints** box to expand this menu
Change the **''Start Constraint''** into Tangency to Face
Change the ''Start Tangent Length'' into 4

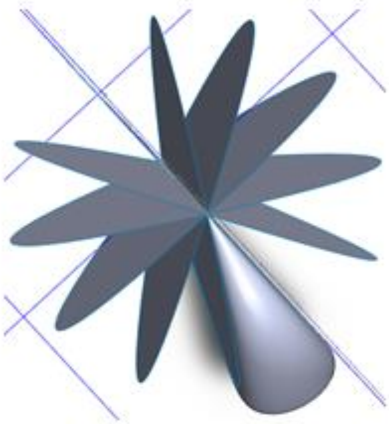
Select the Apply to all boxes

Change the **''End Constraint''** into Tangency to Face
Change the ''End Tangent Length'' into 4

Select the Apply to all boxes

Make sure that the shape of the Loft looks like the one in the picture. If not, try to change the direction of the Tangency to face by clicking on the **''Reverse Tangent Direction''** button
Click OK

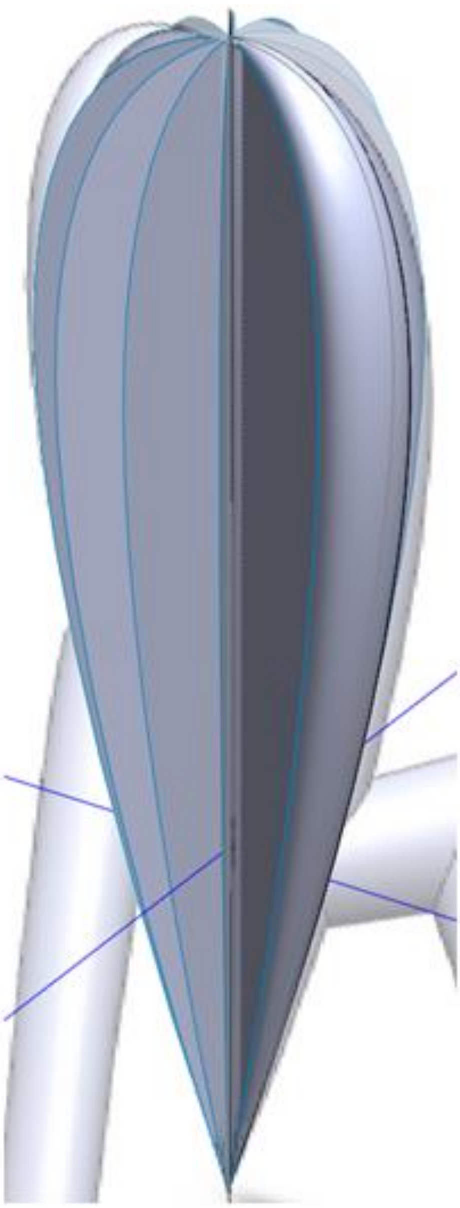


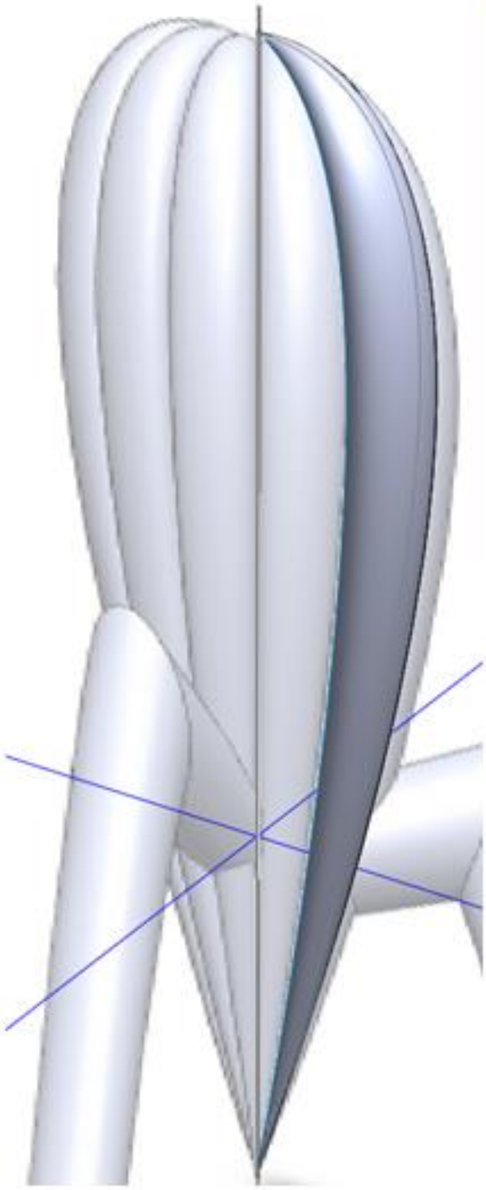


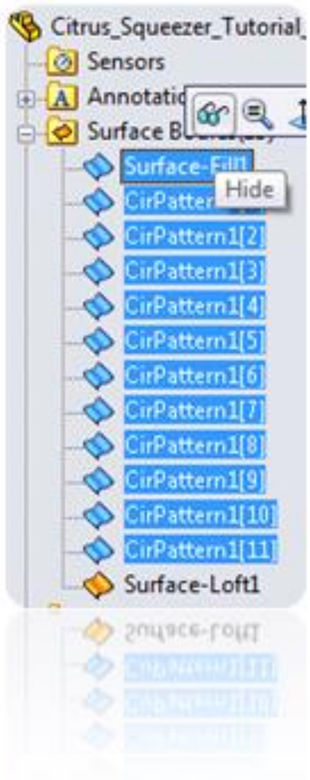
Hide all surface bodies except the Surface Loft

Expand the **Surface Bodies** map in the Feature Tree
Select all Surfaces except Surface-Loft1

Click at the Glasses to hide the Surface Bodies







Create an Axis

Go to **Insert >Reference Geometry > Axis** or click at the Axis icon

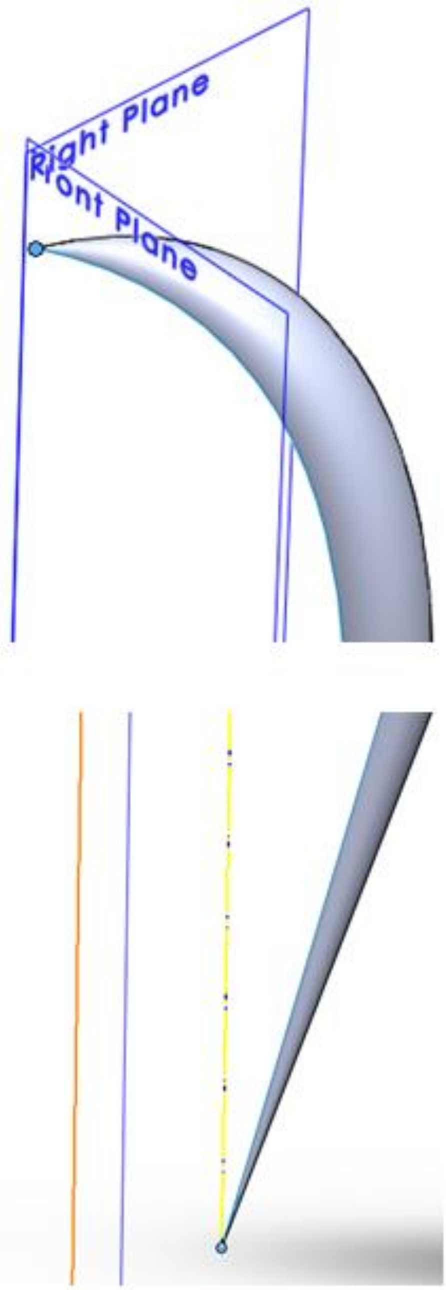
Select the **''Two Points/Vertices''** option

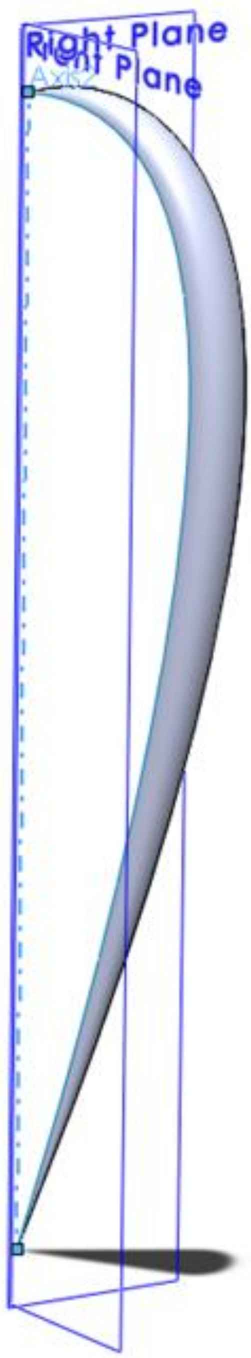
Click at one of the endpoints of the Surface Loft

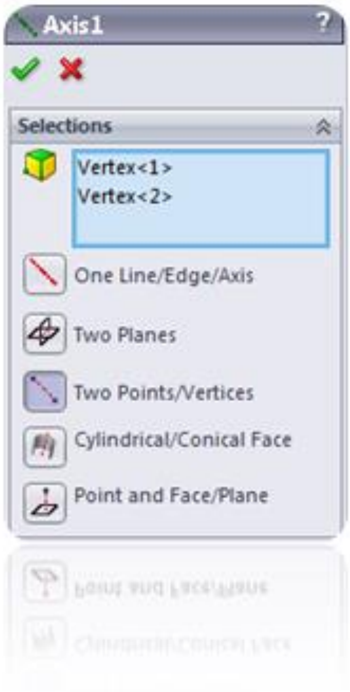
Hold down the Control key

Select the other endpoint of the Surface Loft as well

Click OK







Create another Circular Pattern

Go to **Insert > Pattern/Mirror > Circular Pattern** or click at the Circular Pattern icon

