LEARN**SW**

HOW TO MODEL A
CITRUS SQUEEZER
IN SOLIDWORKS?

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CITRUS
SQUEEZER
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''A step by step SolidWorks Tutorial''

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 design of Starck range from a luxury private yacht to mass produced consumer products such as chairs, dinnerwair 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.

In this tutorial I will show you the following features:

- Draw 2D sketches
- Insert a blueprint
- Create a new Plane
- Create a new Axis
- Surface Fill
- Surface Knit
- Solidify surfaces
- Circular Pattern
- Surface Loft
- Solid Loft
- Fillet

Hope you like it,

Jan

Render of Starck's Citrus Squeezer rendered in PhotoView360

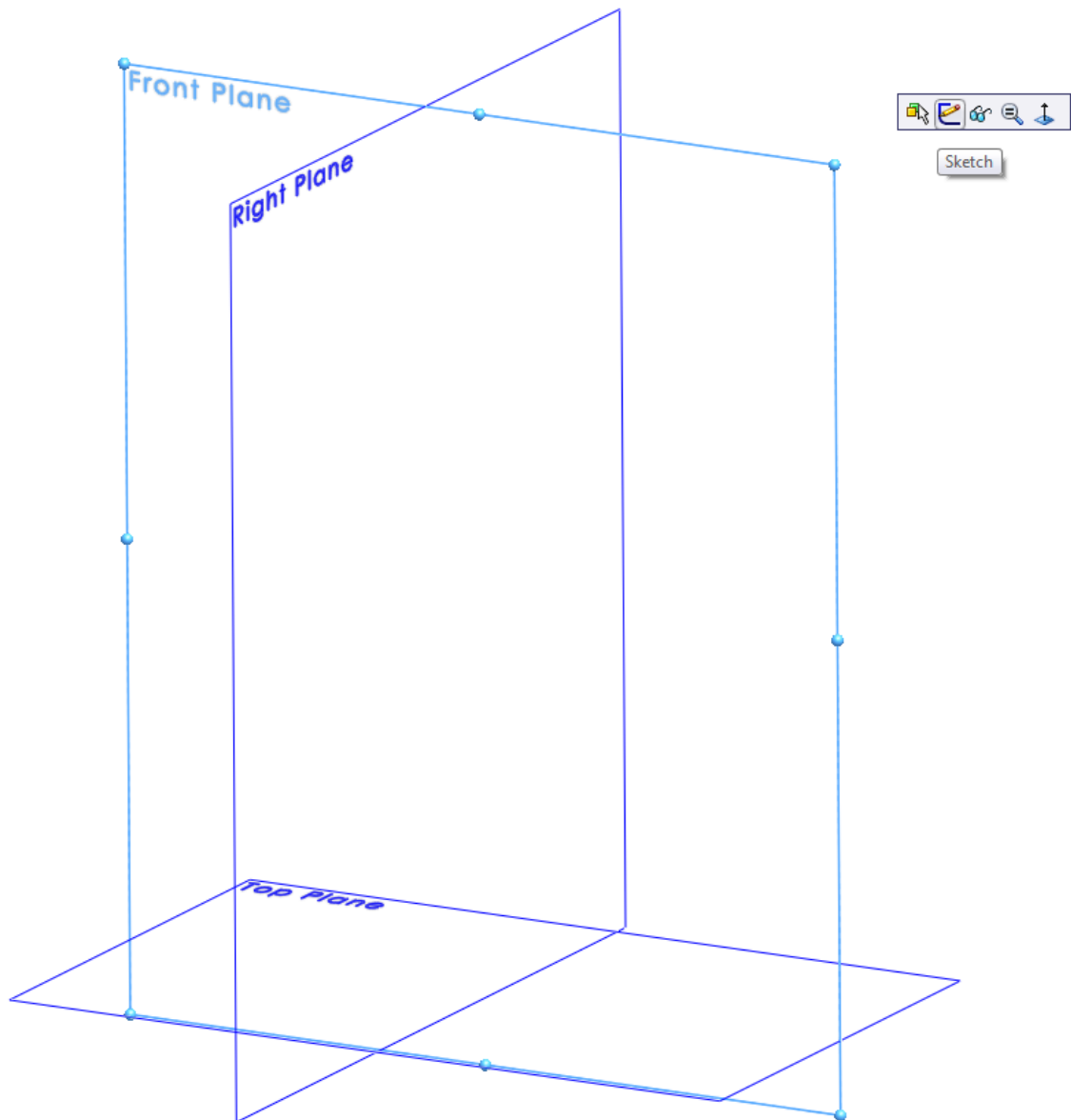**Open a new part with model units set to millimeters**

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
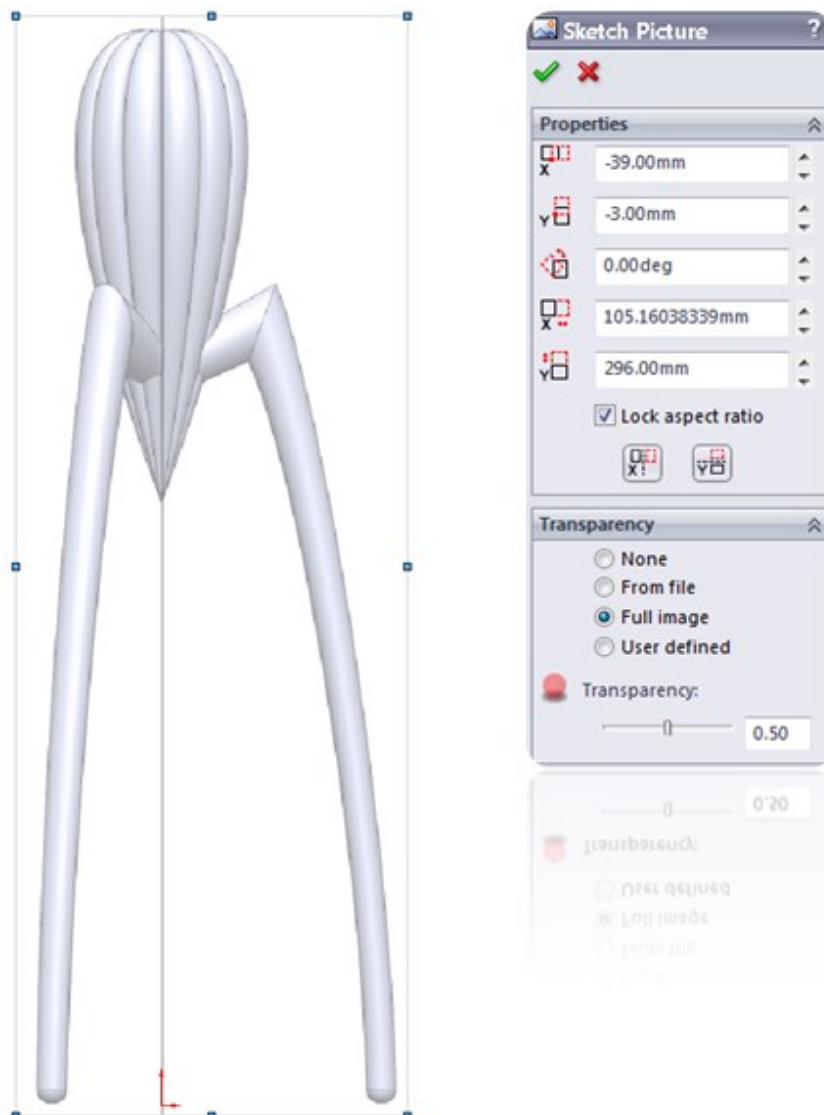
Create another 2D sketch

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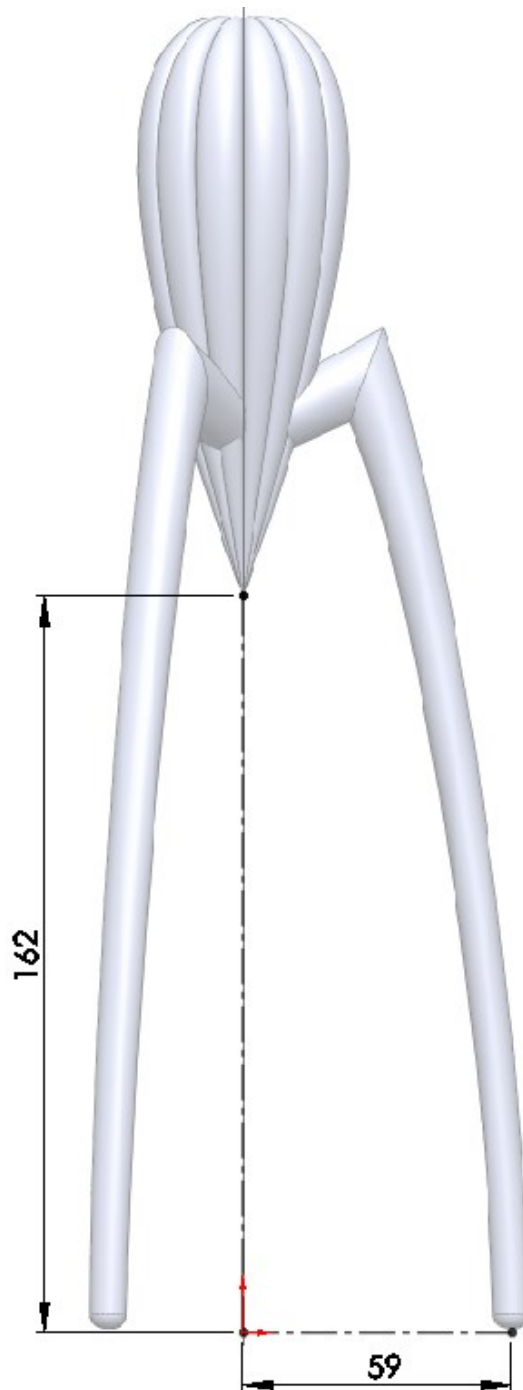
Draw two centerlines

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 🔷

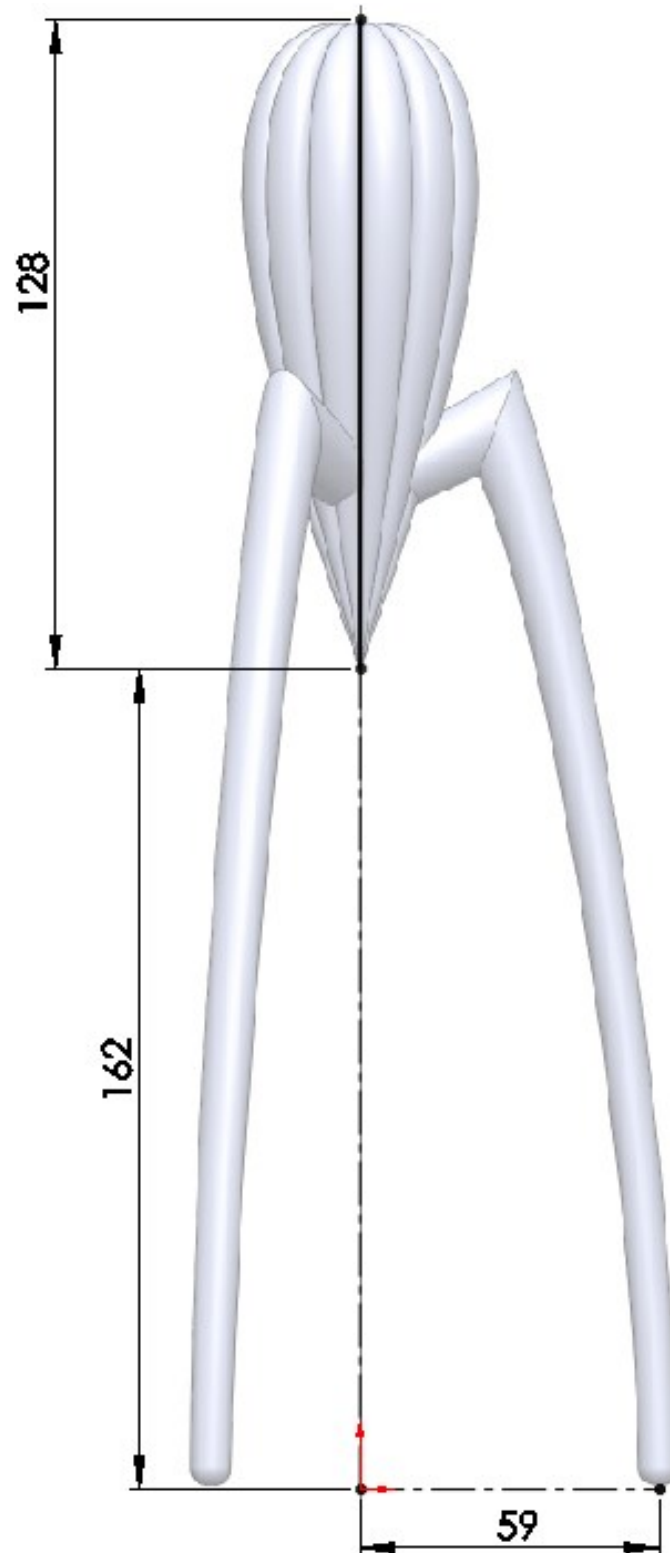Draw a horizontal centerline that starts at the origin. ⌐

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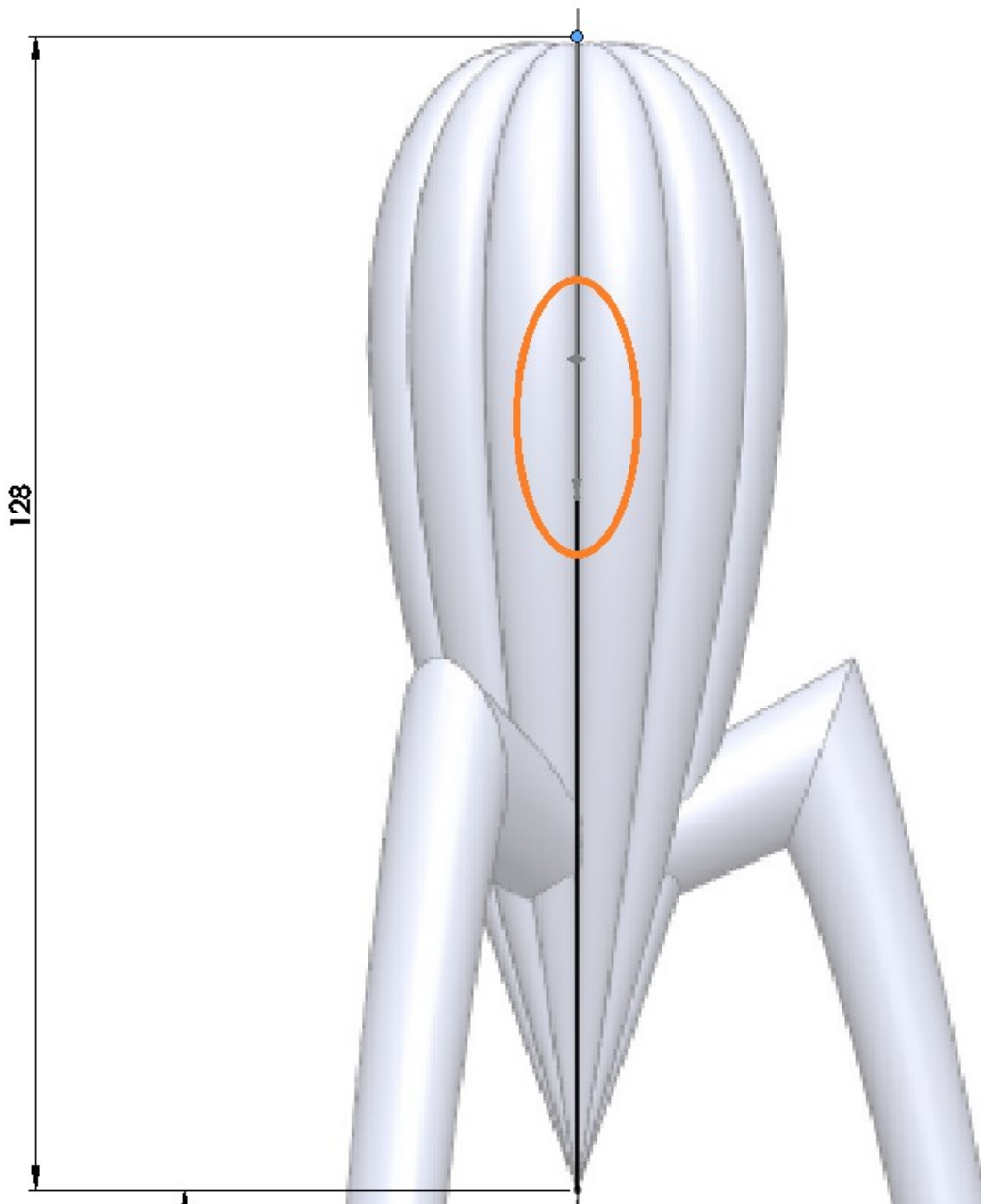
Click and drag the round endpoint of the grey arrow as shown in the picture (the orange dot)

