LEARNSW

HOW TO MODEL A PANTON CHAIR IN SOLIDWORKS?

J.W.ZUYDERDUYN

HOW TO MODEL A PANTON CHAIR IN SOLIDWORKS?

"A step by step SolidWorks Tutorial"

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I often make updates and additions to this book because SolidWorks also keeps changing over the years. You can download the latest version of this book here:

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About the Author

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

I've been working since 2000 with 3d CAD software and since 2004 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 SolidWorks teacher of "Advanced Surface Modeling 3". I am specialized in concept design, 3d modeling and visualizations.

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 9 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 goal is to help as many people as I can with learning SolidWorks. That's why I've created the website, LearnSolidWorks.com. (By the way, I am not related or affiliated with SolidWorks in any way)

I offer this free eBook because I sell SolidWorks training courses. And there's a good possibility you will get inspired to become a real SolidWorks Pro. So if you find value in the help I give you, you might want to check my premium SolidWorks training where I teach you everything you need to know into becoming a SolidWorks Pro fast.

Feel free to share this eBook with your colleagues and friends.

Have fun modeling!

Jan

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How to Model a Panton Chair in SolidWorks?

In this SolidWorks tutorial I will show you how to model the famous Panton Chair in SolidWorks. Although the design of this organic chair dates from the sixties, it's still a modern design chair.

The designer of this chair is Verner Panton. He was one of the most influential industrial designers of the sixties and seventies. Verner Panton was born in Denmark and later relocated to Switzerland. He became famous for his original and modern furniture designs. The Panton Chair is for sure one of the most recognizable chairs in the world. The design of this chair was inspired by the Zigzag chair of **Gerrit Rietveld**.

One of the ambitions of Verner Panton was to create a plastic chair molded in one single piece. Panton searched a long time for a manufacturer of this modern chair. Finally he found the company **Vitra** to develop the chair for series production. The Panton chair received many different awards and is now recognized as a classic of modern furniture design.

Because the shape of this chair is so organic and challenging (especially for SolidWorks users) I thought it would be great to make a SolidWorks tutorial about this chair. I hope you will learn a lot of it. If you want to share this tutorial with your friends, or just want to leave a reaction, you can do that **here**. I am also looking forward to hear from you and will personally read all your comments!

In this tutorial you will learn how to use the following functions:

- Draw 2D Sketches
- Draw 3D Sketches
- Insert a Blueprint or Reference Picture
- Improve the shape of a Spline
- Create new Planes
- Create Projected Curves
- Pierce Multiple Sketches with each other
- Create Surface Lofts
- Use Guide Curves
- Hide Bodies
- Hide Pictures
- Knit Surfaces
- Solidify Surface Bodies
- Create Fillets
- Create Variable Fillets
- Change Display Styles
- Mirror and Merge bodies
- Change Colors
- Create Wall Thickness

Let get it started! ☺

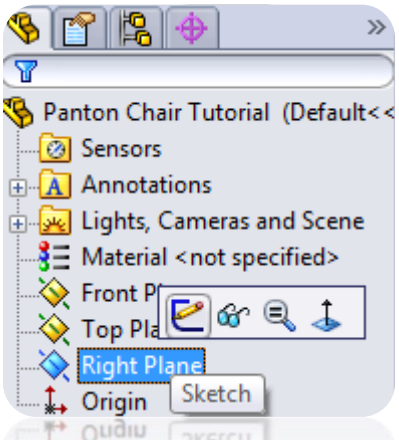
Open a new part with model units set to millimeters