Click in the **''Parameters''** box at the white Pattern-Axis box

Select the new Axis1 as Pattern Axis

Change the Total Angle into 360 degrees

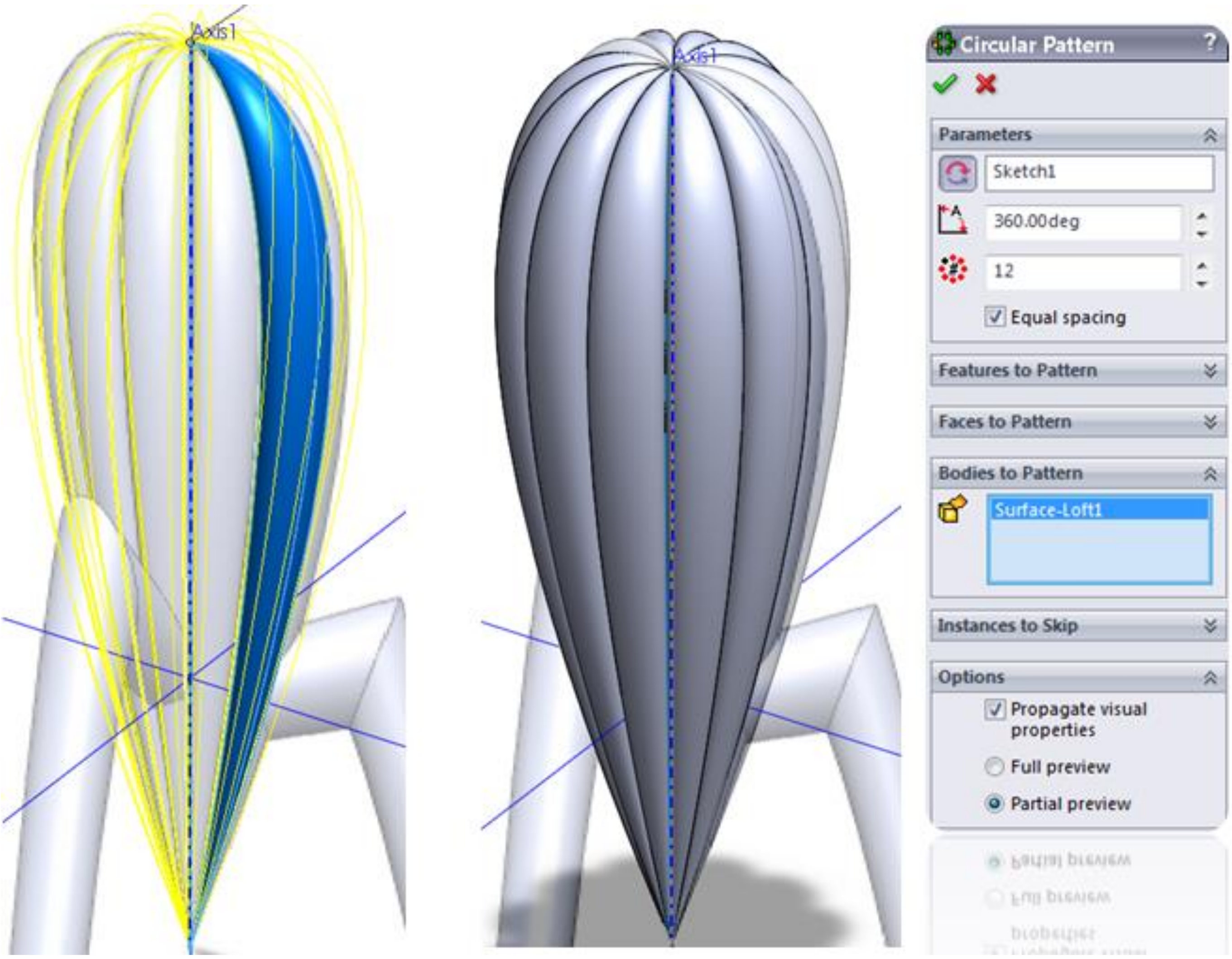
Change the Number of Instances into 12

Select the ''Equal Spacing'' option

Click at the **''Bodies to Pattern''** box

Select Surface-Loft1 as ''Surface to Pattern''

Click OK



Knit the 12 surfaces and create a solid body

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

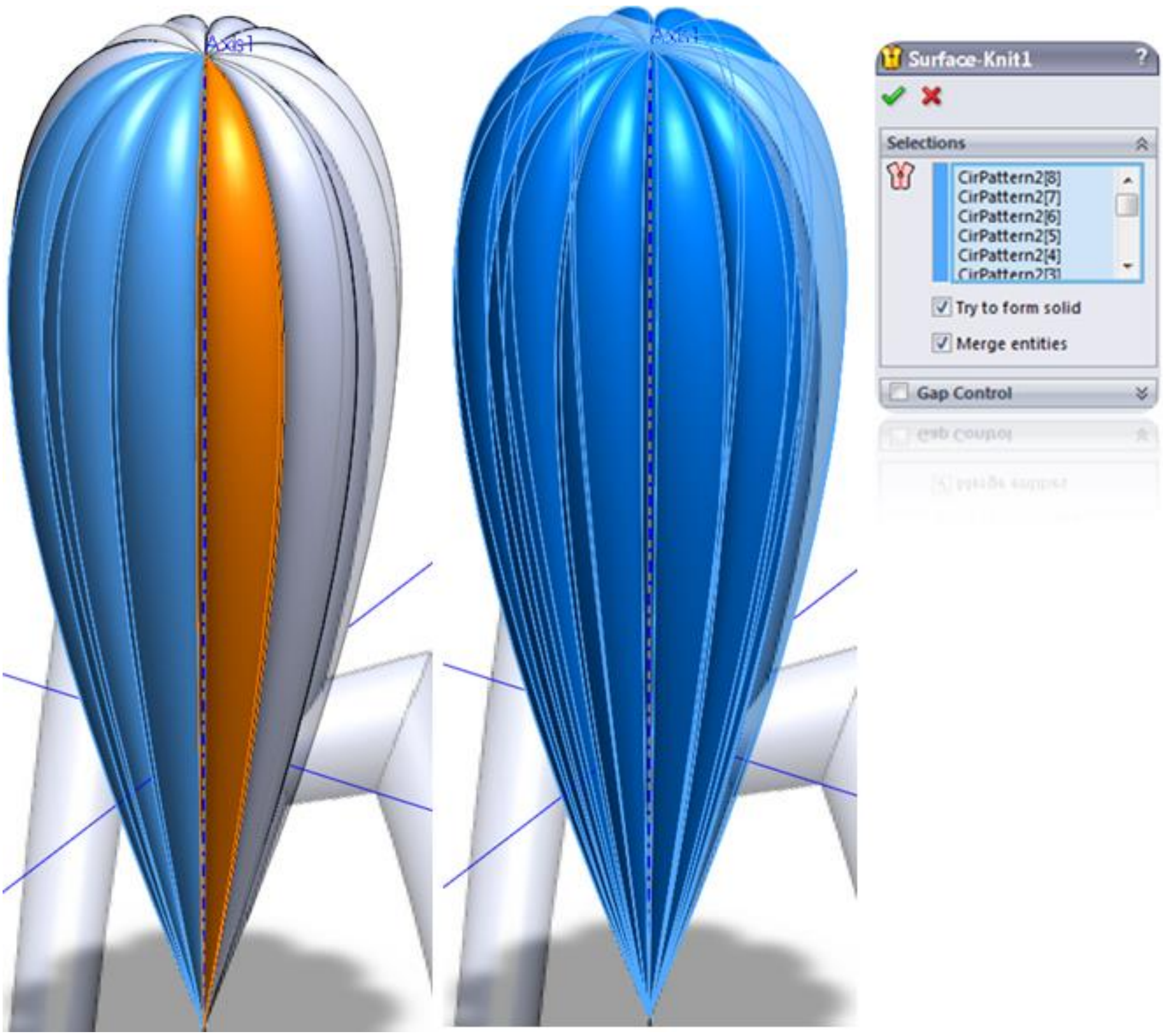
Click in the Selections box and select the 12 Surface Lofts

Select the **"Try to form solid"** option

Select the **"Merge entities"** option

Deselect the **"Gap Control"** option

Click OK



Create another 2D sketch

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

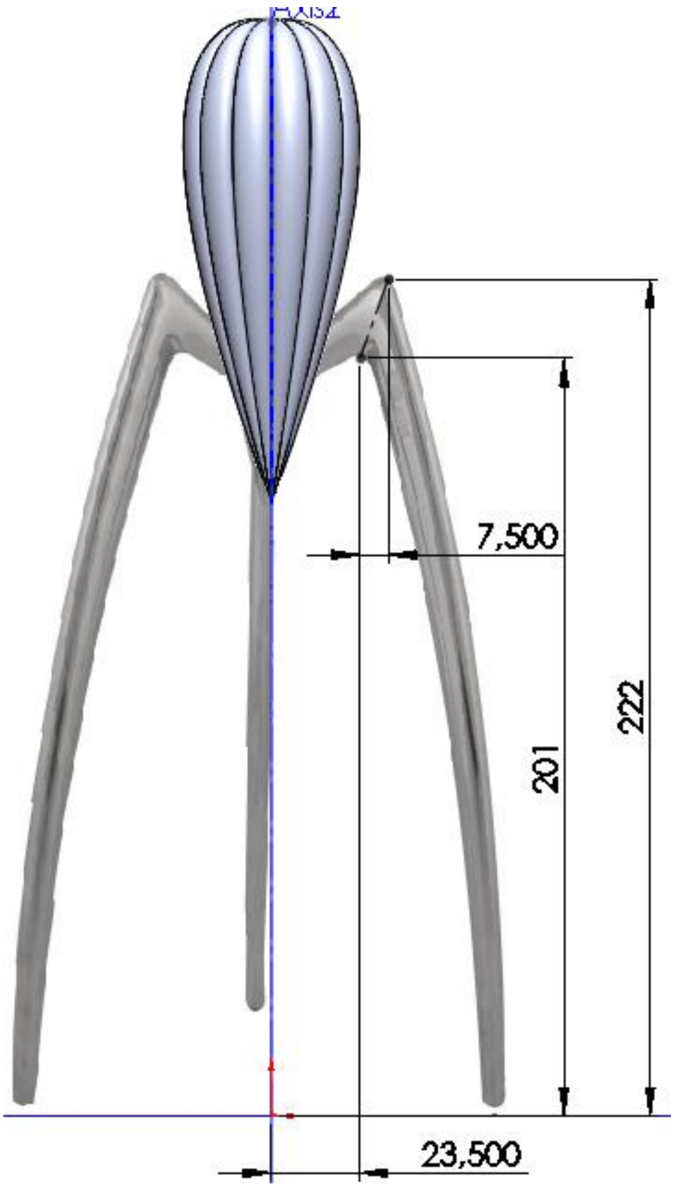
Draw a centerline

Go to **Tools > Sketch Entities > Centerline** or click at the Centerline icon
Draw a diagonal centerline

Change the dimensions of the centerline by clicking at the dimension button
Add the dimensions as shown in the picture