128
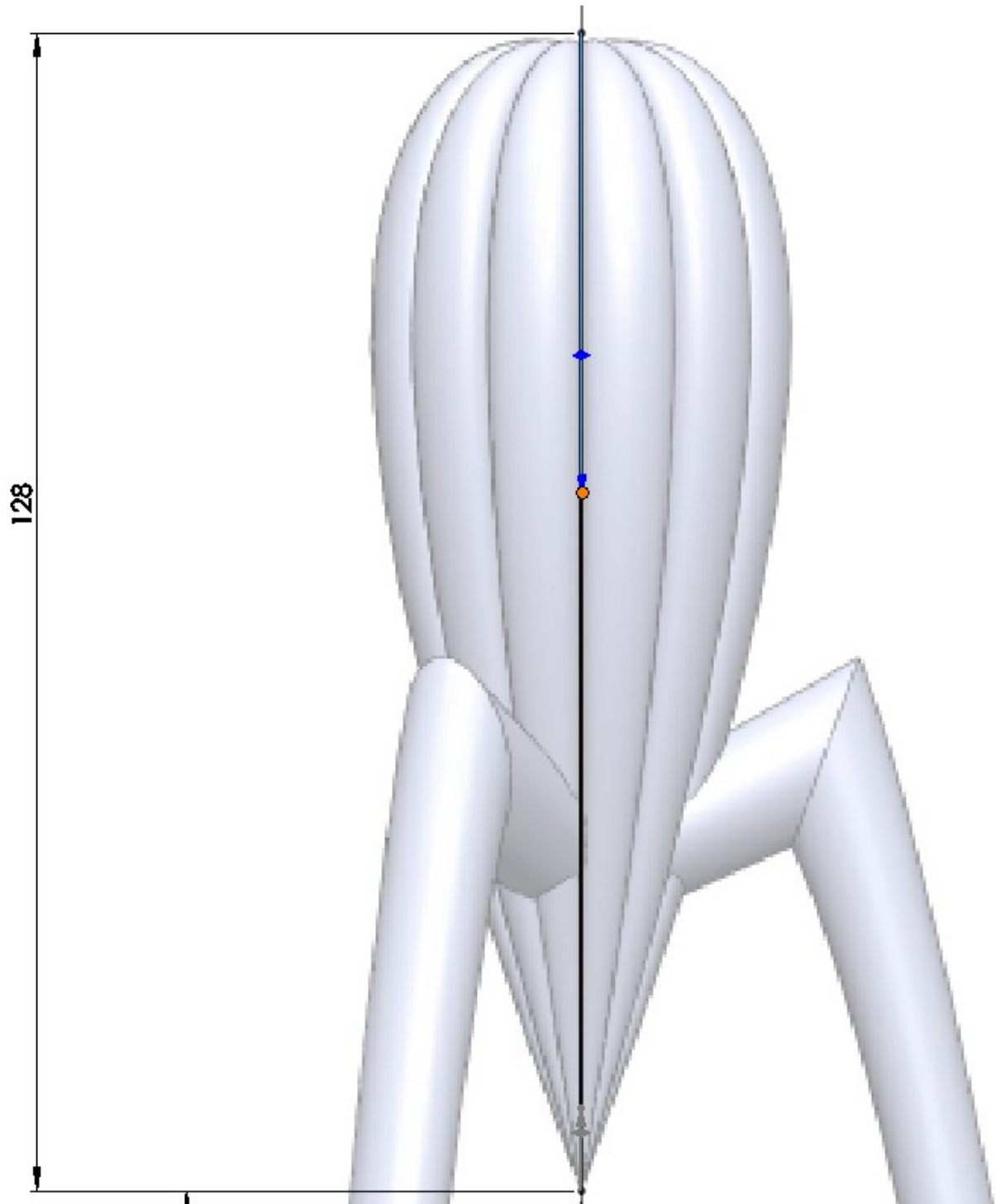
Try to create the inner curve of the blueprint as shown in the picture
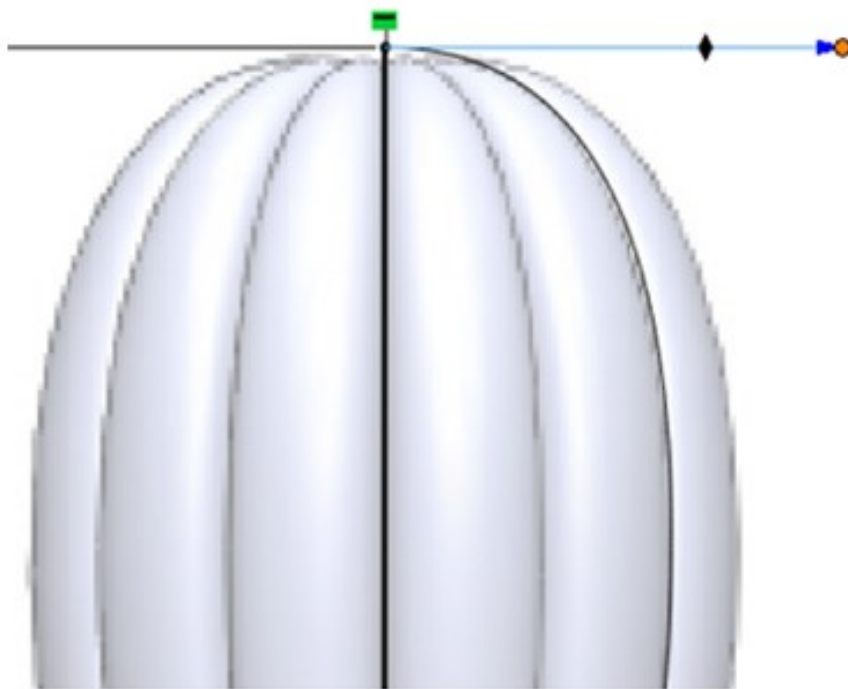

128
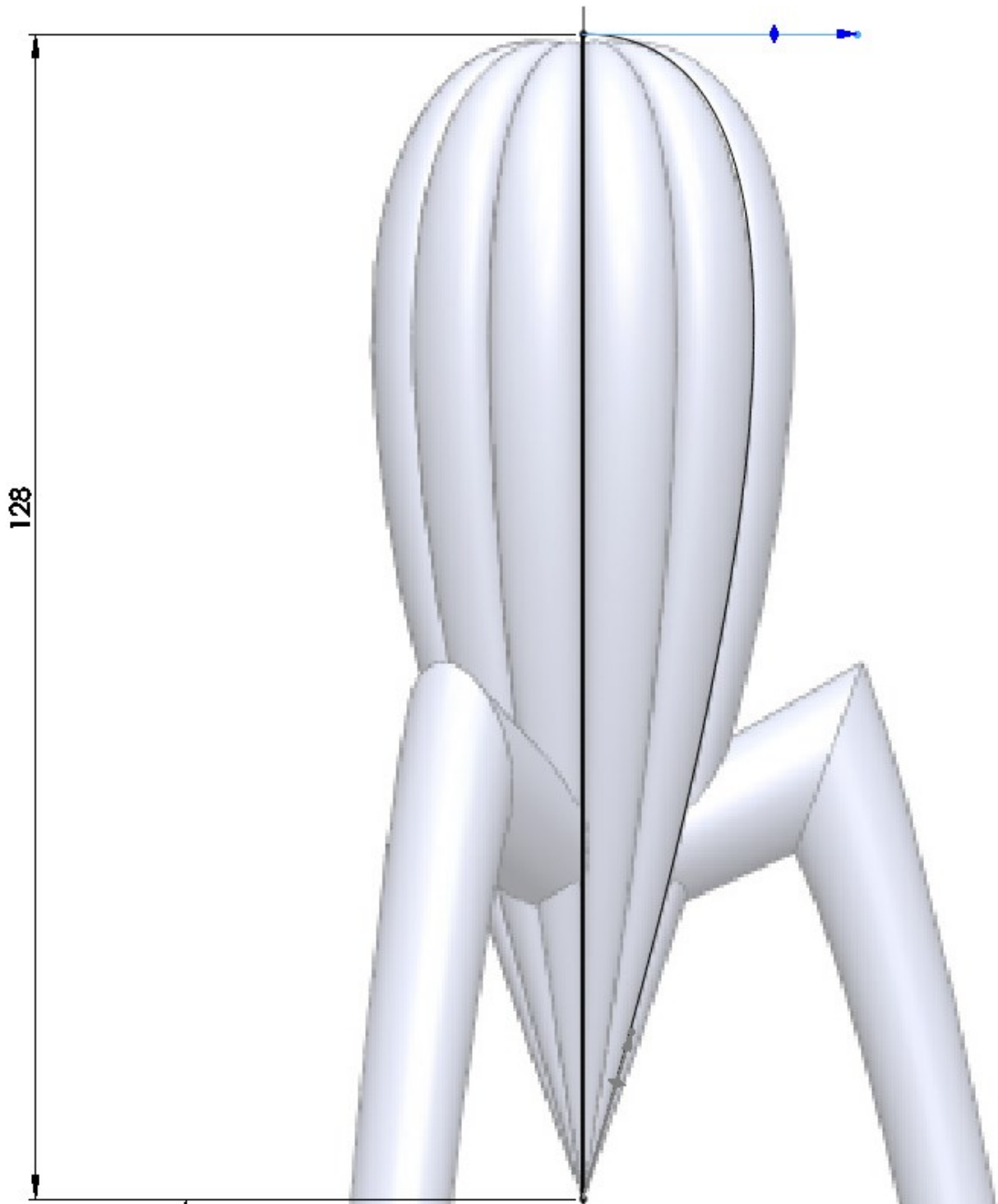
Add a tangency relation add the end of the spline

Click at the orange dot as shown in the picture

Select the Horizontal relation in the Spline menu bar at the left side ⊟
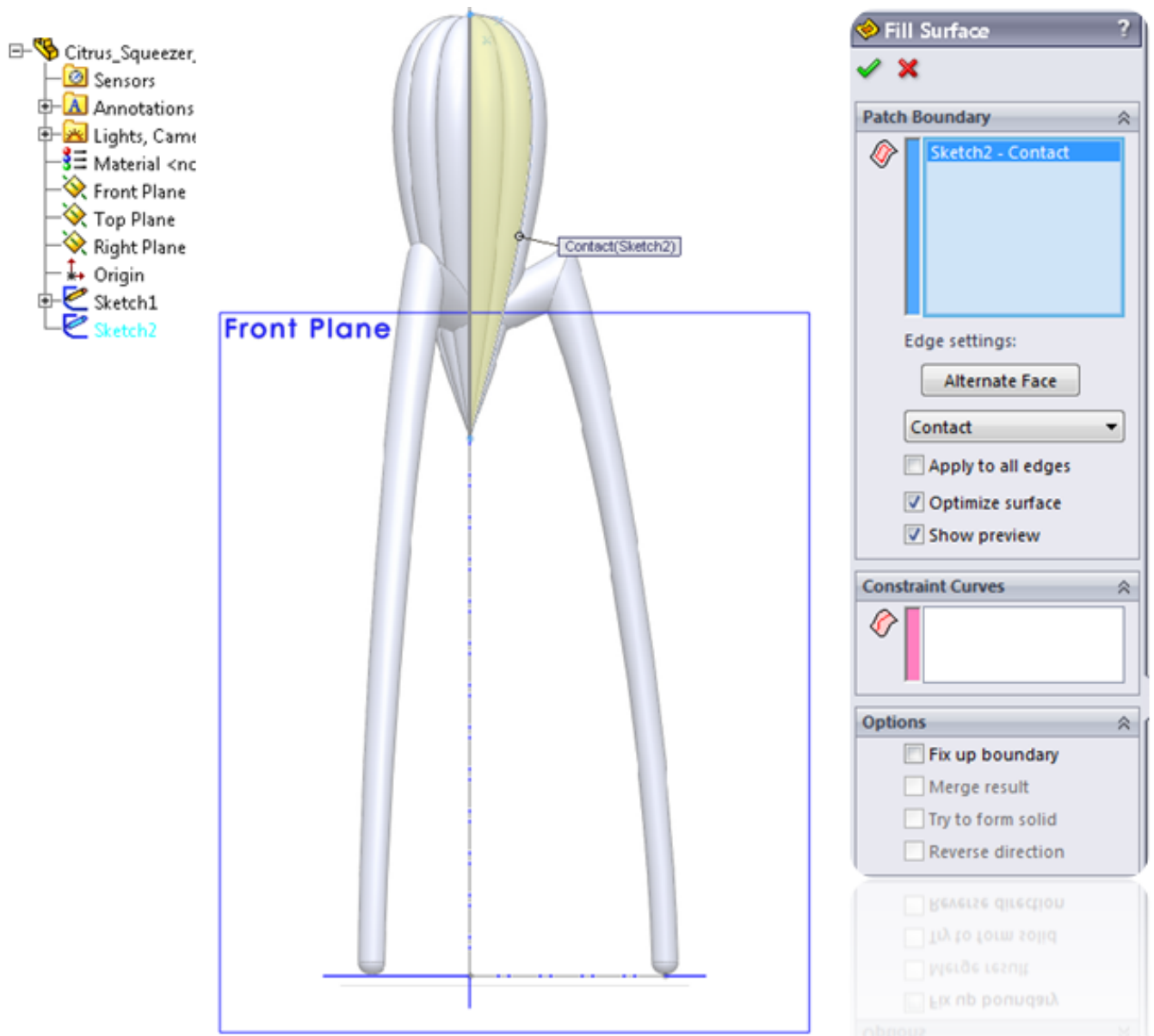
The endpoint of the spline is fully tangent now
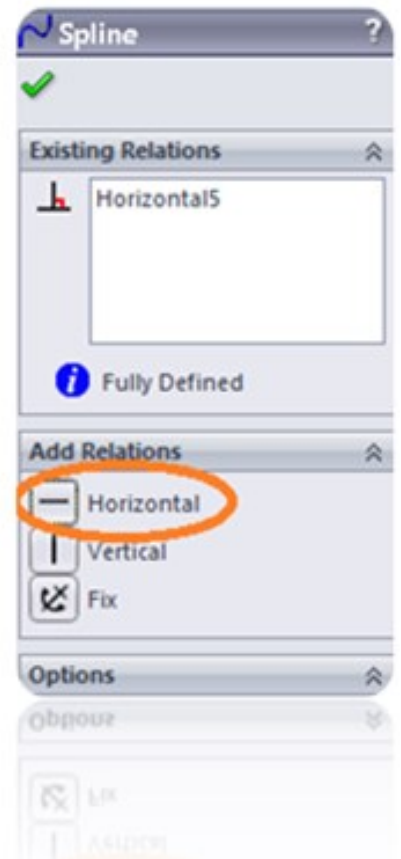
Click at the Sketch button in the upper right corner close the 2D Sketch