This single centerline we will use to create a new plane

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



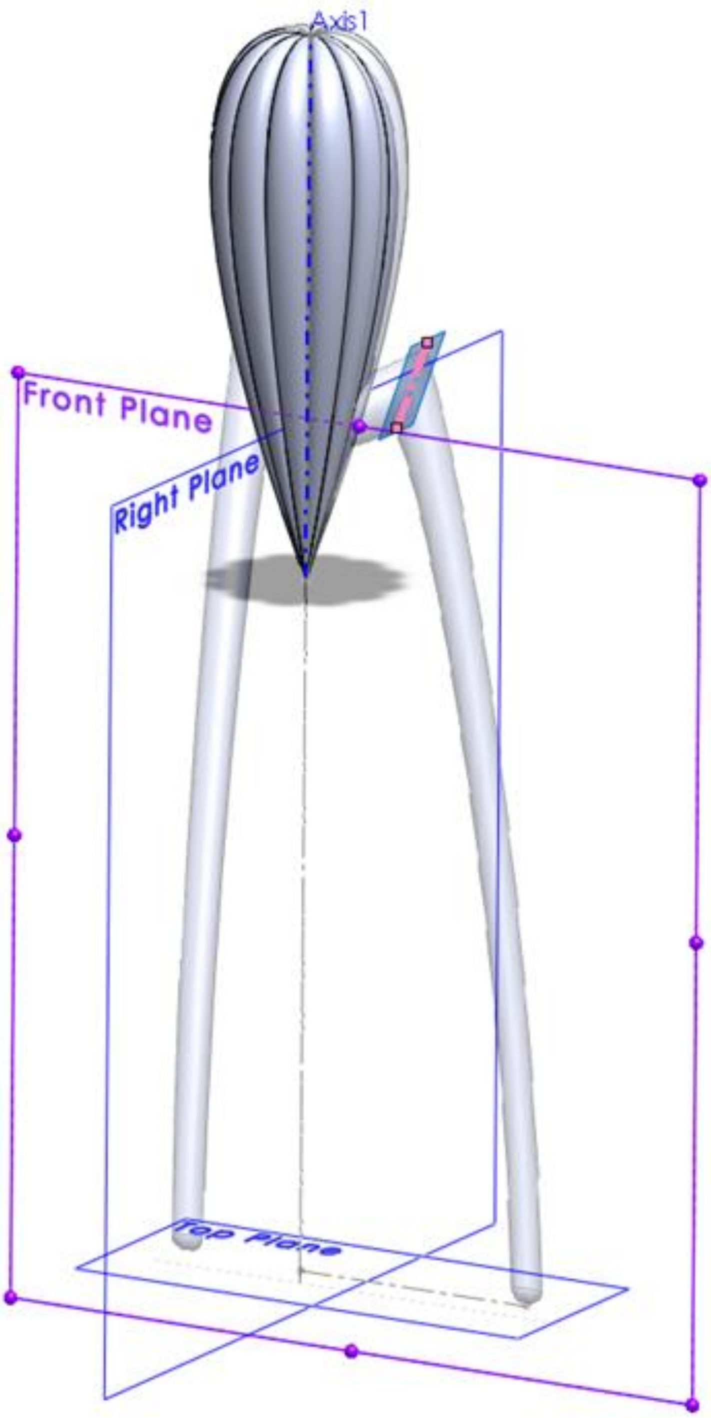
Create a new plane

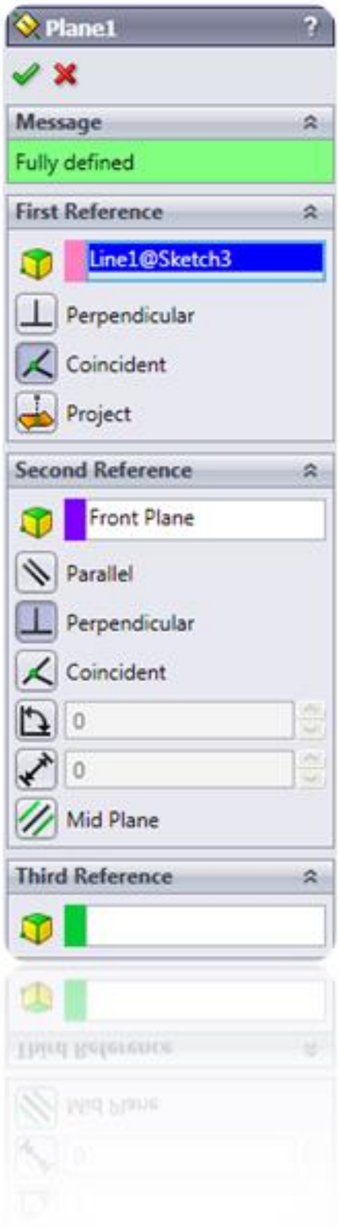
Go to: **Insert > Reference Geometry > Plane** or click at the New Plane icon
Select the new Sketch3 in the feature tree

Hold the Control button and select the Front Plane as shown in the picture

The new plane appears in blue

Click OK





Create another 2D sketch

Select the new Plane1 and create a sketch by clicking on the 2D Sketch icon 

Convert Sketch 3 into the new Sketch4

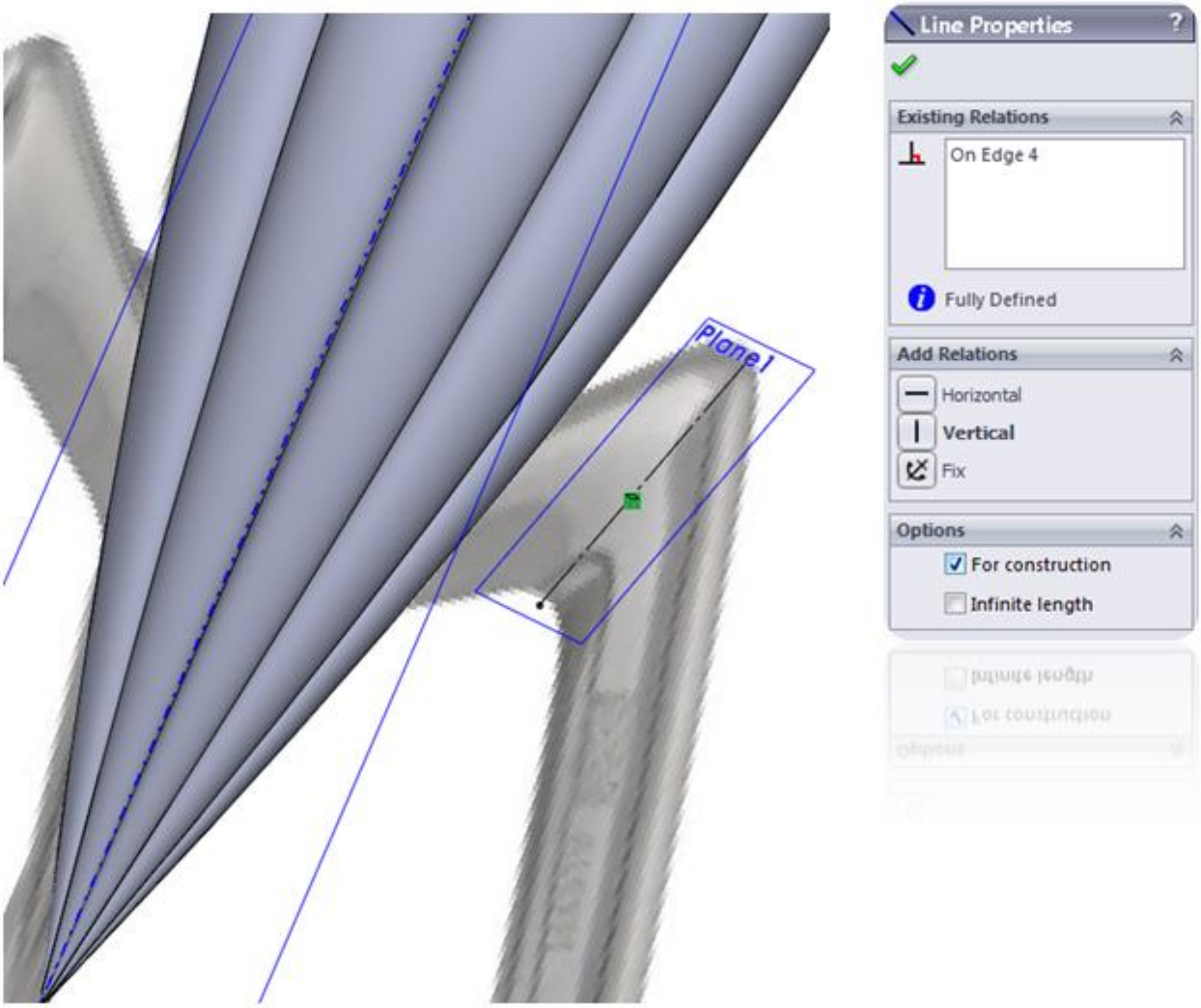
Click at the grey centerline of Sketch3

Go to: **Tools > Sketch Tools > Convert Entities** or click at the Convert Entities icon 

Click OK 

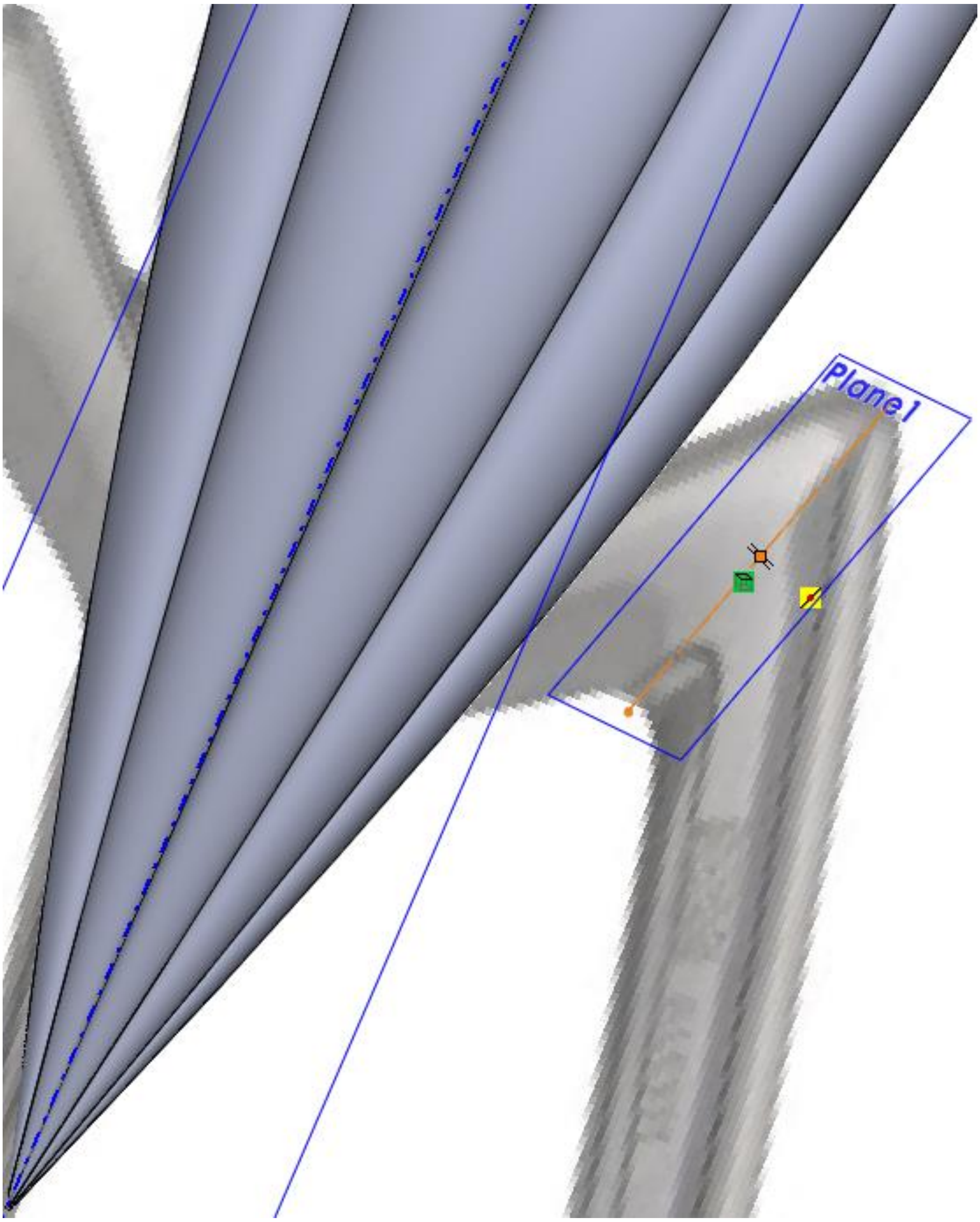
Change the solid line into a centerline

Click at the new line > Select the **For Construction** option in the menu at the left side