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
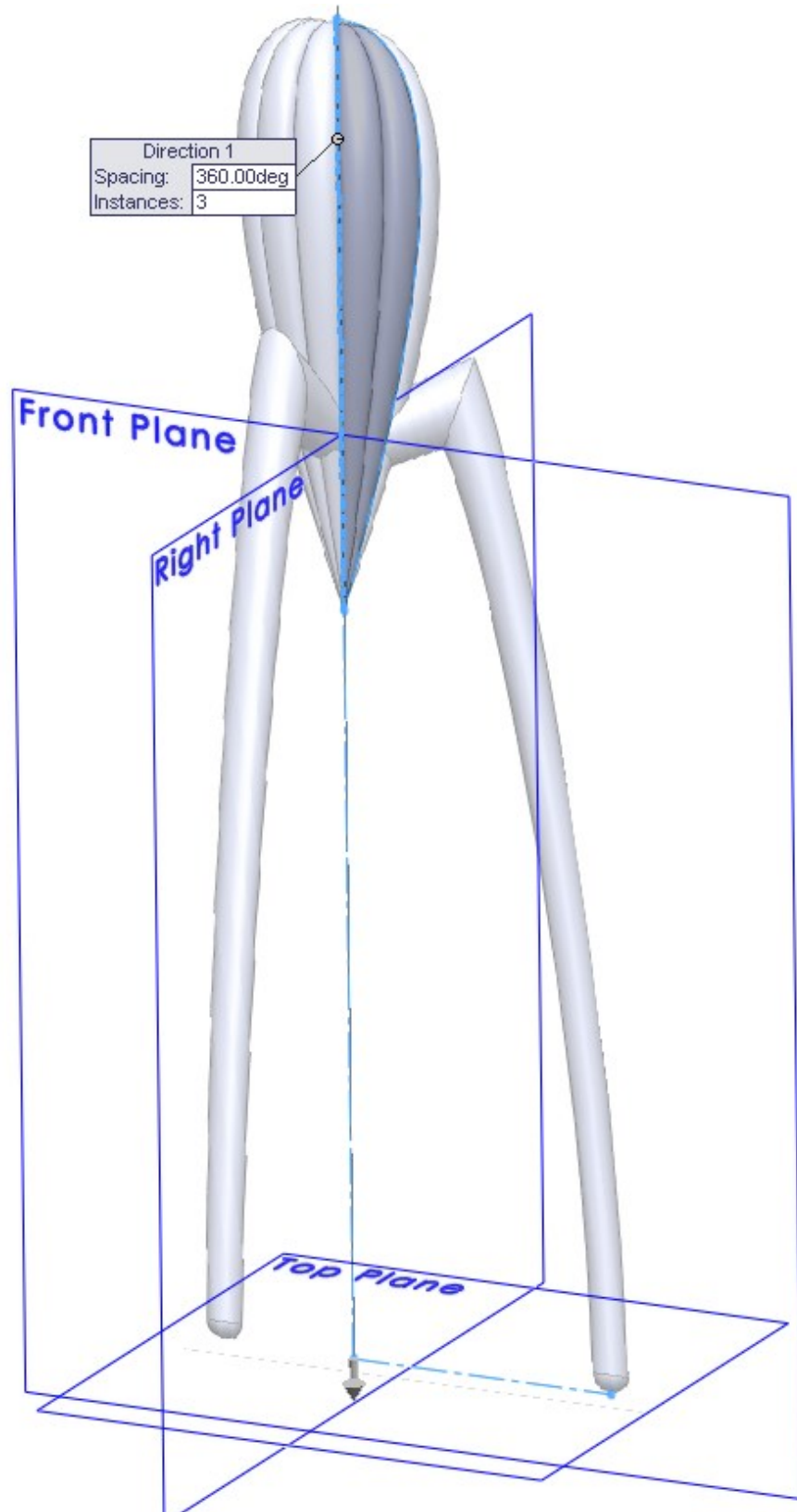
Click OK

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
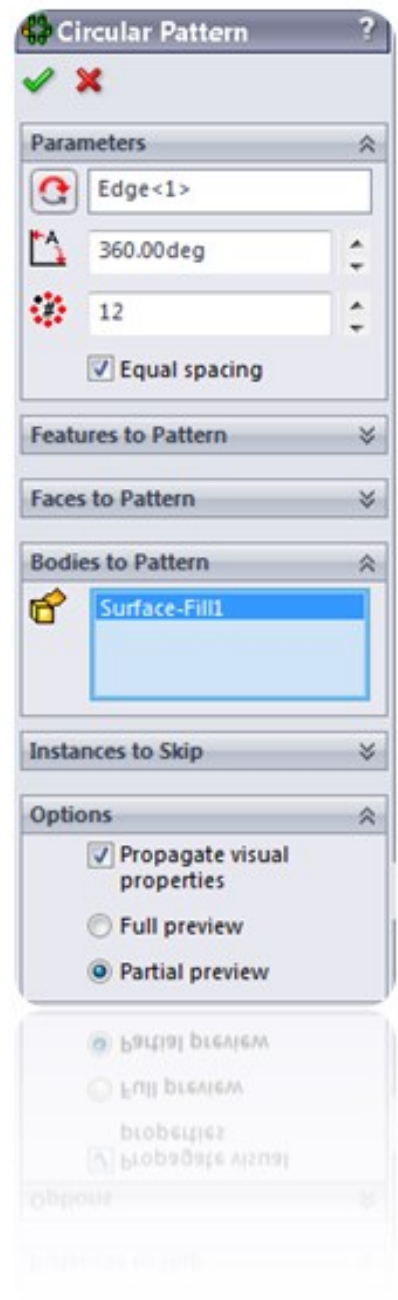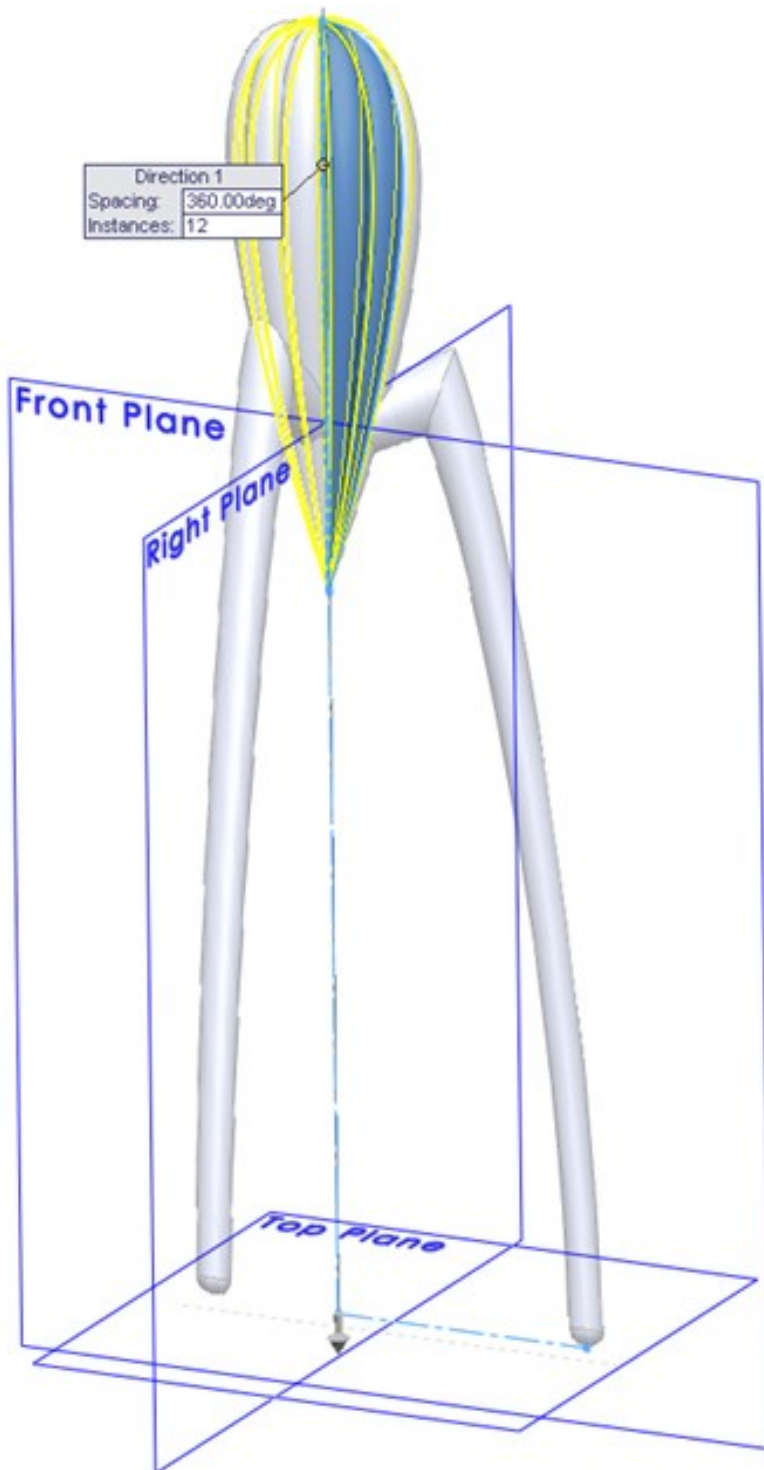
Change the Total Angle into 360 degrees

Change the Number of Instances into 12

Select the ''Equal Spacing'' option

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

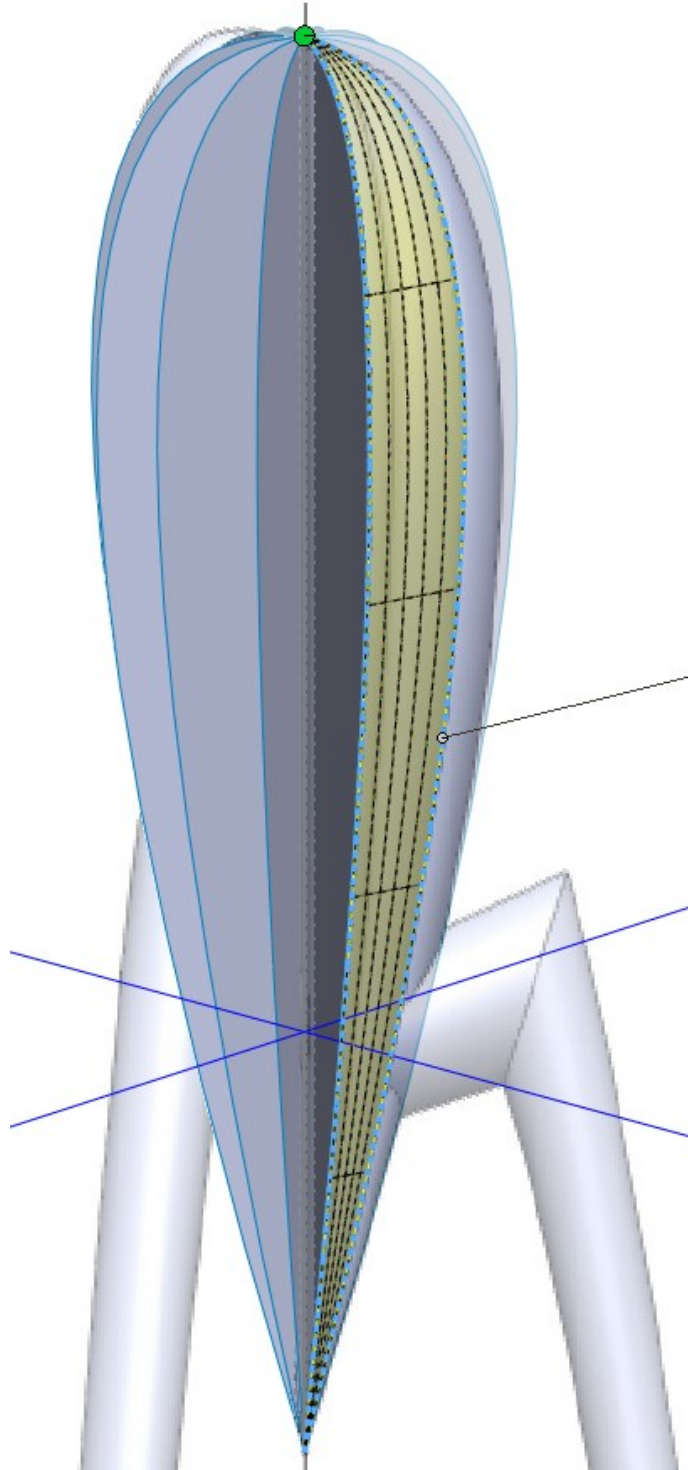Select Surface-Fill1 as ''Surface to Pattern''

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 box

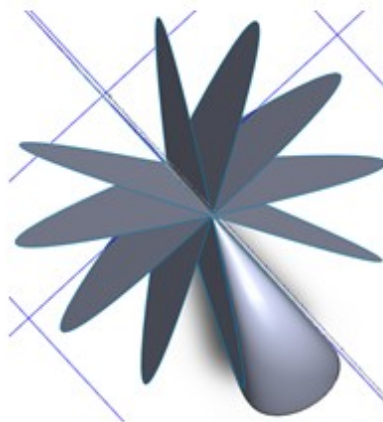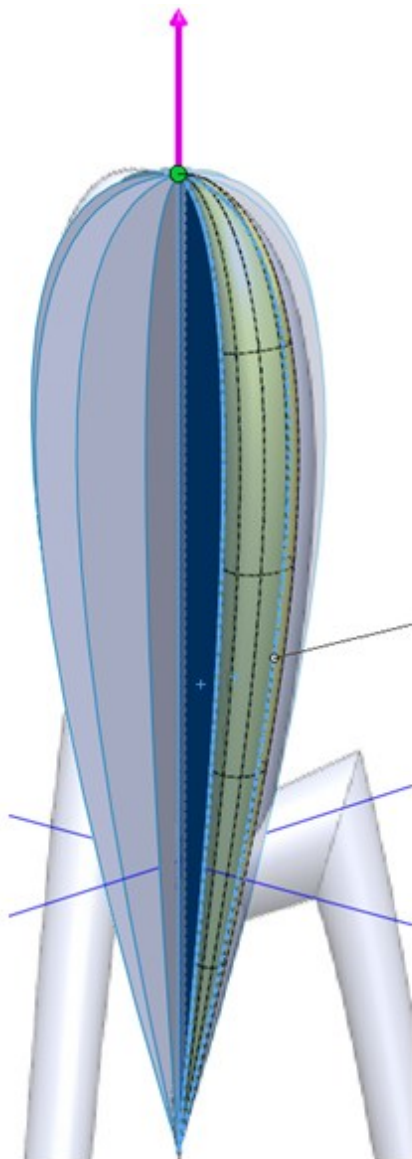Change the **''End Constraint''** into Tangency To Face