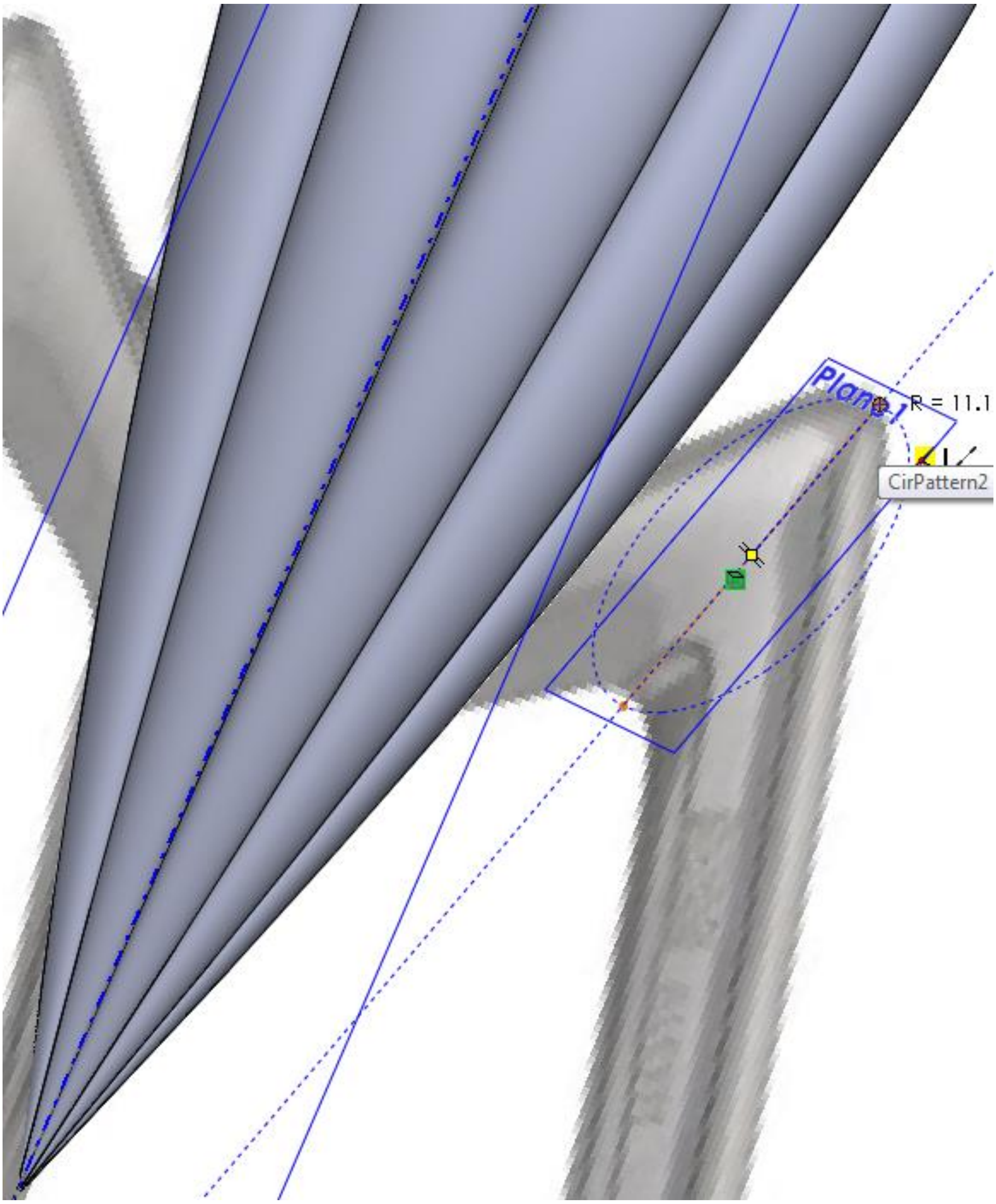


Create an Ellipse

Go to: **Tools > Sketch Entities > Ellipse** or click at the Ellipse icon 
Start the Ellipse at the midpoint of the centerline



Click at one of the ends of the centerline to set the height of the Ellipse

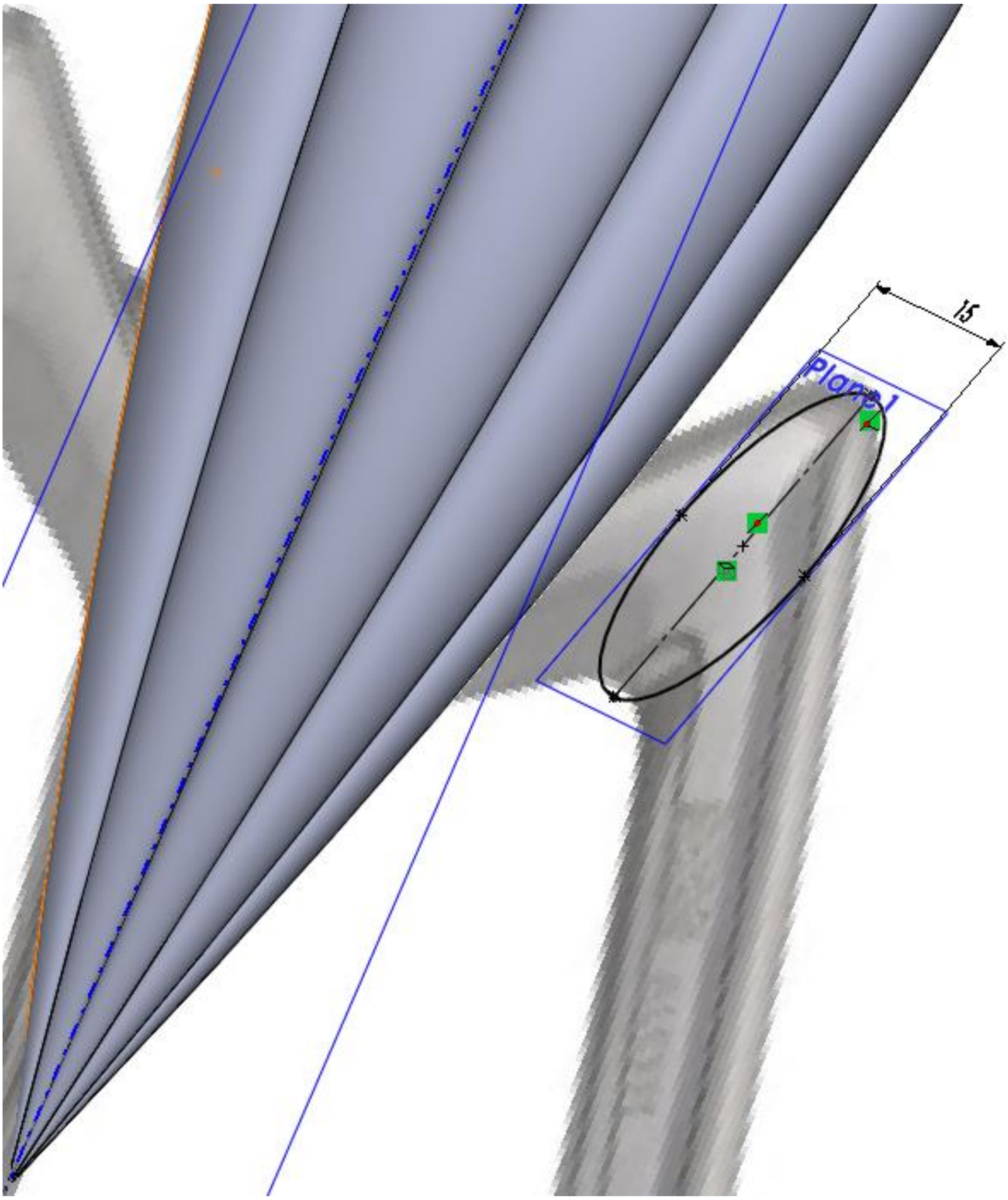


Click somewhere next to the construction line to set the width of the Ellipse

Change the width by clicking at the dimension button

The Ellipse is fully defined

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



Create another 2D sketch

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

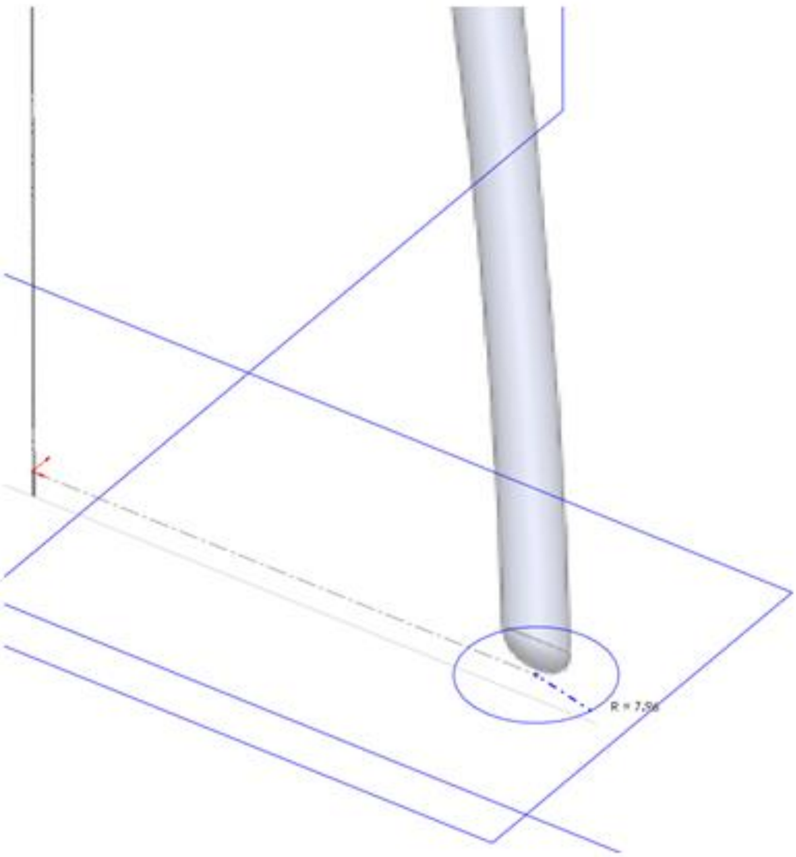
Create a circle at the end of the construction line of Sketch2

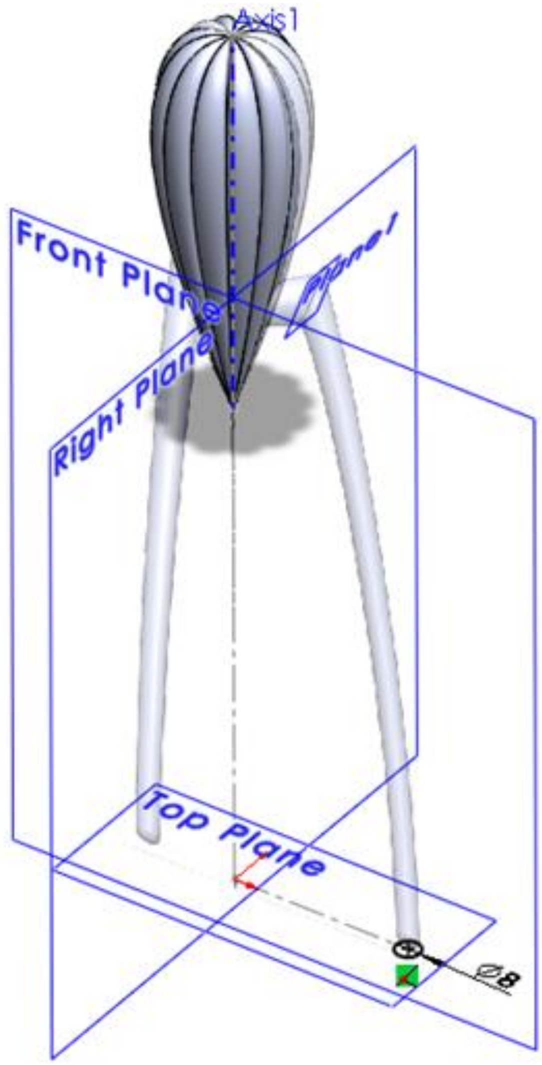
Go to: **Tools > Sketch Entities > Circle** or click at the Circle icon

Draw a circle witch starts at the endpoint of the construction line of Sketch3

Change the dimension of the circle into 8 mm as shown in the picture

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



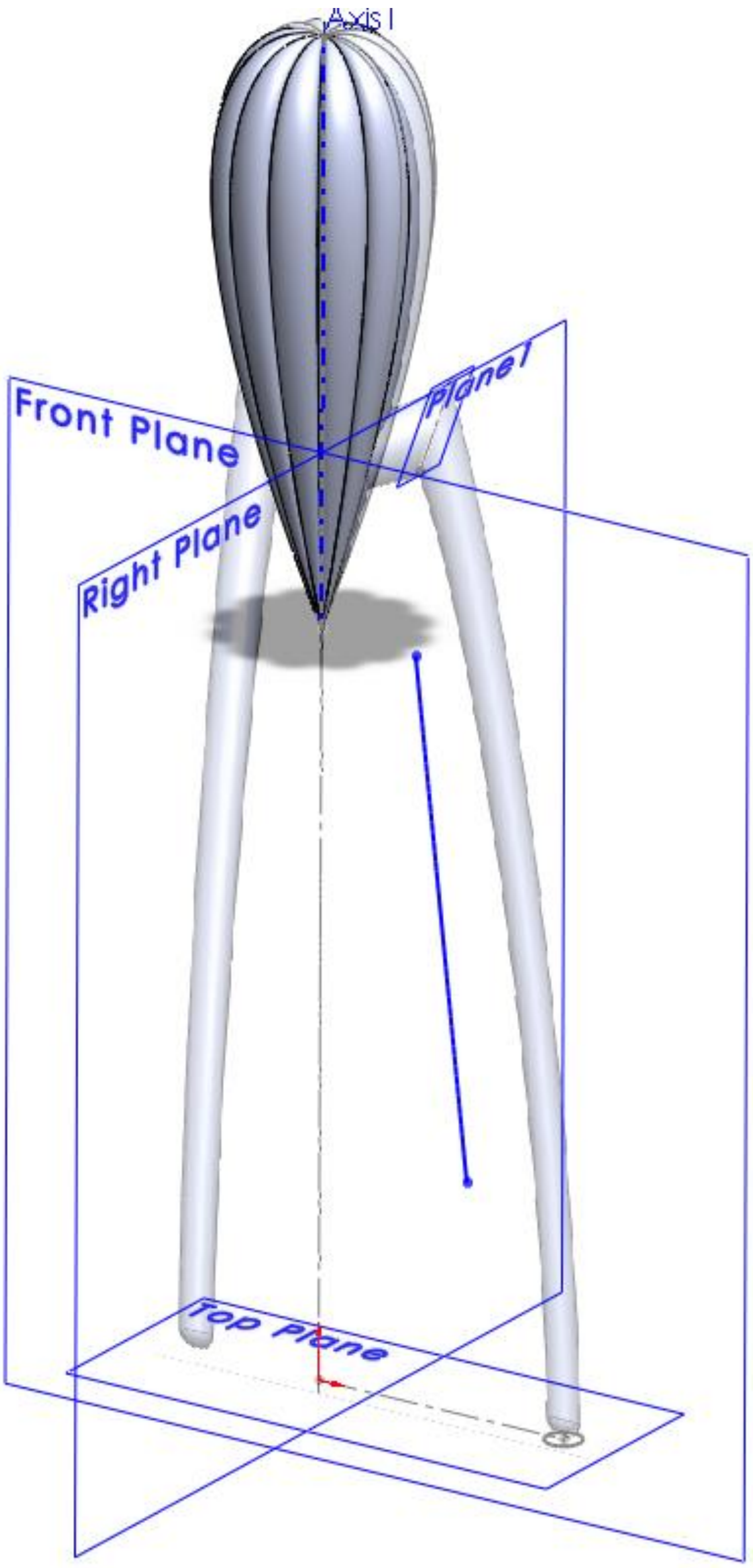


Create another 2D sketch

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

Draw a spline without midpoints

Go to: **Tools > Sketch Entities > Spline** or click at the Spline icon



Connect the Spline with Sketch4 and Sketch5

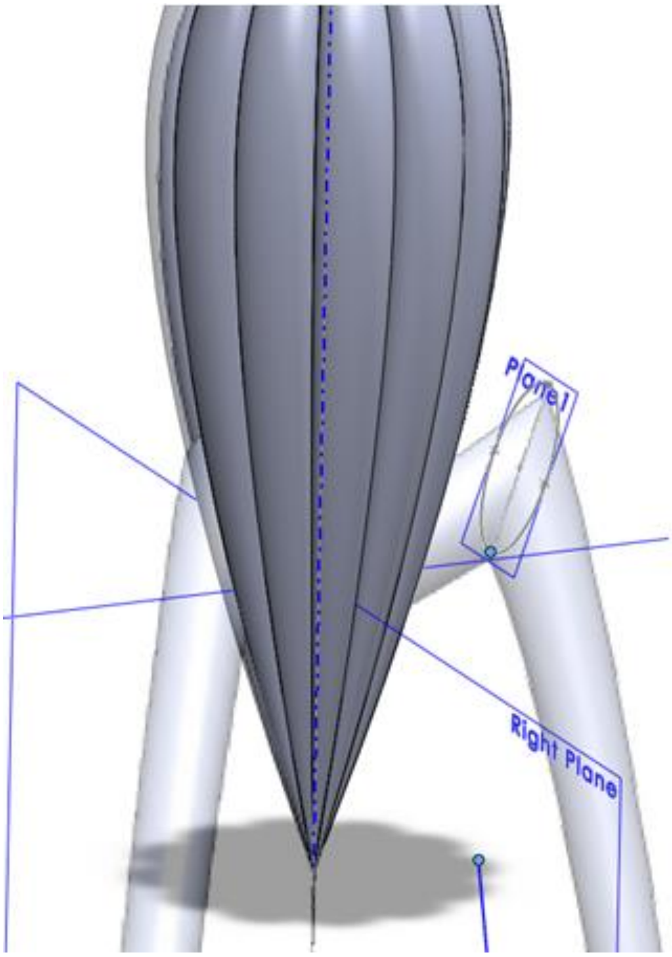
Select the upper endpoint of the Spline

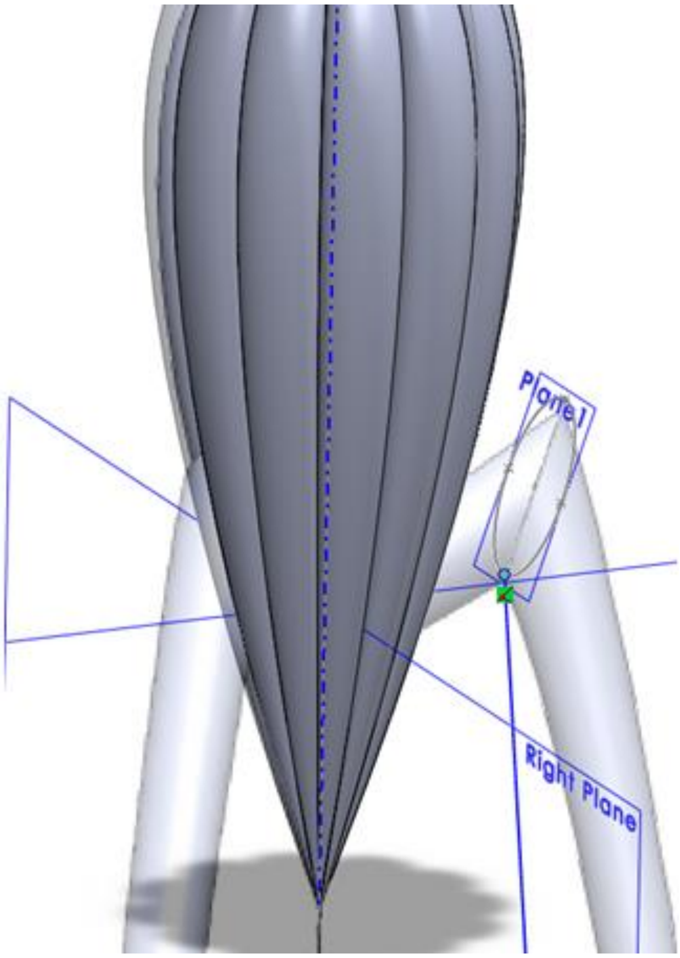
Hold down the Control key on your keyboard