Change the ''End Tangent Length'' into 4

Select the Apply to all box

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
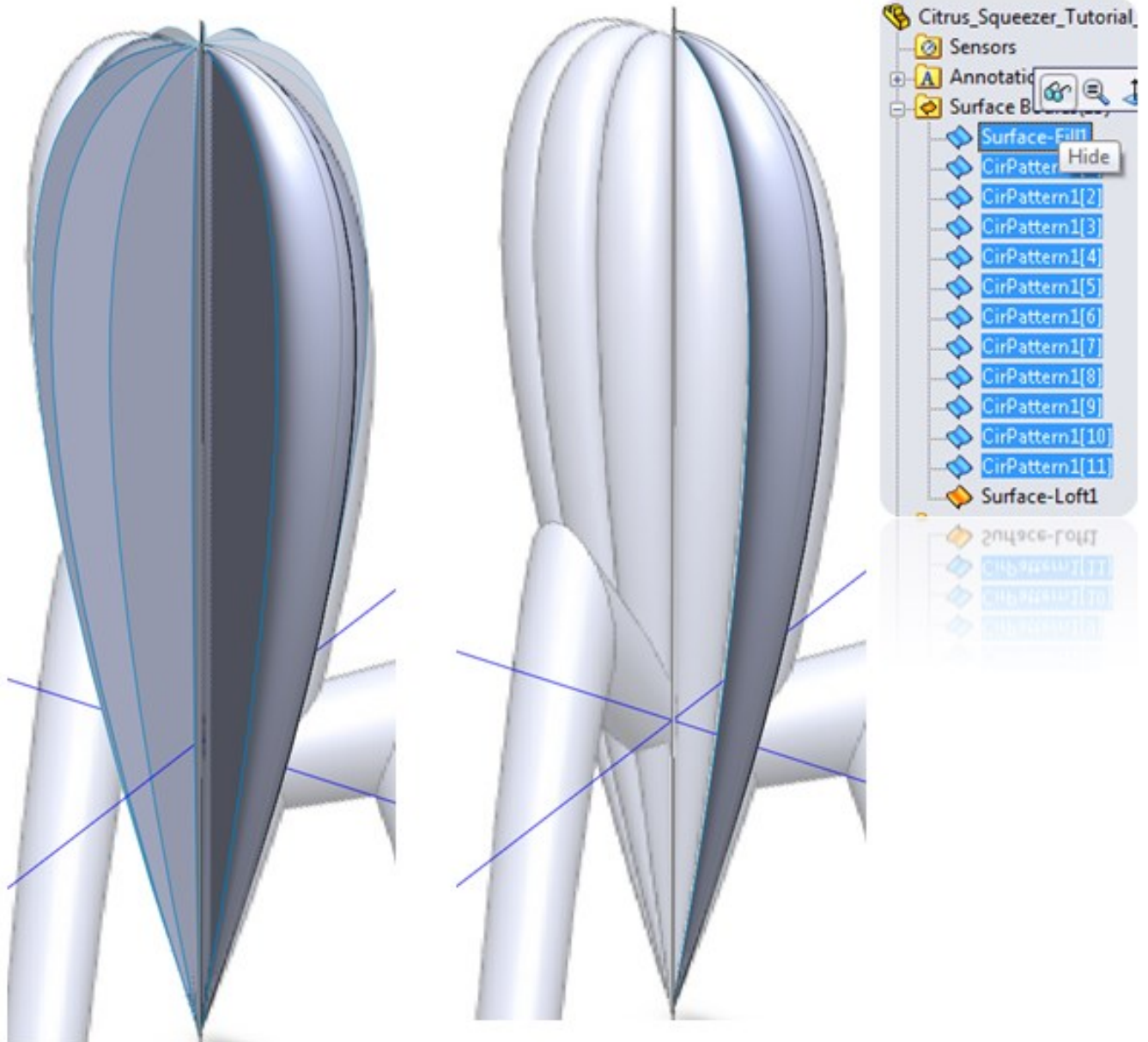
Click OK

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

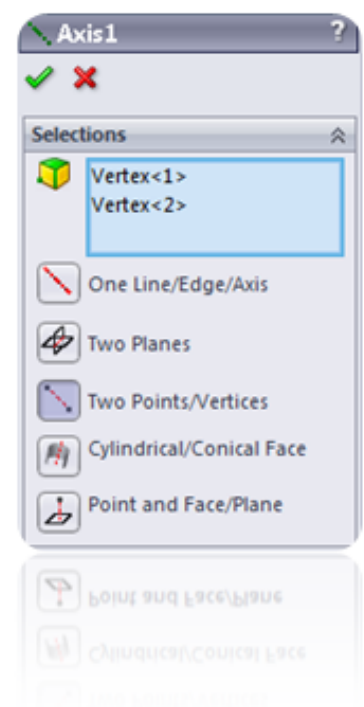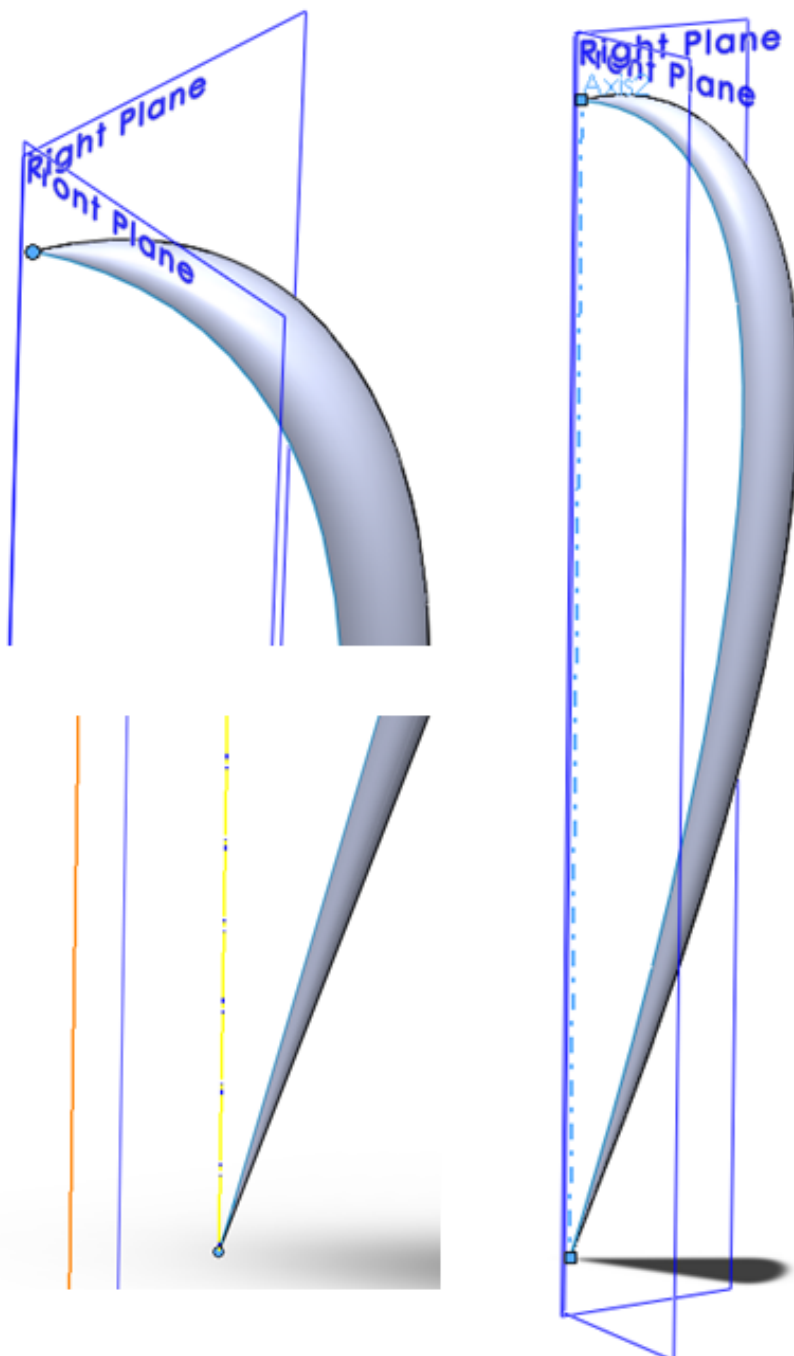Go to **Insert >Reference Geometry > Axis** or click at the Axis icon

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
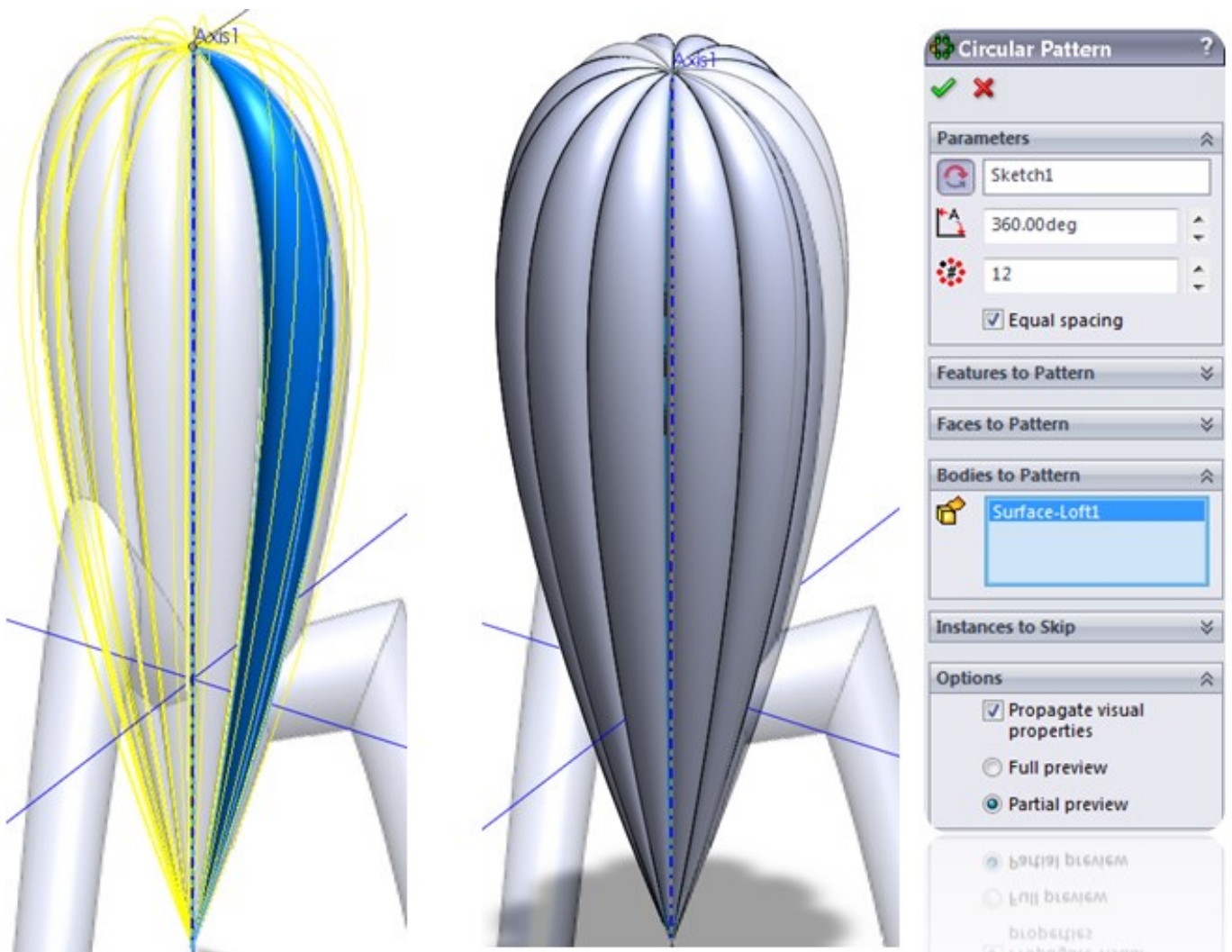
Change the Total Angle into 360 degrees

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

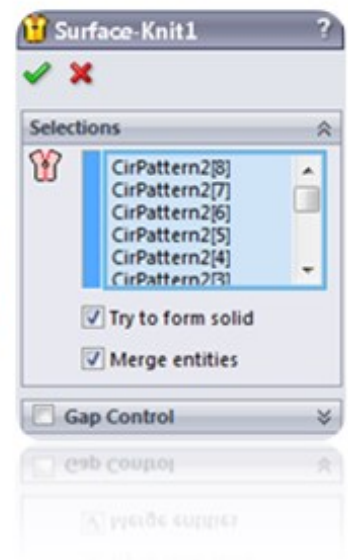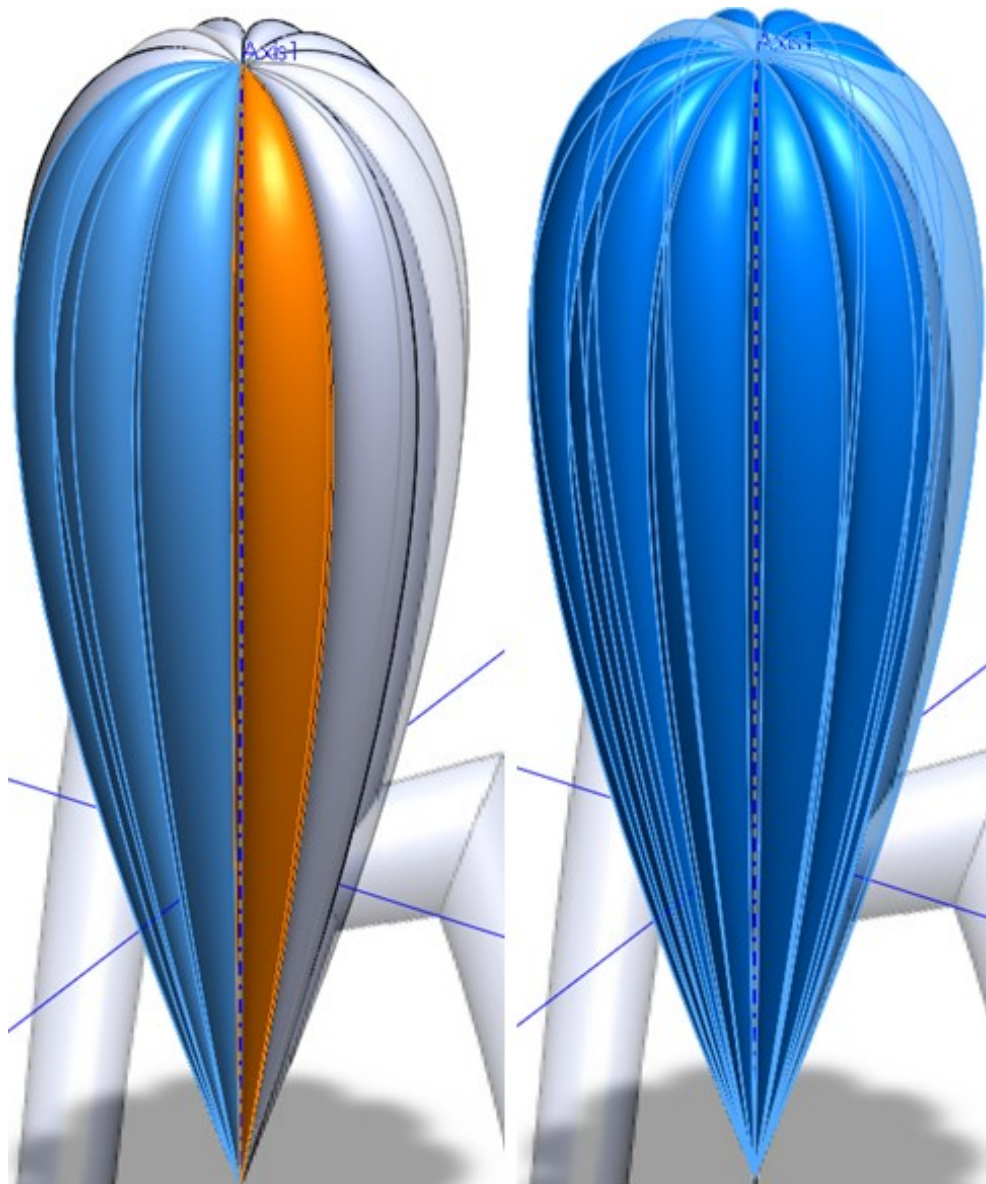Go to **Insert > Surface > Knit** or click at the Surface Knit icon

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
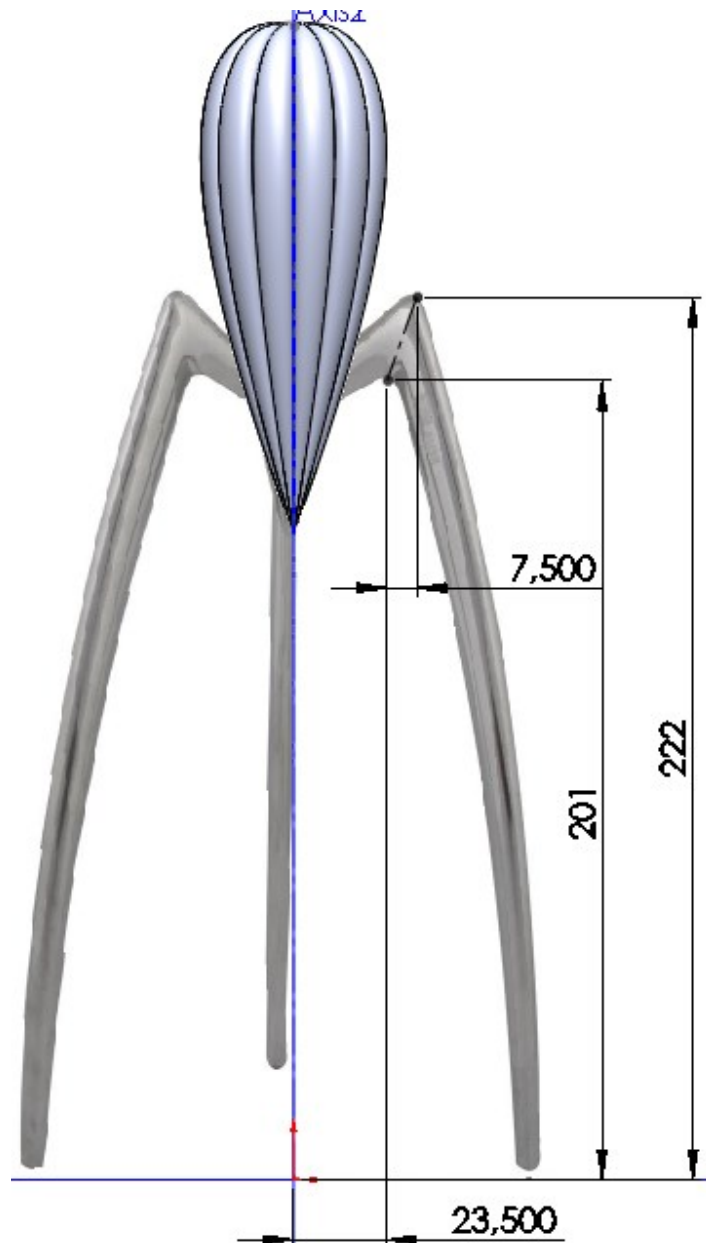
Draw a diagonal centerline

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
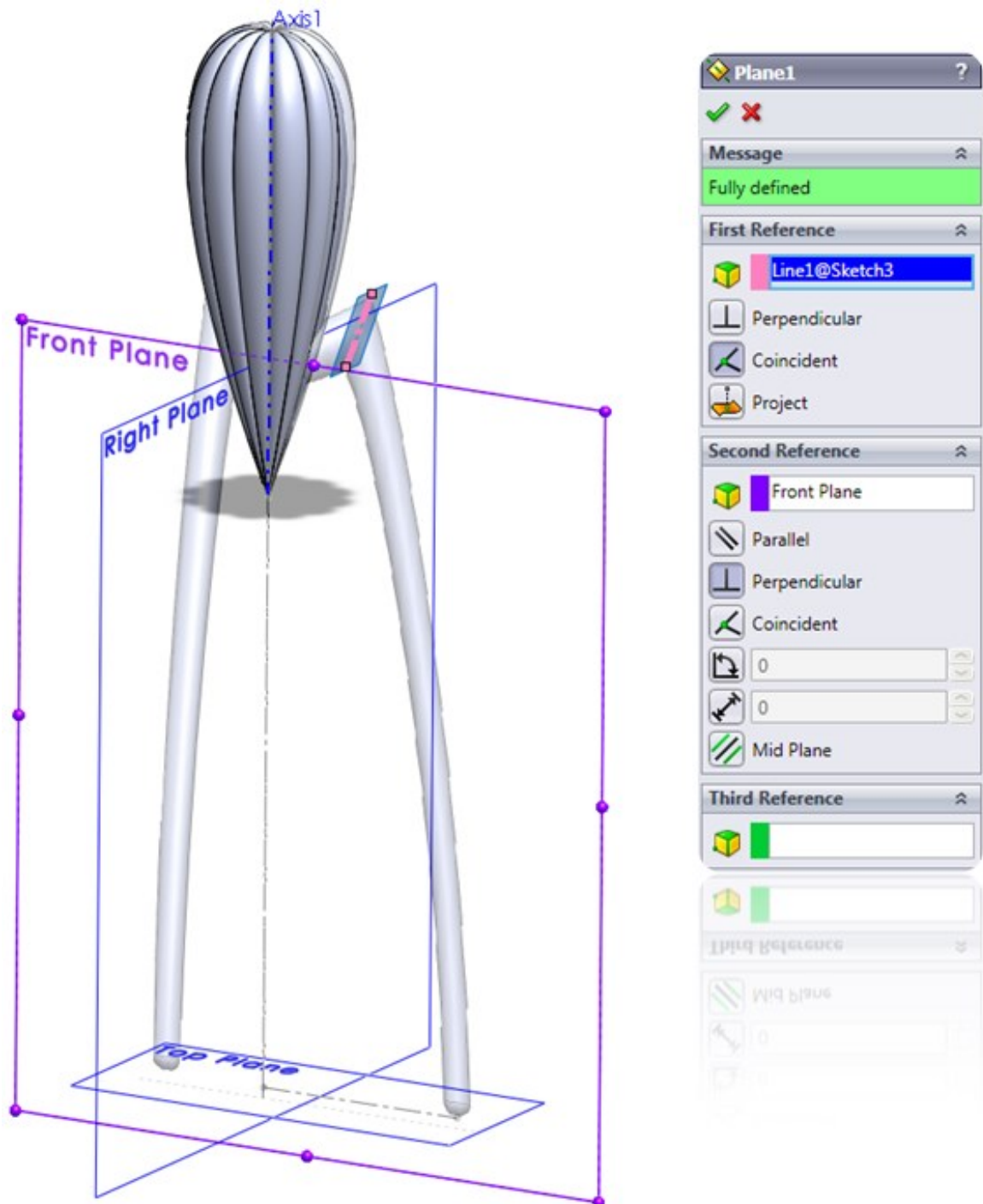
Create a new plane

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

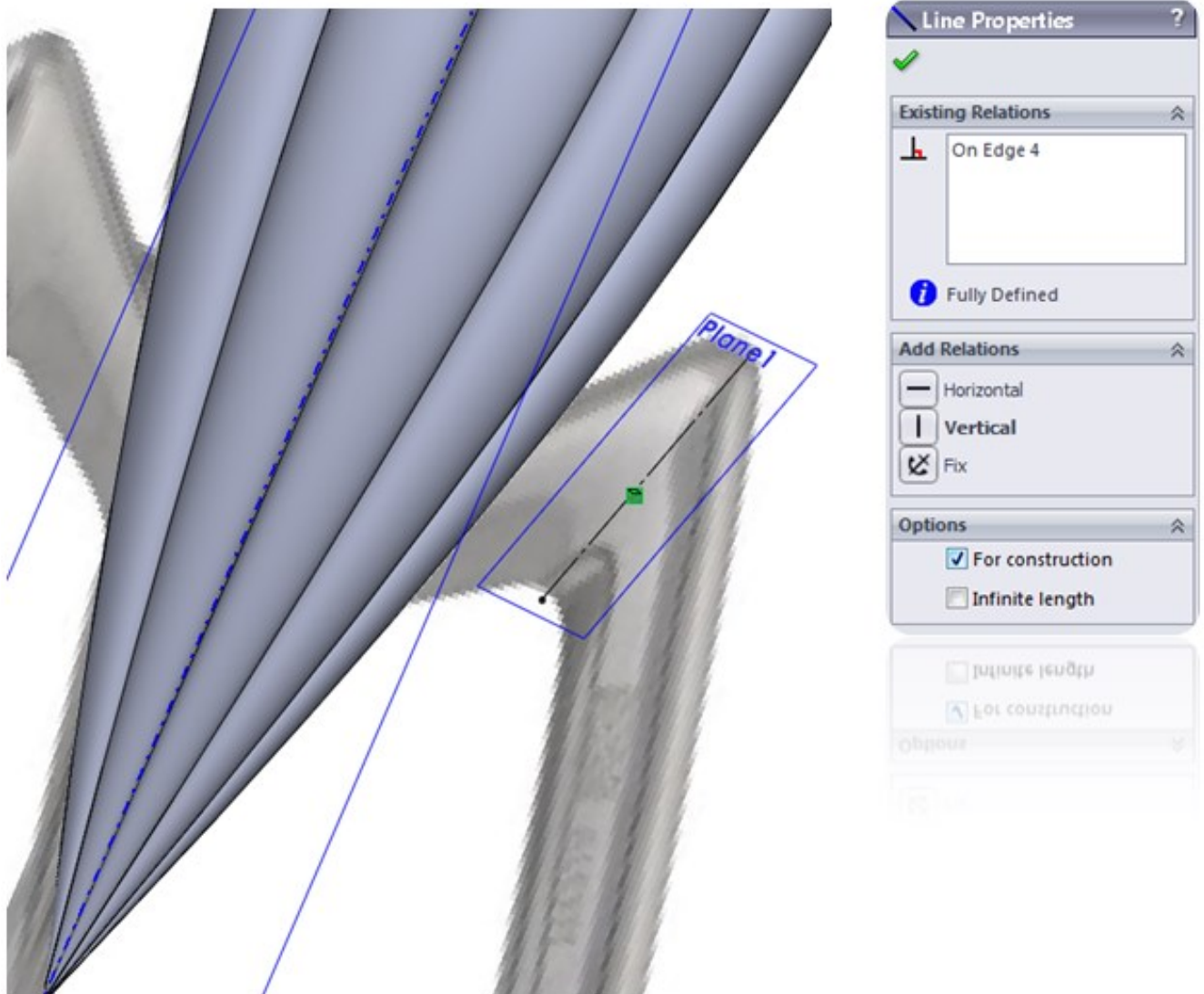**Convert Sketch 3 into the new Sketch4**

Click at the grey centerline of Sketch3

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

Click OK

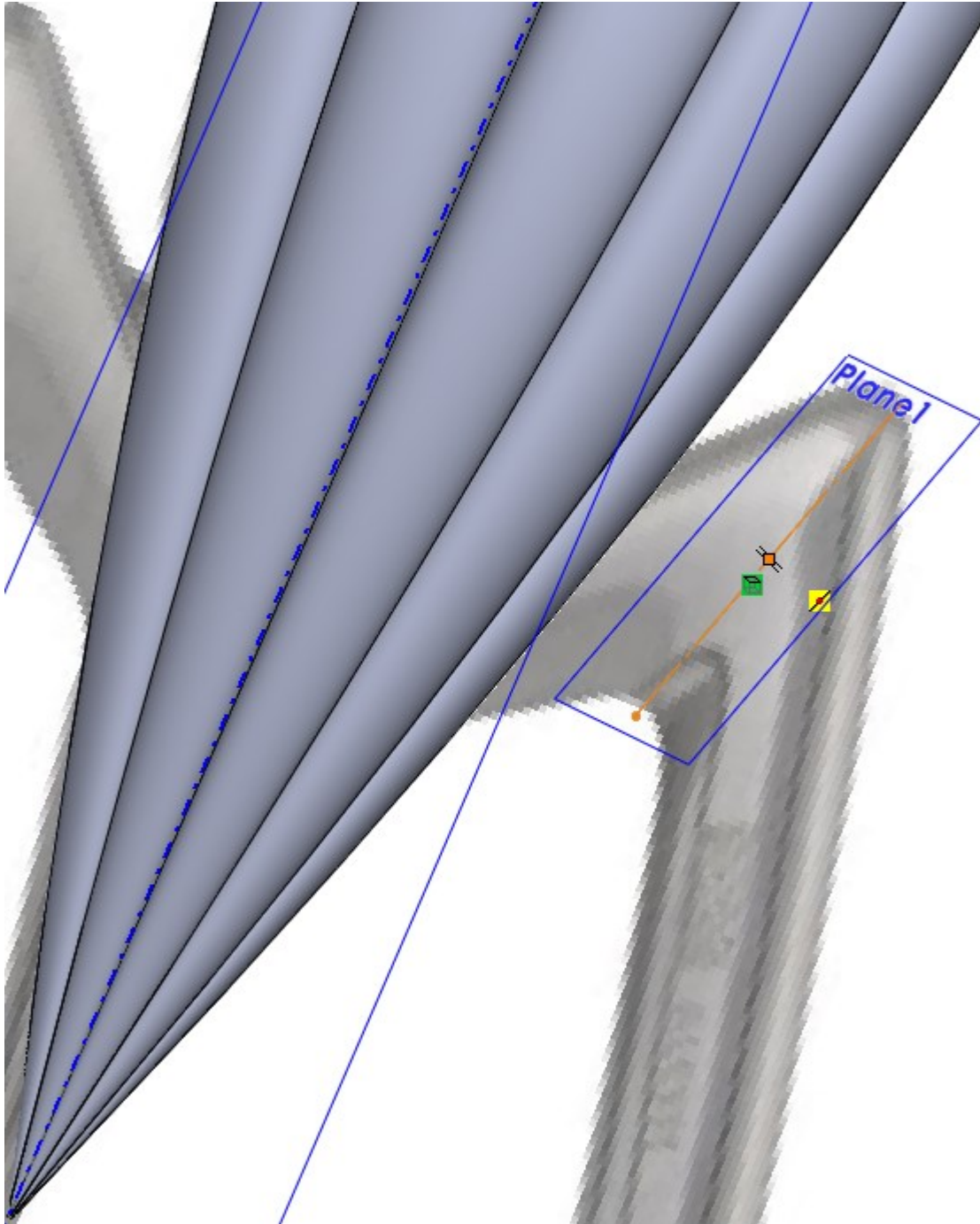Change the solid line into a centerline

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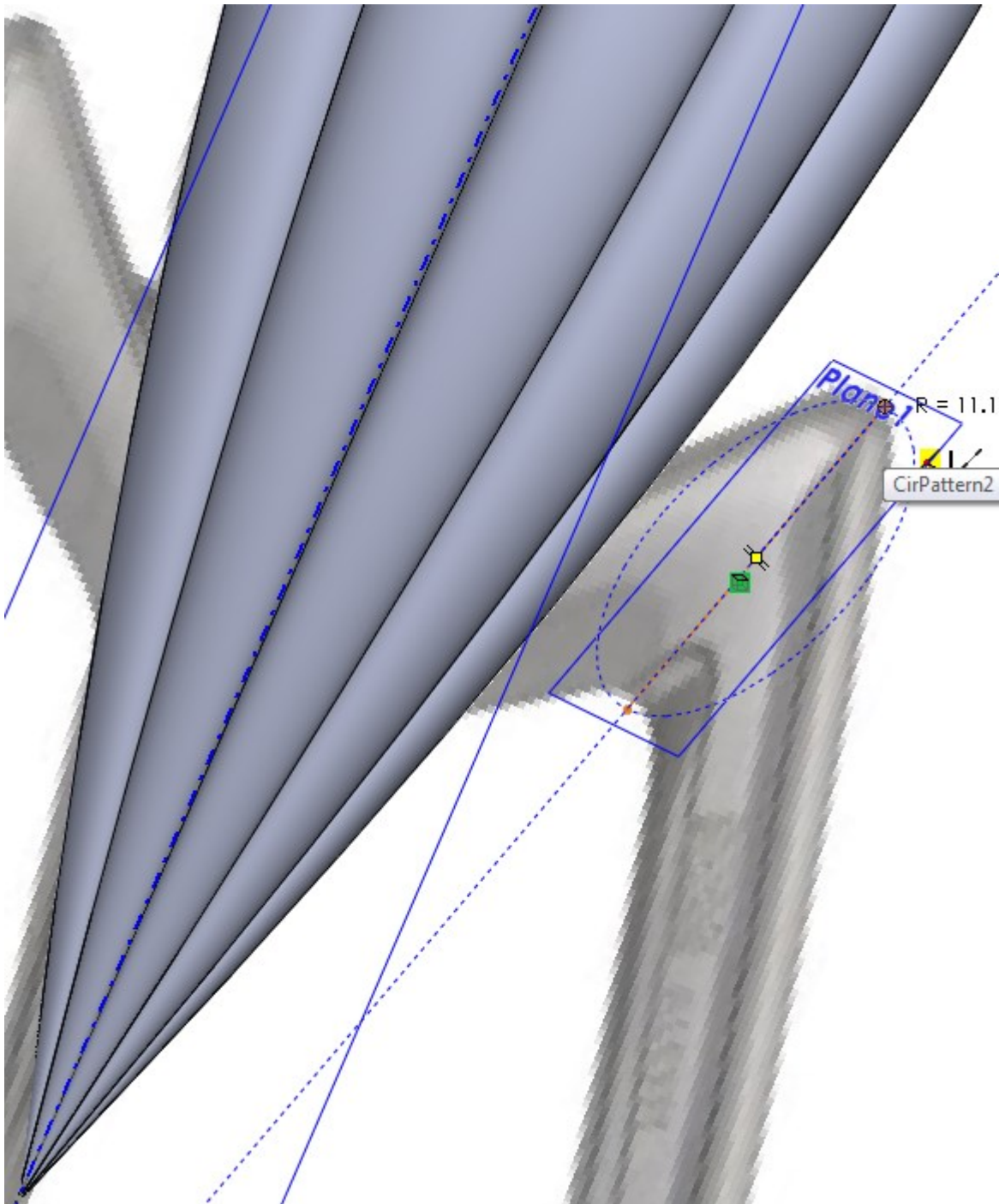
Create an Ellipse