Click at the lower endpoint of the Ellipse in Sketch4 as shown in the picture

Click at the **Coincident** icon in the **Add Relations** box

Click OK





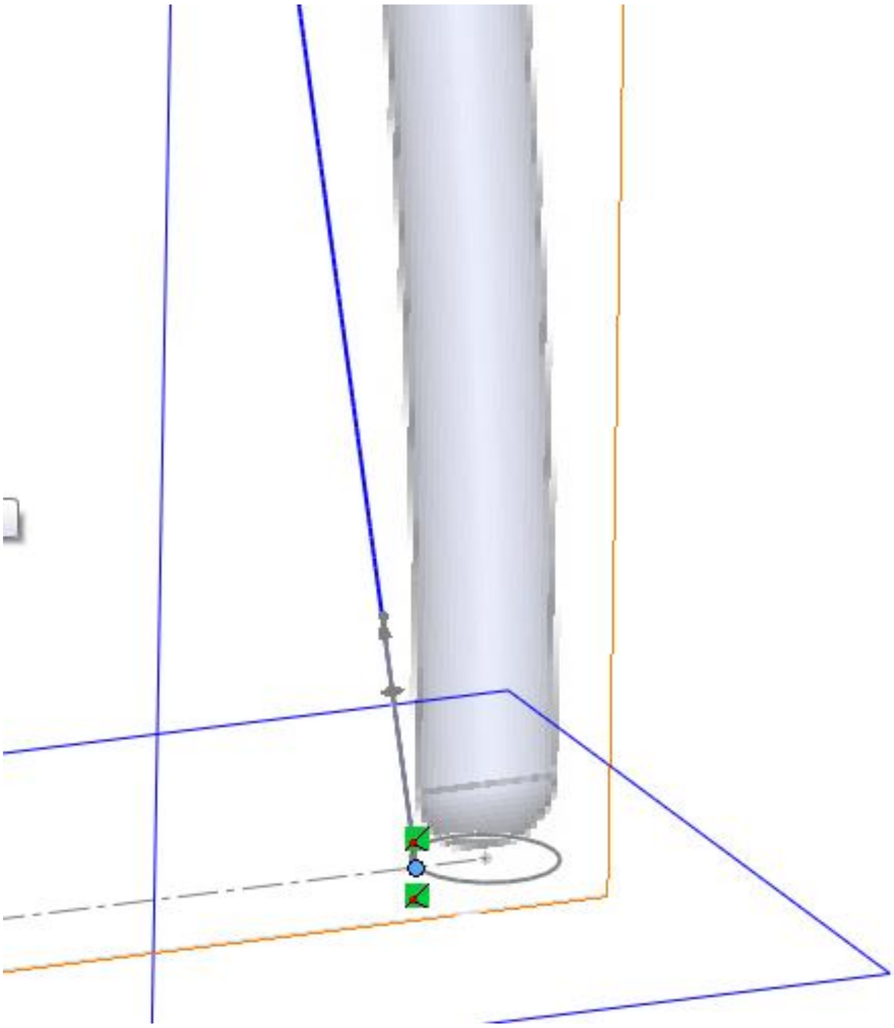
Select the lower endpoint of the Spline

Hold down the left key of your mouse and drag the spline to the circle of Sketch5.

SolidWorks will automatically add a **Coincident** relation with the circle.
Make sure that the Spline is connected with the edge of the circle instead of the center as shown in the picture

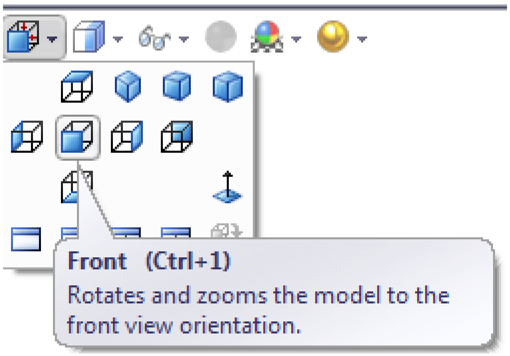
The spline is now connected with two different Sketches

Click OK



Change viewport to Front Plane

Click at Control+1 or click at the View Orientation box and click at Front Plane



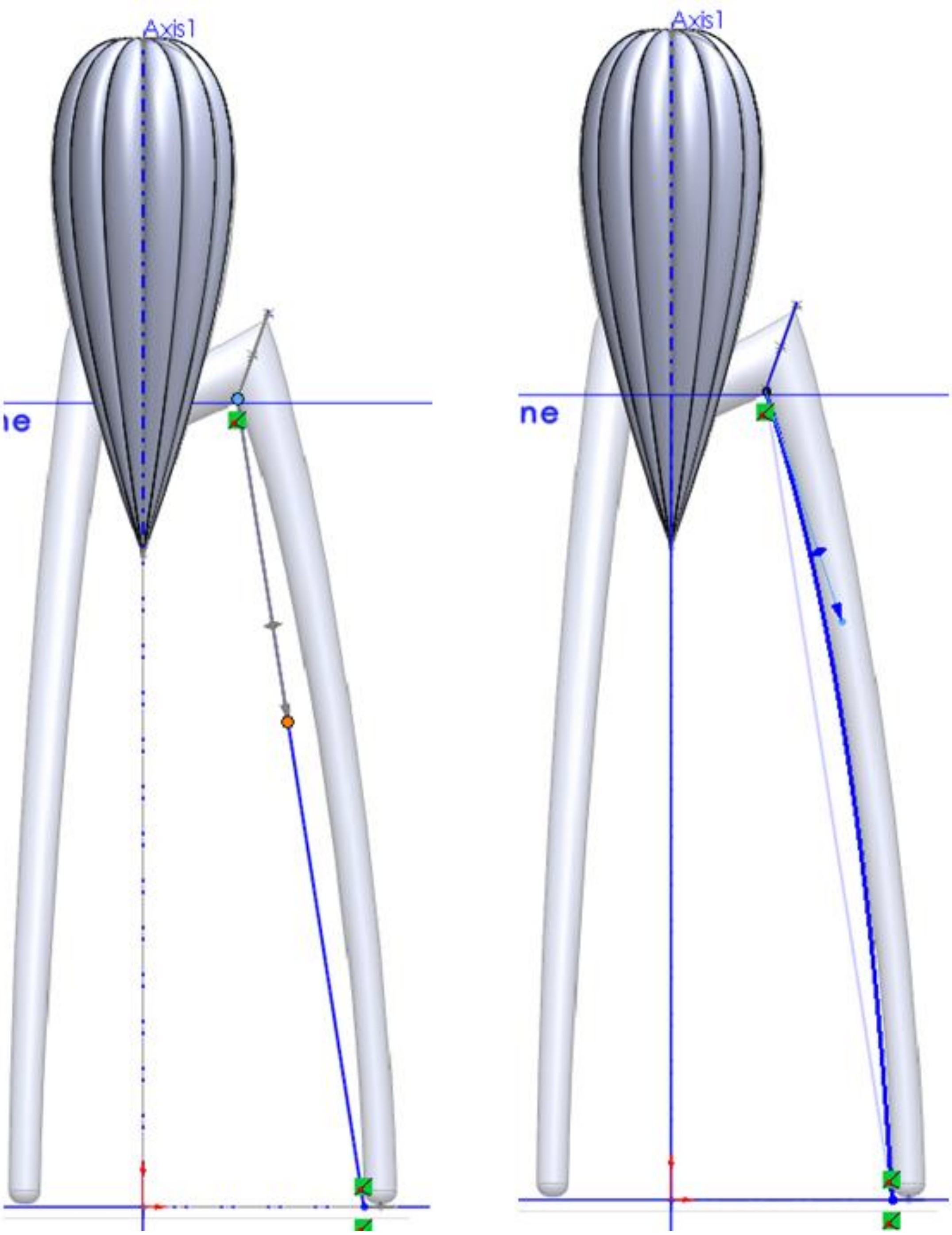
Change the curve of the Spline like the blueprint

Click at the Top point of the spline > the grey arrows of the Spline appear

Click and drag the round endpoint of the grey arrow as shown in the picture **(the orange dot)**