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
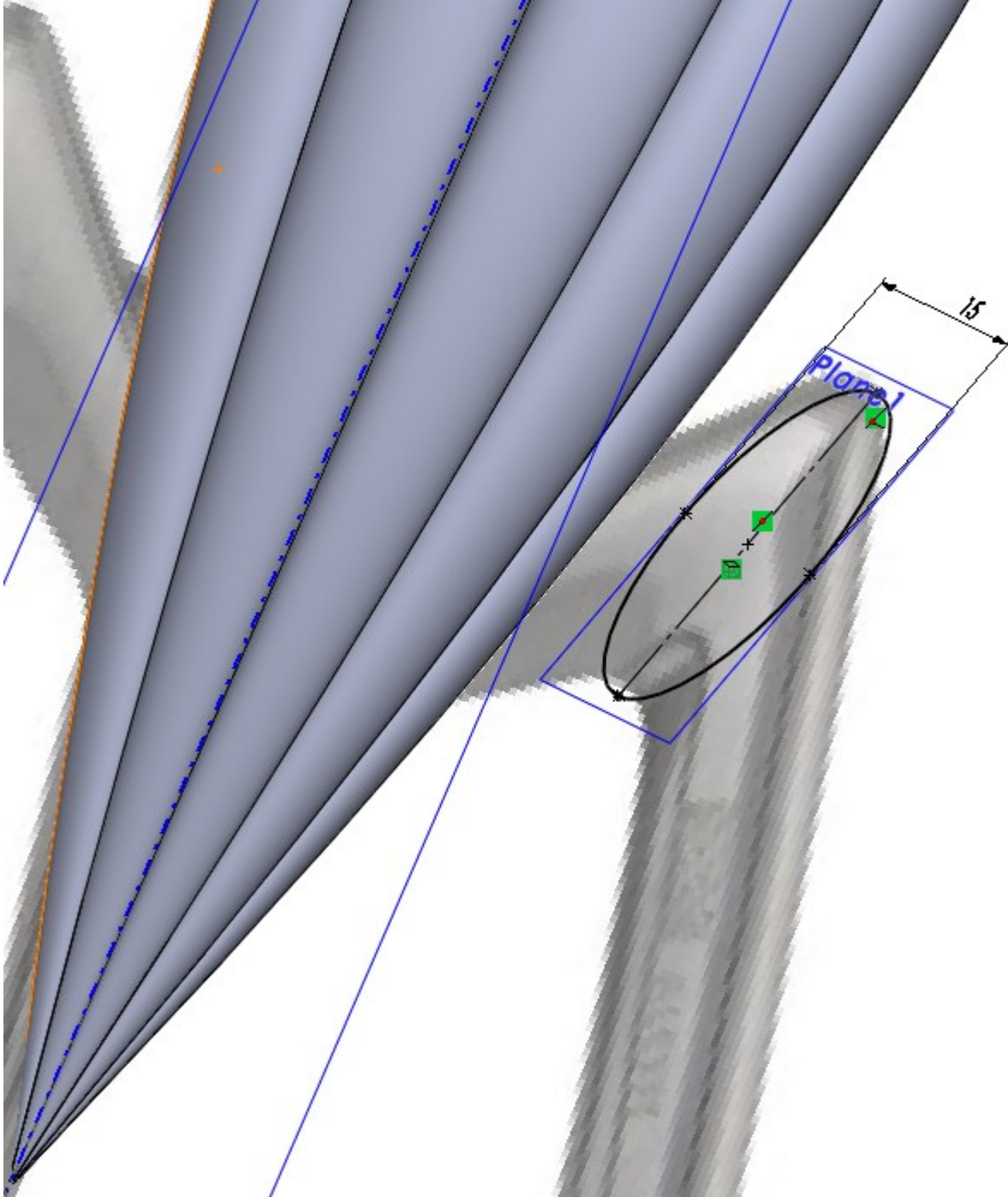

Click somewhere next to the construction line to set the width of the Ellipse

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

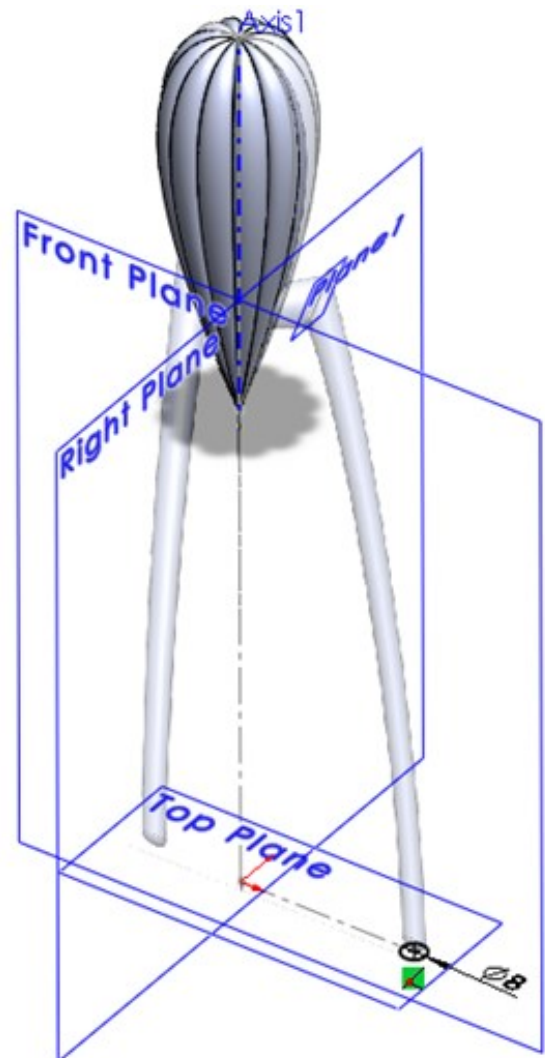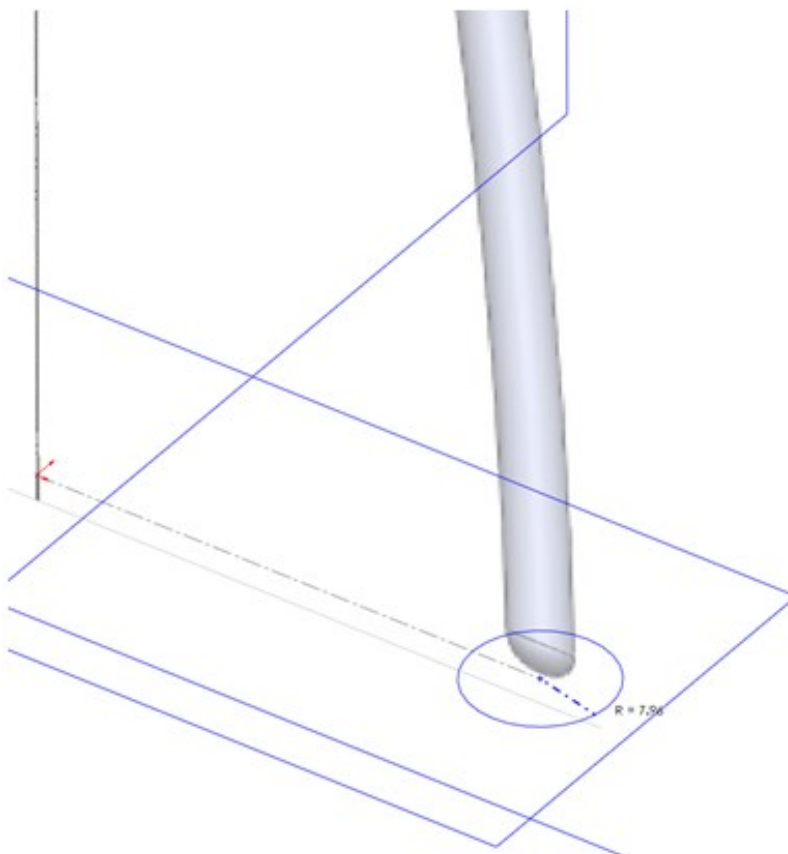Select the Top Plane and create a sketch by clicking on the 2D Sketch icon

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

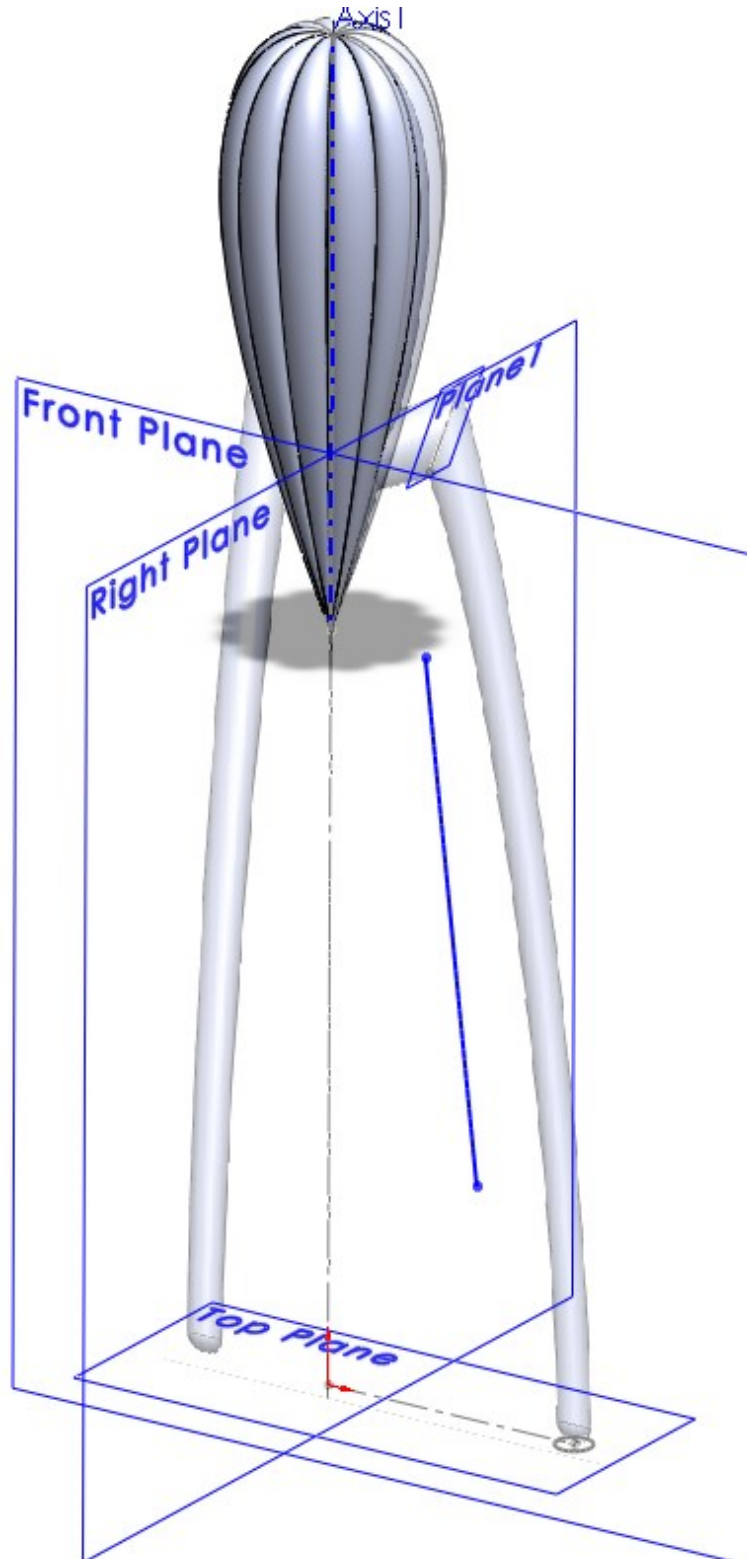Click at the Sketch button in the upper right corner close the 2D Sketch

Create another 2D sketch

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

Draw a spline without midpoints

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

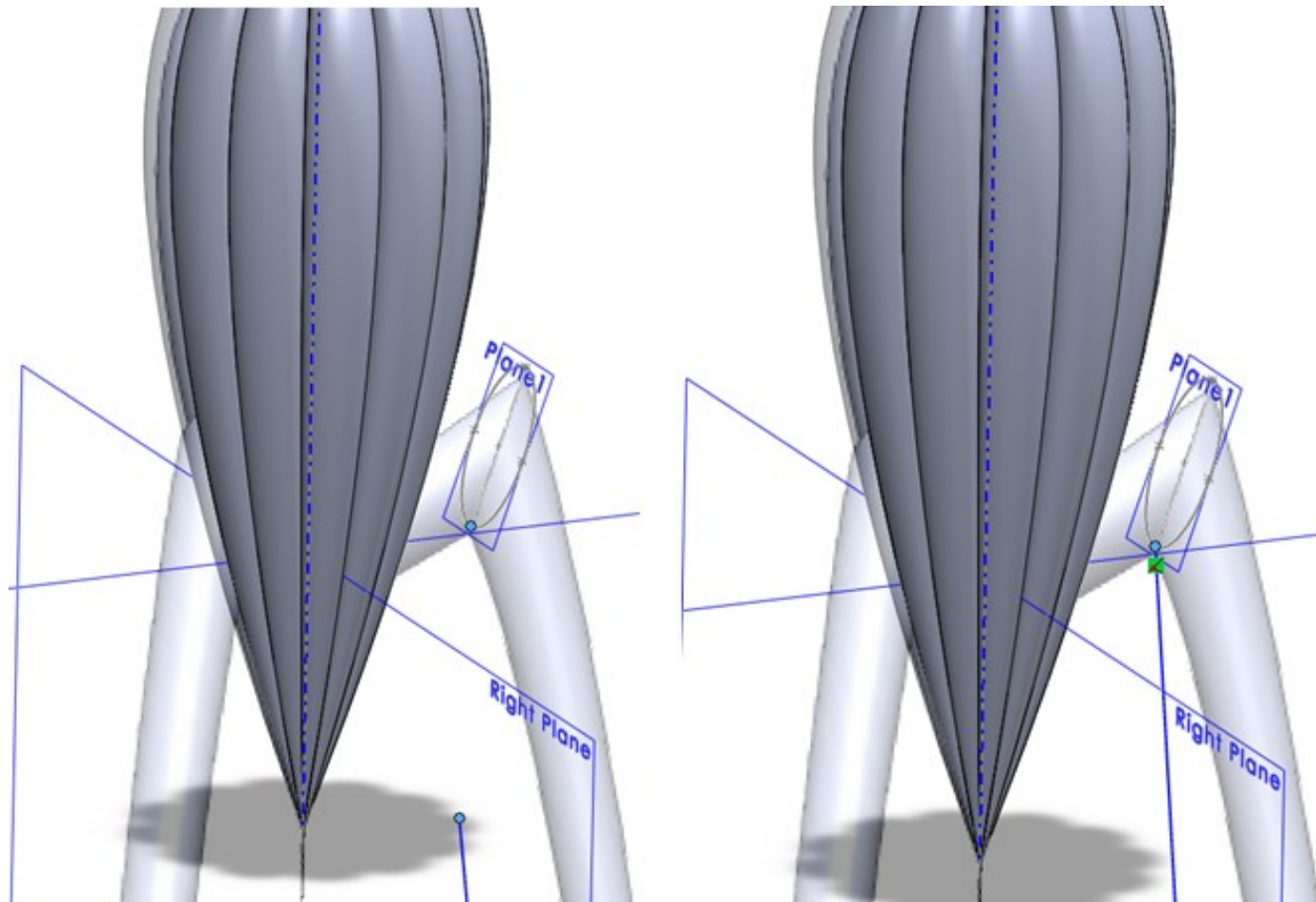**Connect the Spline with Sketch4 and Sketch5**