Try to create the curve of the blueprint as shown in the picture

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



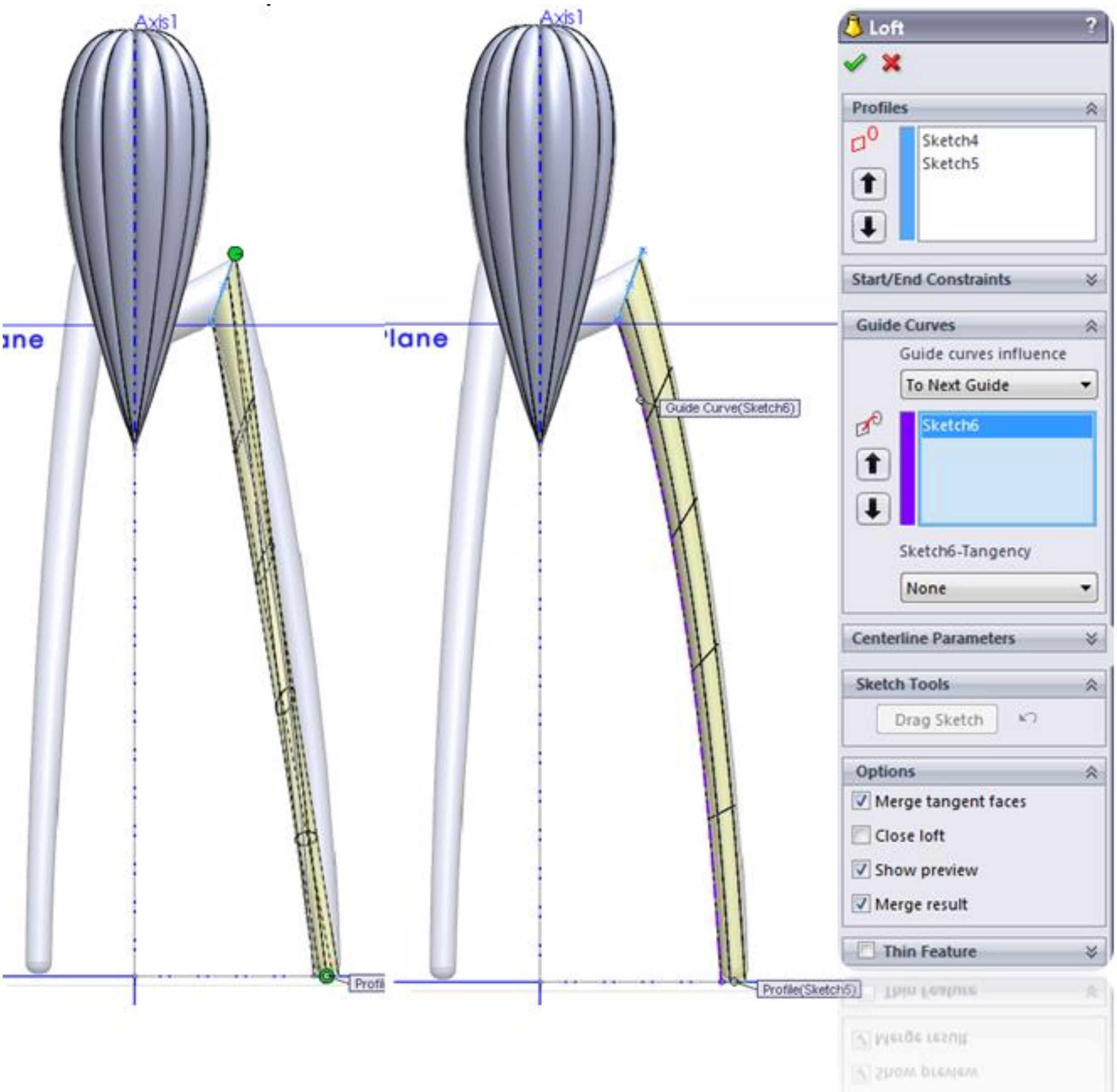
Create a Solid Loft

Go to: **Insert > Boss/Base > Loft** or click at the Loft icon
Click in the Profiles box
Select Sketch4 and Sketch5 as shown in the picture

NOTE: The easiest way to select all the profiles and curves for this loft is to select them in the feature tree. This avoids errors and saves time.
Click in the Guide Curves box
Select Sketch6 as shown in the picture

Guide curves influence: To Next Guide

Click OK



Create another 2D sketch

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

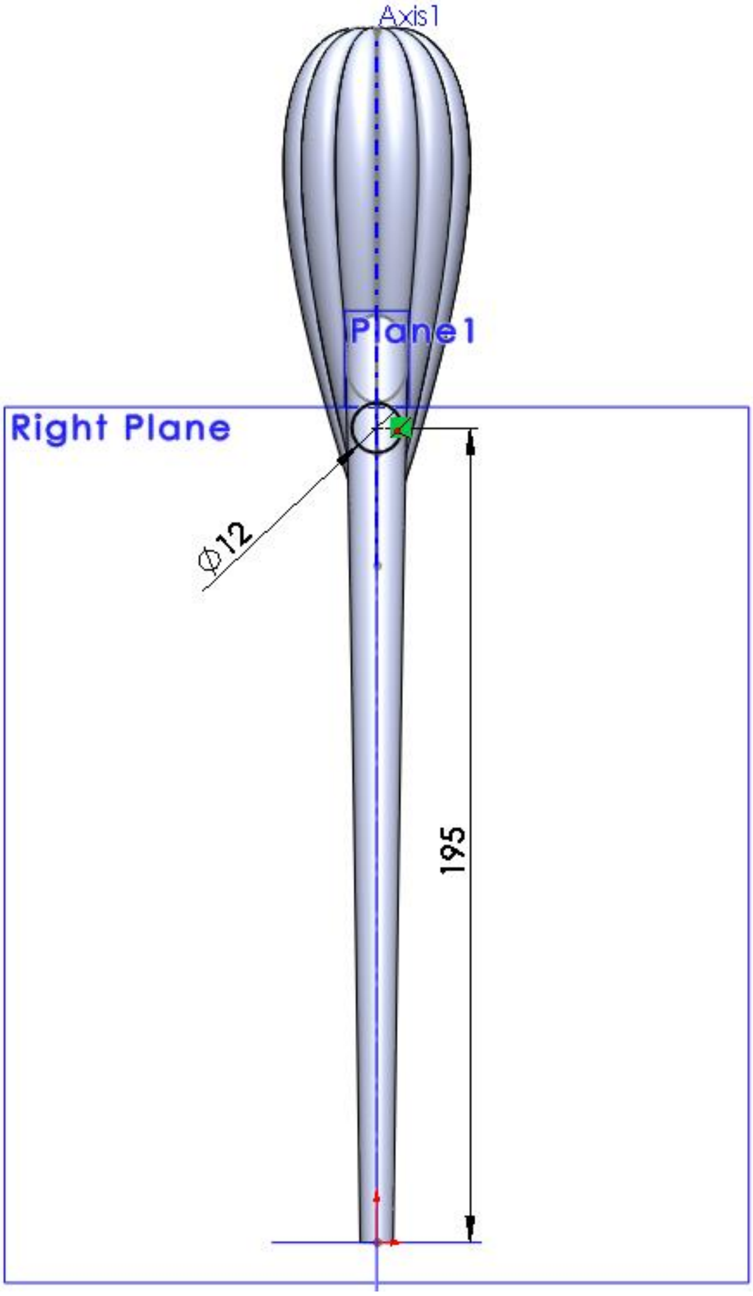
Create a circle

Go to: **Tools > Sketch Entities > Circle** or click at the Circle icon
Draw a circle witch starts at the Right Plane as shown in the picture

Change the dimension of the circle into 12 mm as shown in the picture

Change the height of the circle into 195 mm starting at the origin

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



Create a Solid Loft

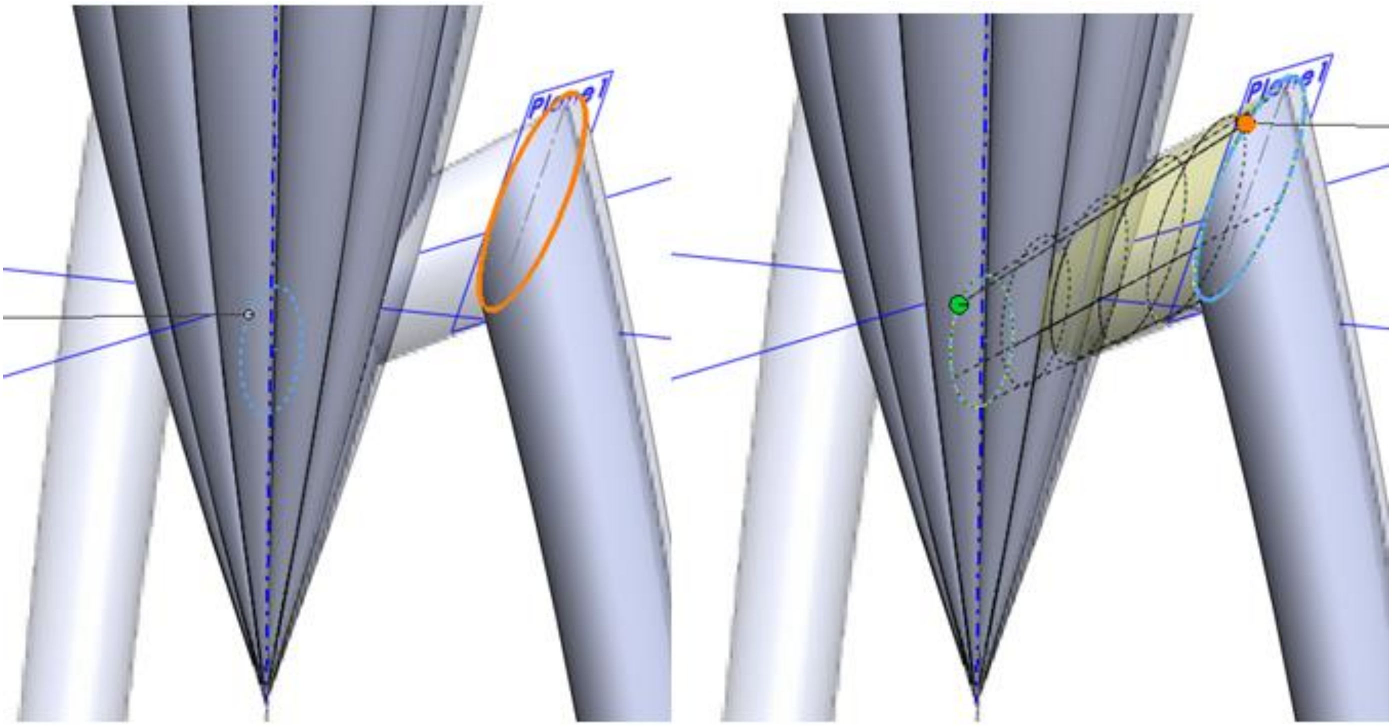
Go to: **Insert > Boss/Base > Loft** or click at the Loft icon

Click in the Profiles box

Select the new Sketch7

Select the outer edge of Loft1 as shown in the picture

Click OK



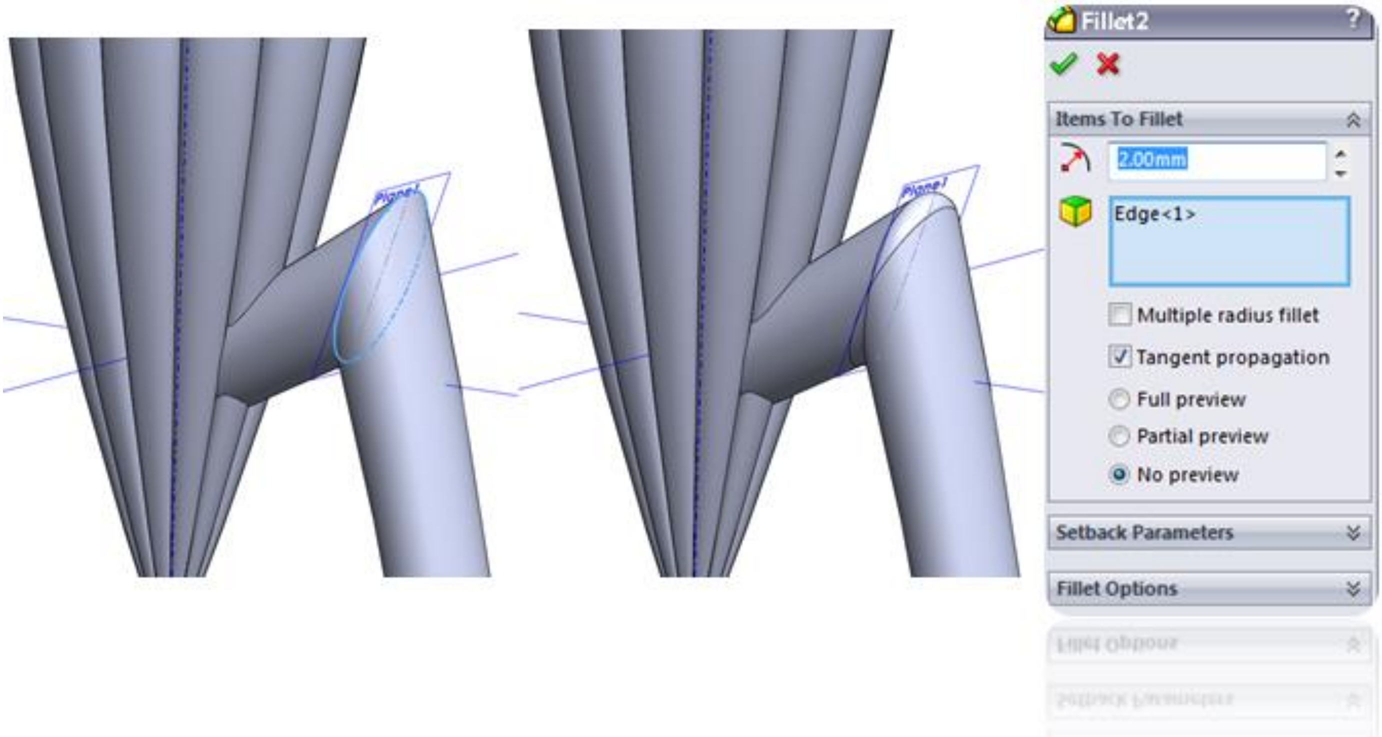
Create a Fillet

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

Click at the blue edge as shown in the picture

Change the Radius into 2 mm

Click OK

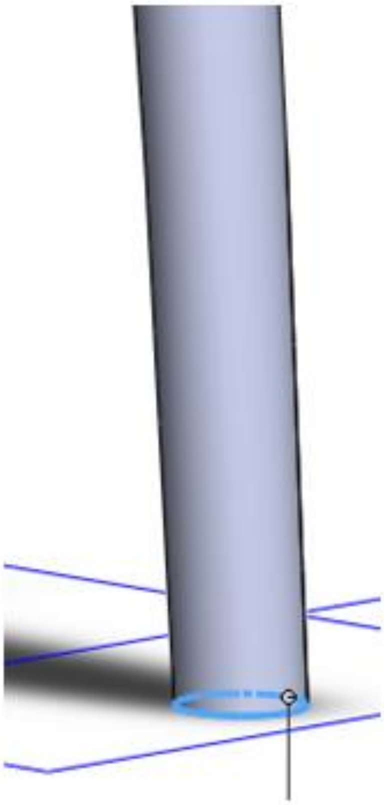


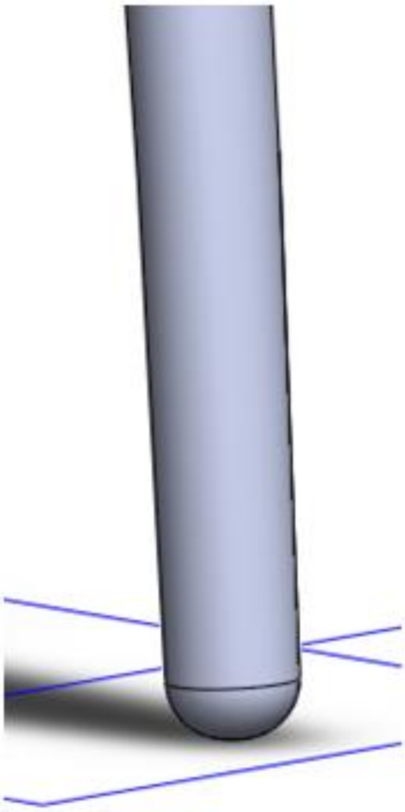
Create a Fillet

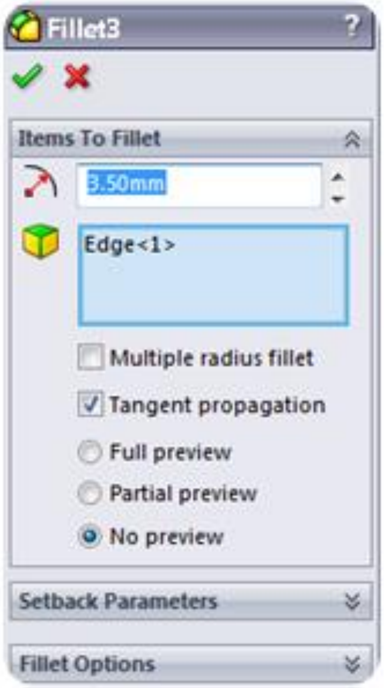
Go to: **Insert > Features > Fillet/Round** or click at the Fillet icon 
Click at the blue edge as shown in the picture

Change the Radius into 2 mm 

Click OK 







Create a Circular Pattern

Go to **Insert > Pattern/Mirror > Circular Pattern** or click at the Circular Pattern icon

Click in the **''Parameters''** box at the white Pattern-Axis box

Select Axis1

Change the Total Angle into 360 degrees

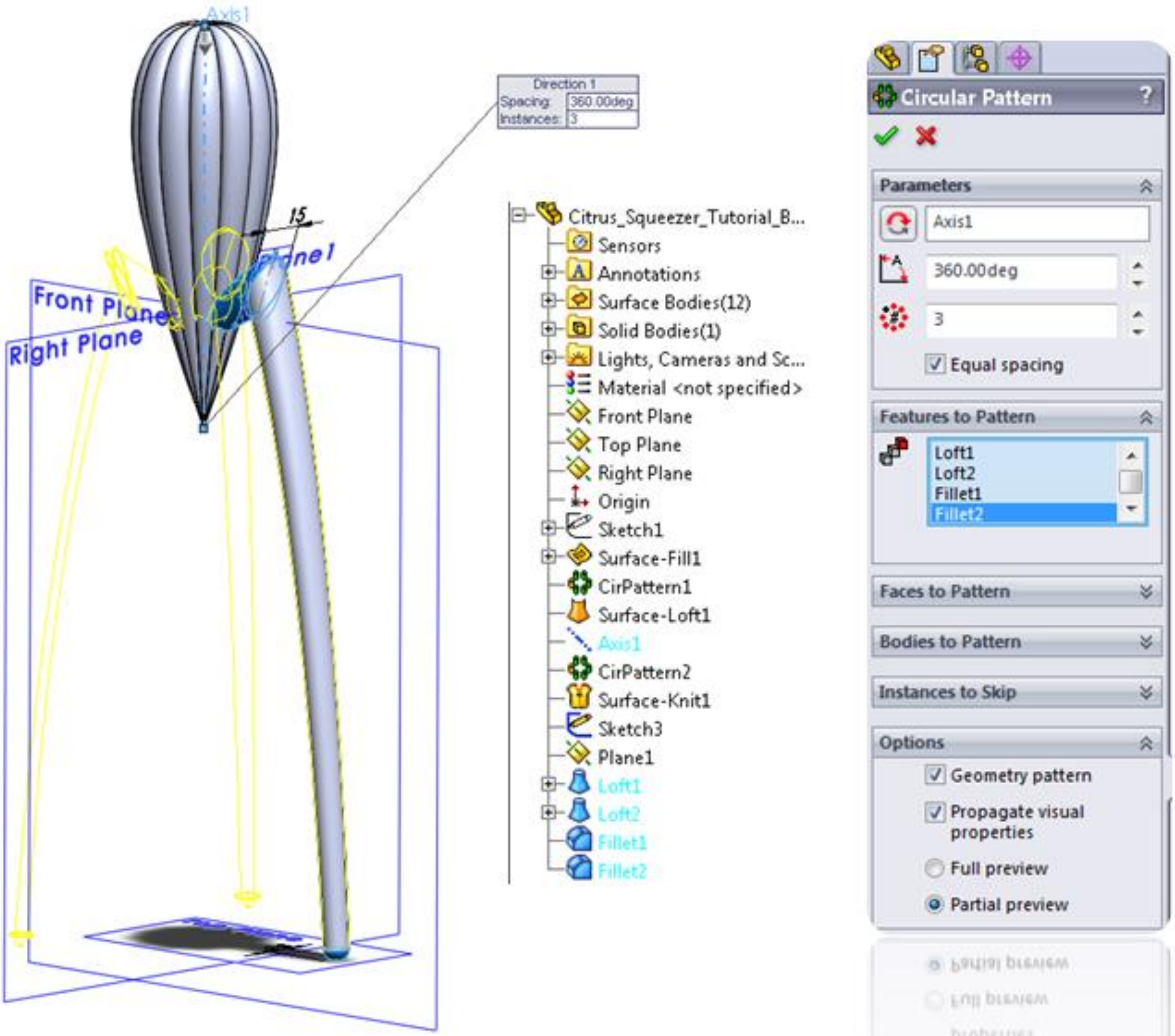
Change the Number of Instances into 3

Select the ''Equal Spacing'' option

Click at the **''Features to Pattern''** box

Select Loft1, Loft2, Fillet1, and Fillet2 in the feature tree as shown in the picture

Click OK



Congratulations, you just finished your own Citrus Squeezer! ☺

Did you like this free eBook and are you eager to learn more? Then you should really attend my free SolidWorks workshop.

In this workshop, you are going to discover how to become a SolidWorks Pro in days instead of years without boring practice, expensive training classes, or any pointless theory:

[Click here to attend my free SolidWorks workshop.]

In this free SolidWorks workshop you will discover:

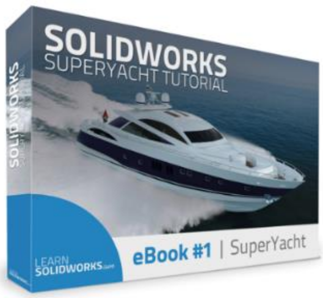
✓ The 4 secret ways to learn SolidWorks and **why you actually need to know what they are** and which is best for you **to be successful**.

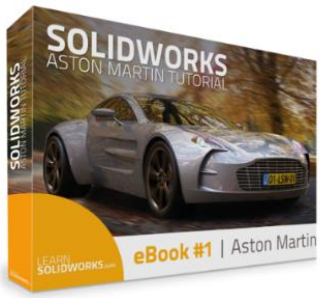
✓ The **simple tricks** that get people with no SolidWorks experience, no design or engineering experience, think they're too old, and procrastinate too much to actually learn SolidWorks.

✓ The 6 secret career hacks to become highly successful as a designer or engineer **to climb the career ladder much faster**.

✓ How I modeled a REAL $37.000.000 SuperYacht in SolidWorks for a famous American entrepreneur and **how you can get similar design projects as well**.

✓ **Free access** to the first eBooks and videos of the SolidWorks Chopper, Yacht, Aston Martin & Boeing 747 course!





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The entire workshop will take around 90 minutes so make sure to reserve enough time.

So take a cup of coffee, grab a pen and paper and enjoy! ☕

Best wishes,

Jan-Willem Zuyderduyn