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
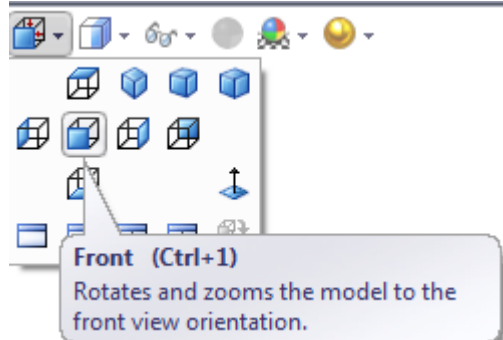
The spline is now connected with two different Sketches

Click OK
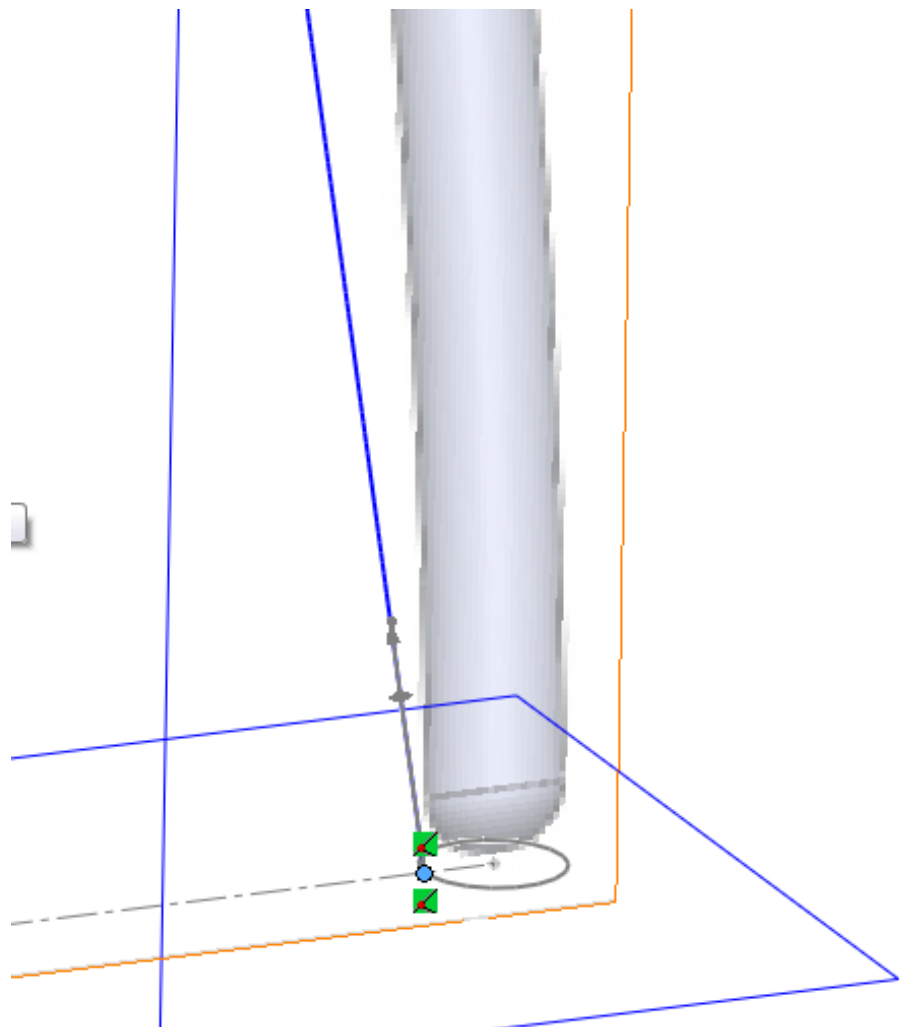
Change viewport to Front Plane

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

Front (Ctrl+1)
Rotates and zooms the model to the front view orientation.

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
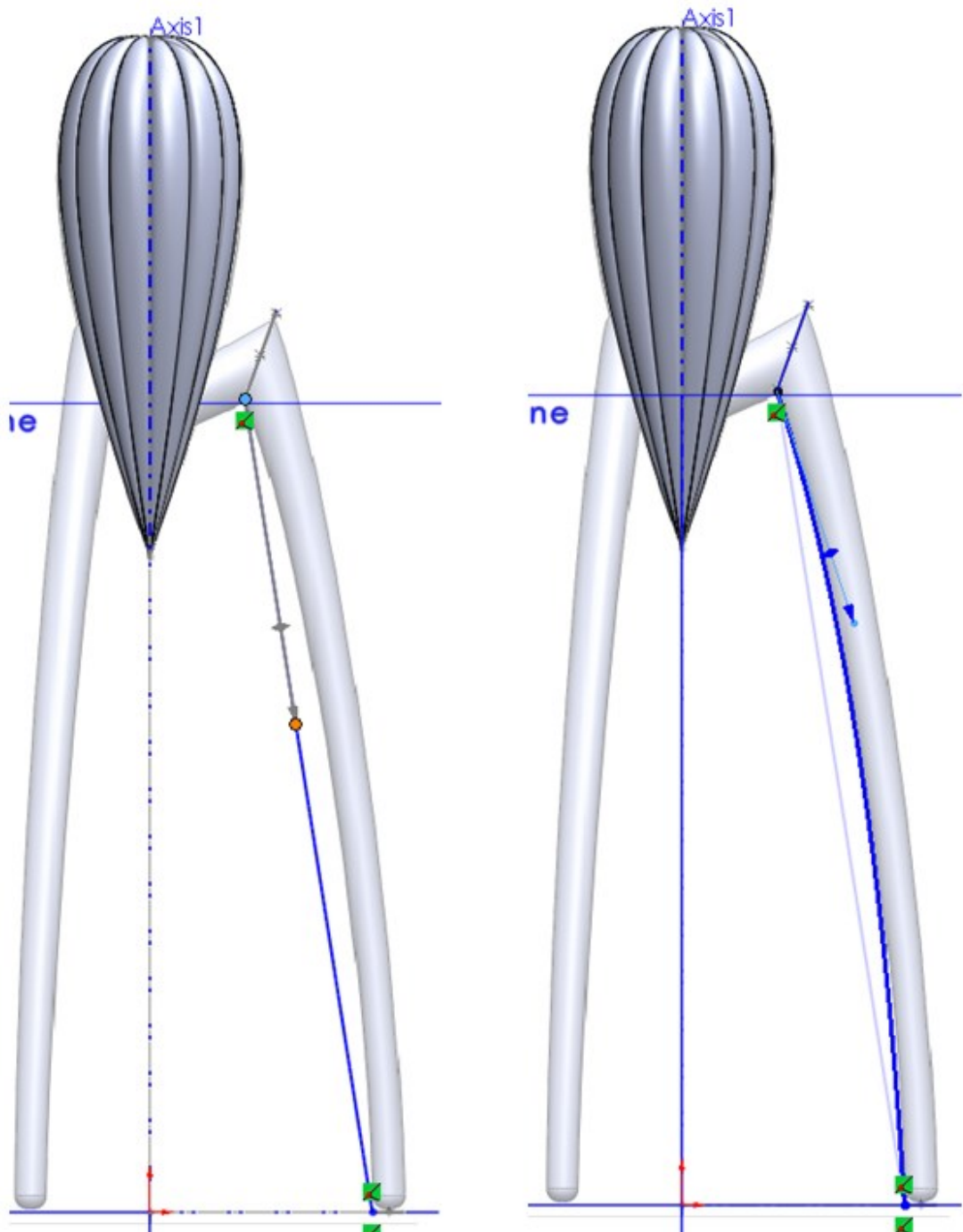
Create a Solid Loft

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

Click in the Profiles box

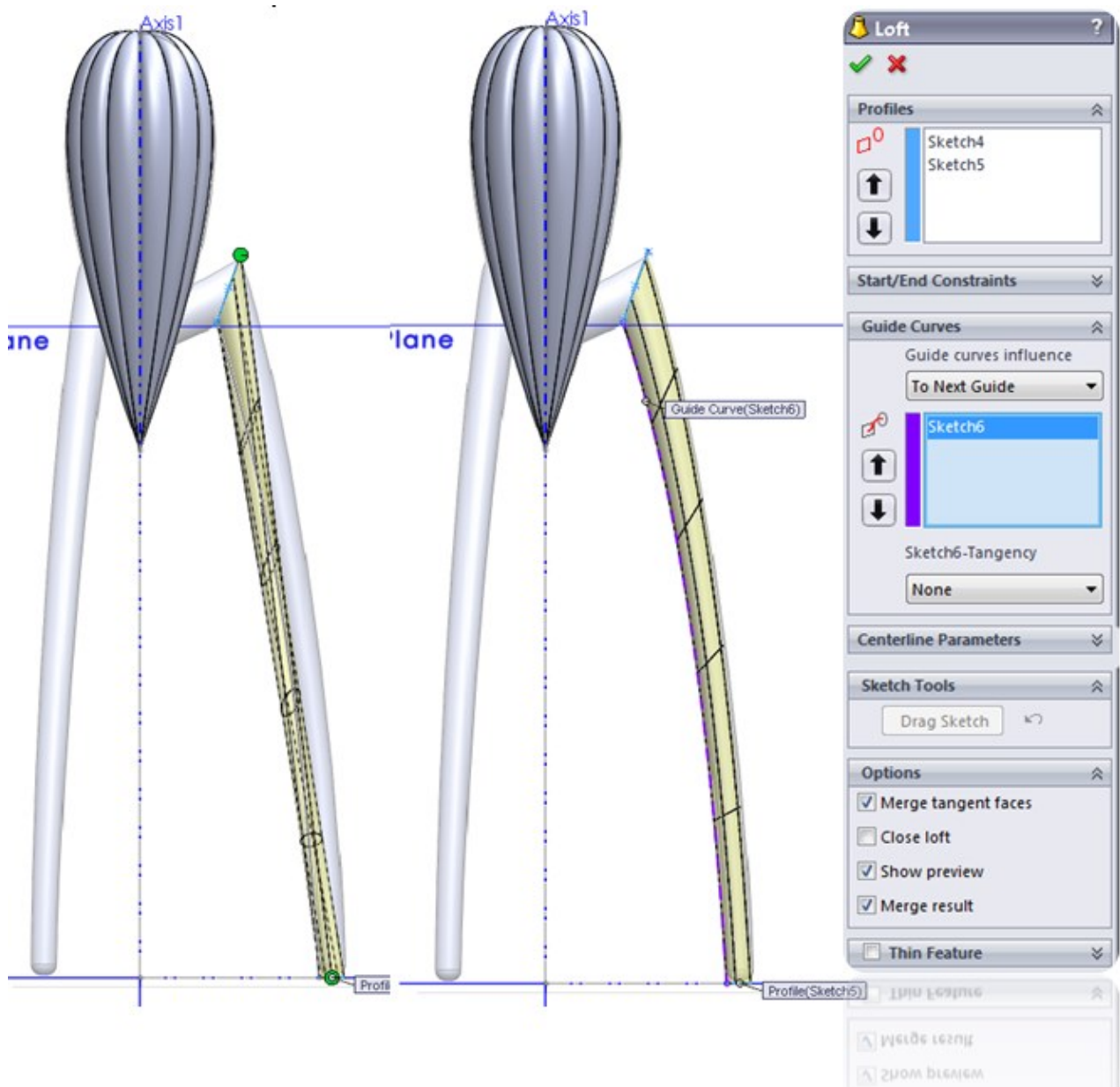Select Sketch4 and Sketch5 as shown in the picture

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
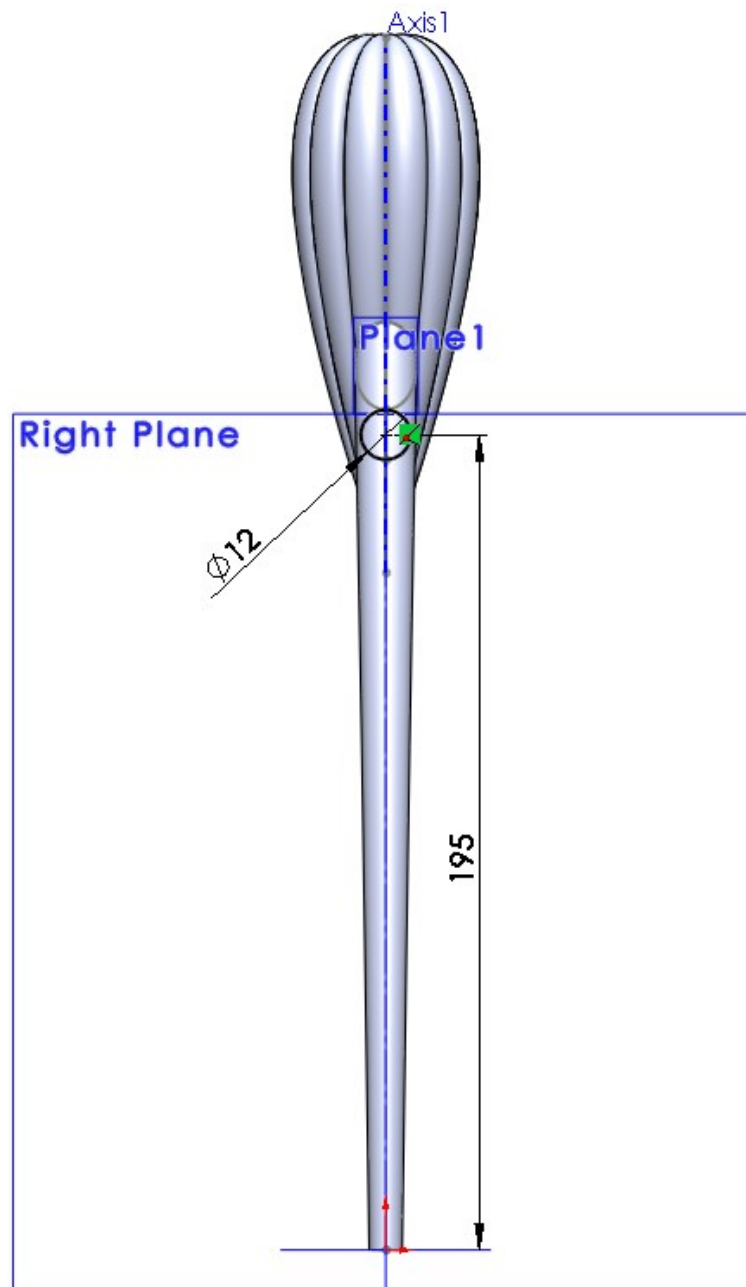
Create a circle

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
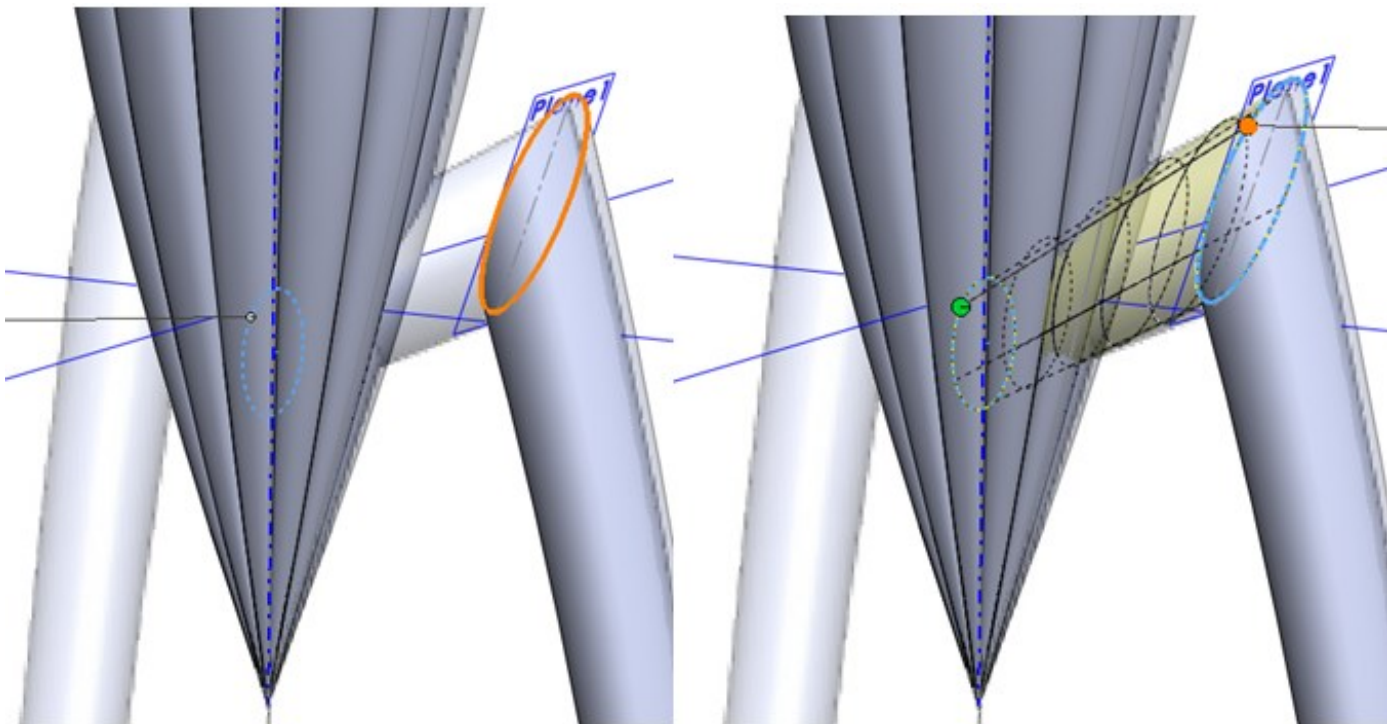
Create a Solid Loft

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
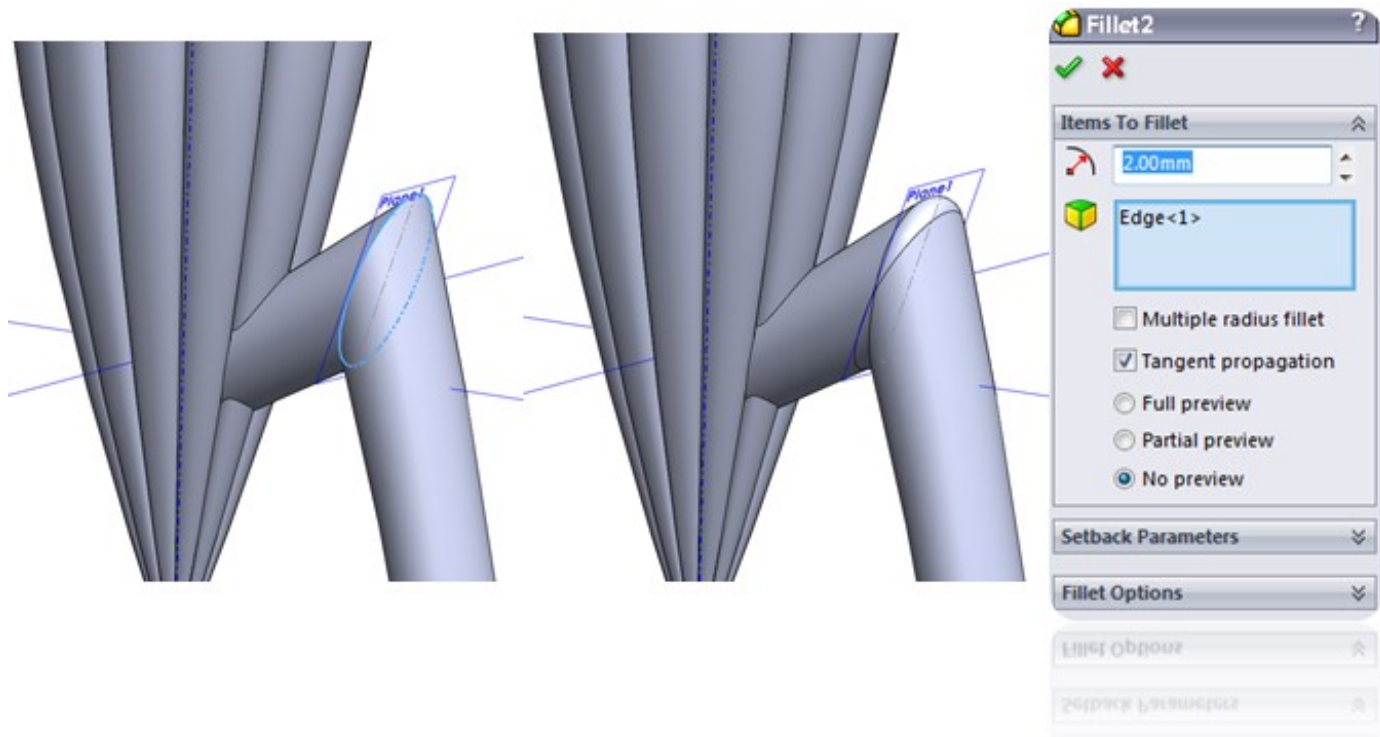
Click OK

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

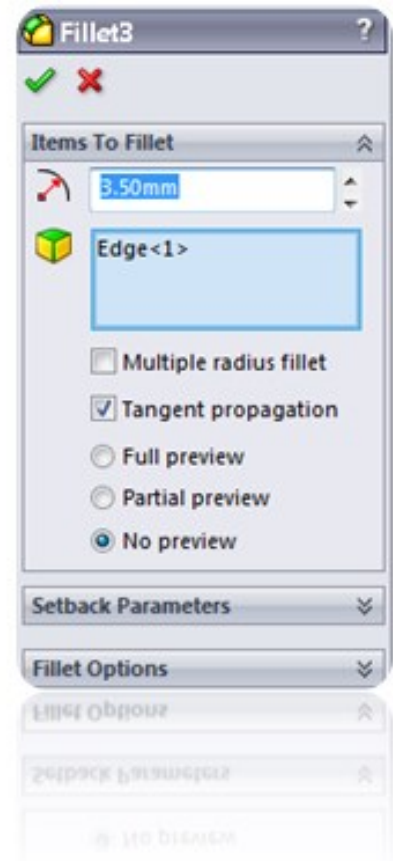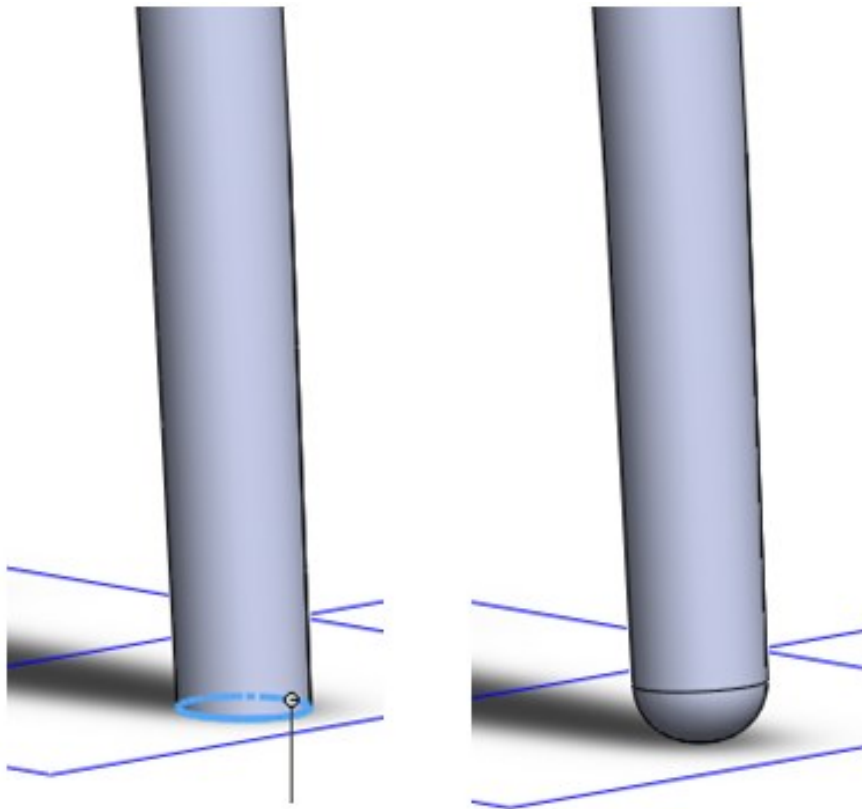Click at the blue edge as shown in the picture

Change the Radius into 2 mm

Click OK

Fillet3

Items To Fillet

3.50mm

Edge<1>

☐ Multiple radius fillet
☑ Tangent propagation
○ Full preview
○ Partial preview
● No preview

Setback Parameters

Fillet Options

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
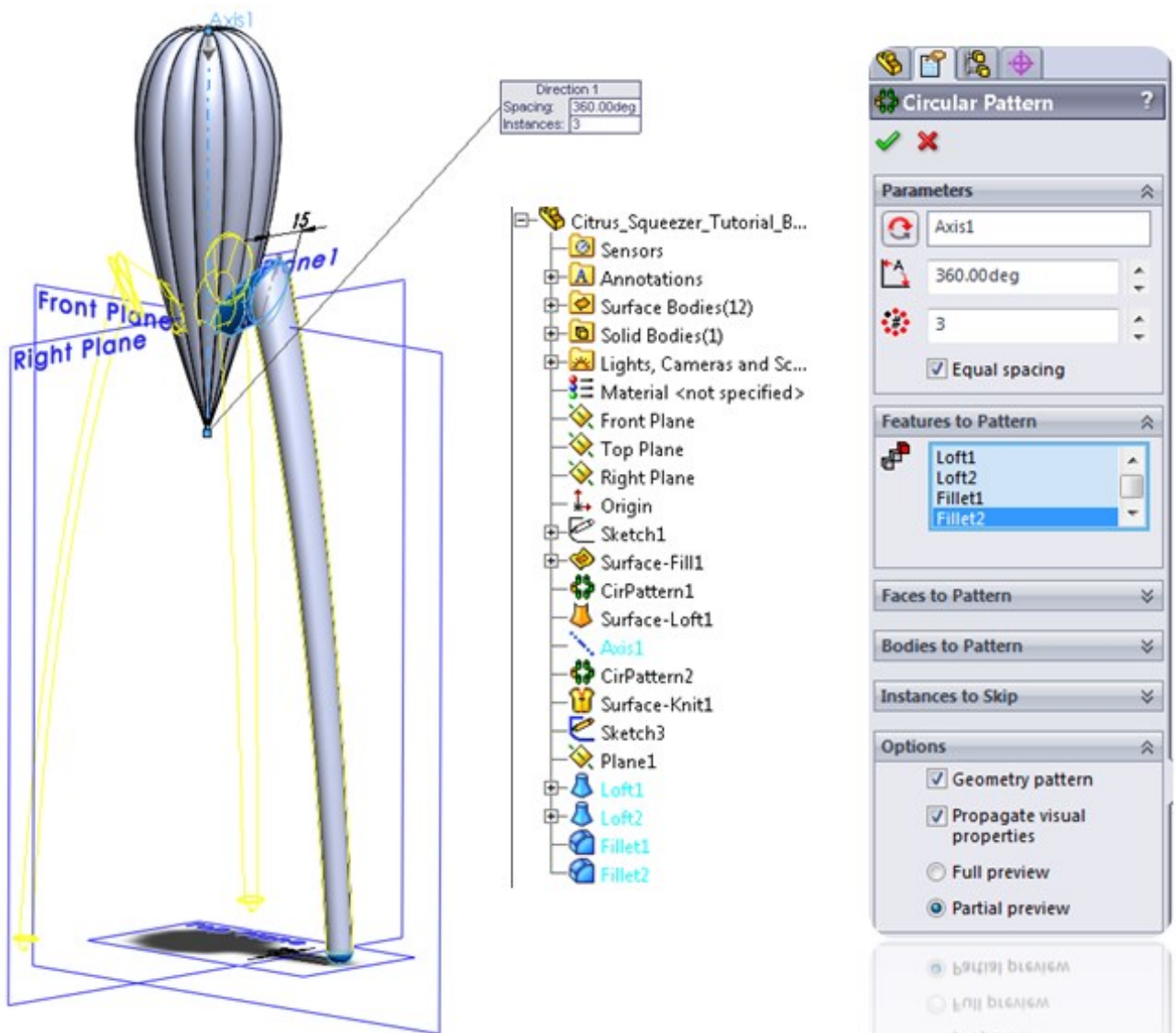
Change the Total Angle into 360 degrees

Change the Number of Instances into 3

Select the ''Equal Spacing'' option

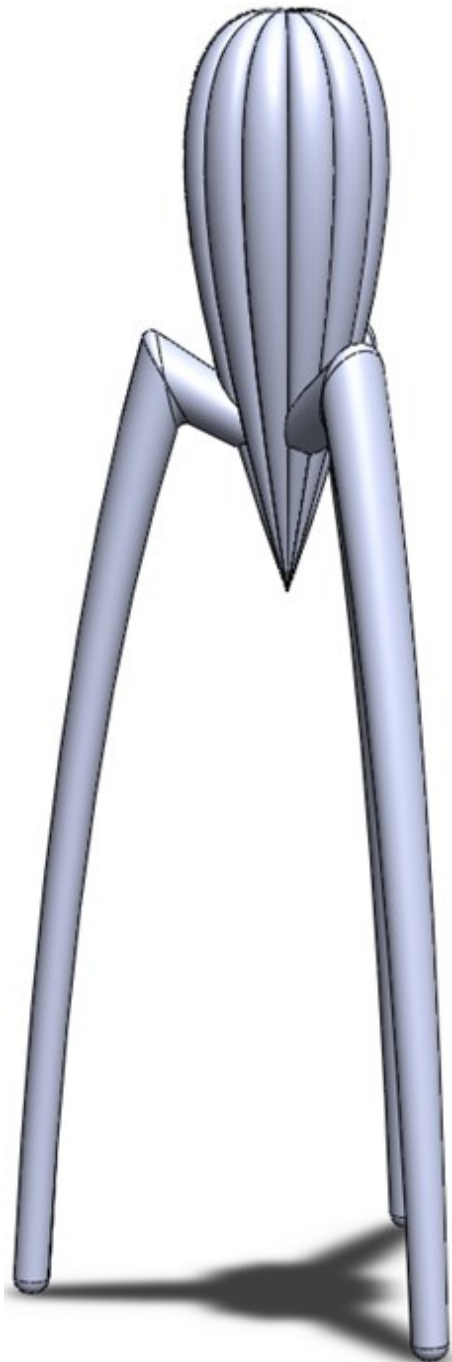Click at the **"Features to Pattern"** box

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

Click OK

Congratulations, you just finished your own Citrus Squeezer! ☺

Pretty impressive end result isn't it? I hope you've learned again something from this eBook!

Citrus Squeezer in SolidWorks   *Citrus Squeezer in PhotoView360*

If you liked this tutorial I also recommend you to take a look to my SolidWorks Starter Package.

In this package I will show you the basics of SolidWorks and PhotoView 360 in a fast and fun way. Click here for more information: http://learnsolidworks.com/starter-package

Best regards,

Jan

[Click here for my SolidWorks Starter Package](#)