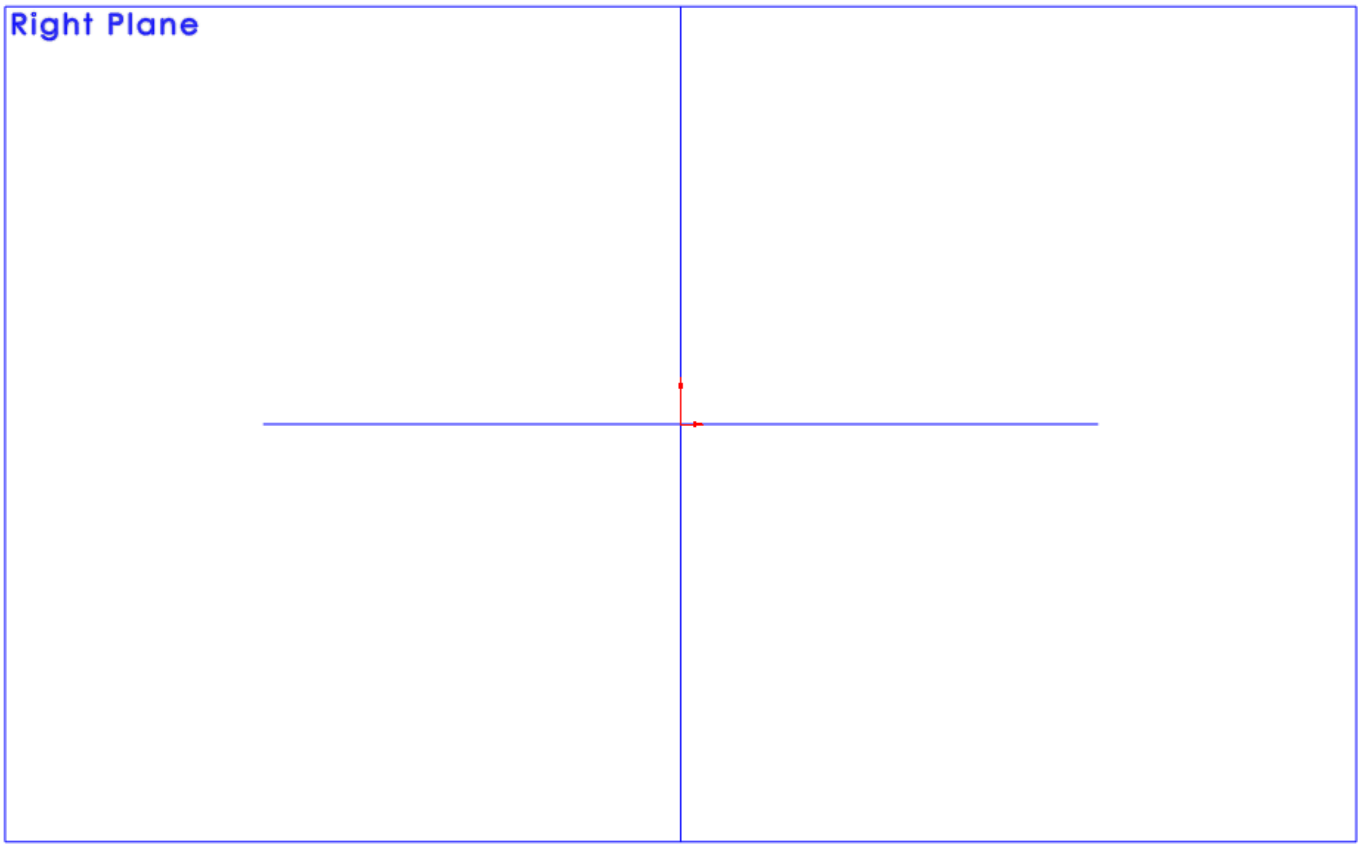
Go to: **File > New > Part**

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 picture of the Panton Chair to approach the nice shape as good as possible.

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

Go to your Blueprint folder and select the picture “SIDEVIEW_PANTON_CHAIR.Jpg” and save it into your SolidWorks folder.

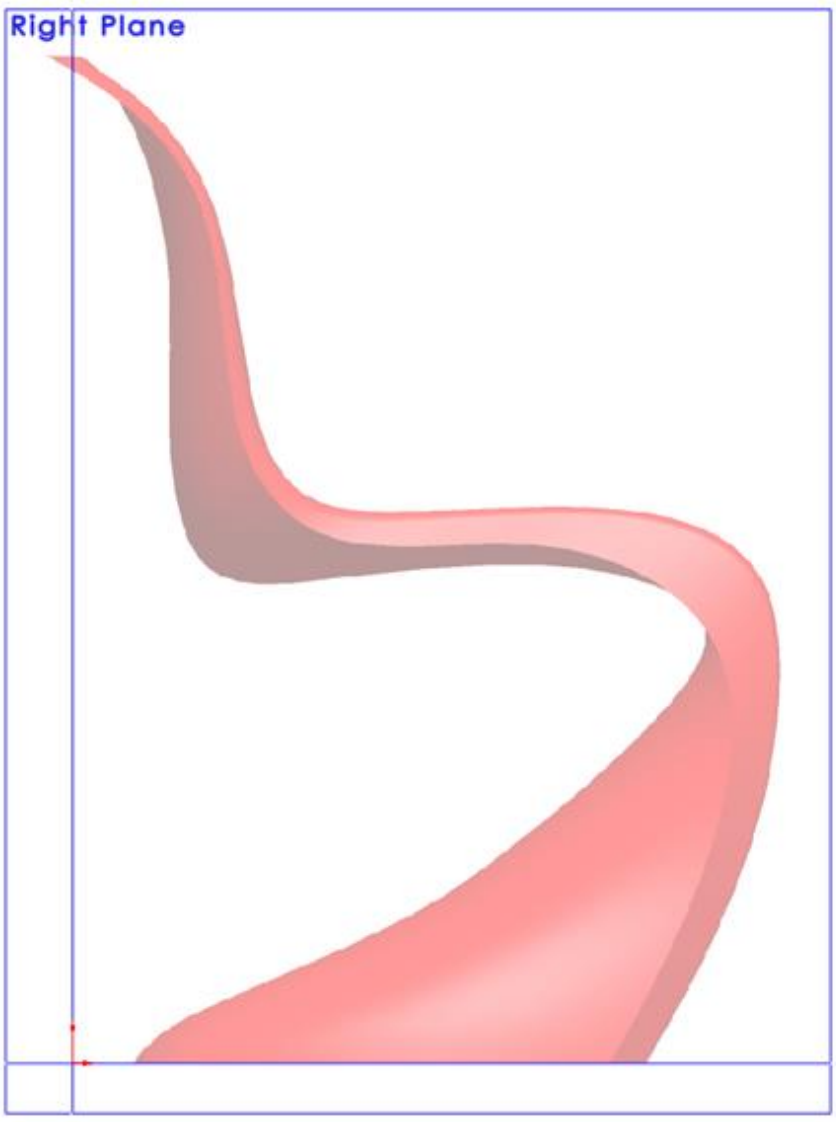
If you don’t have this picture you can download it **here**

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 0.50

Click OK



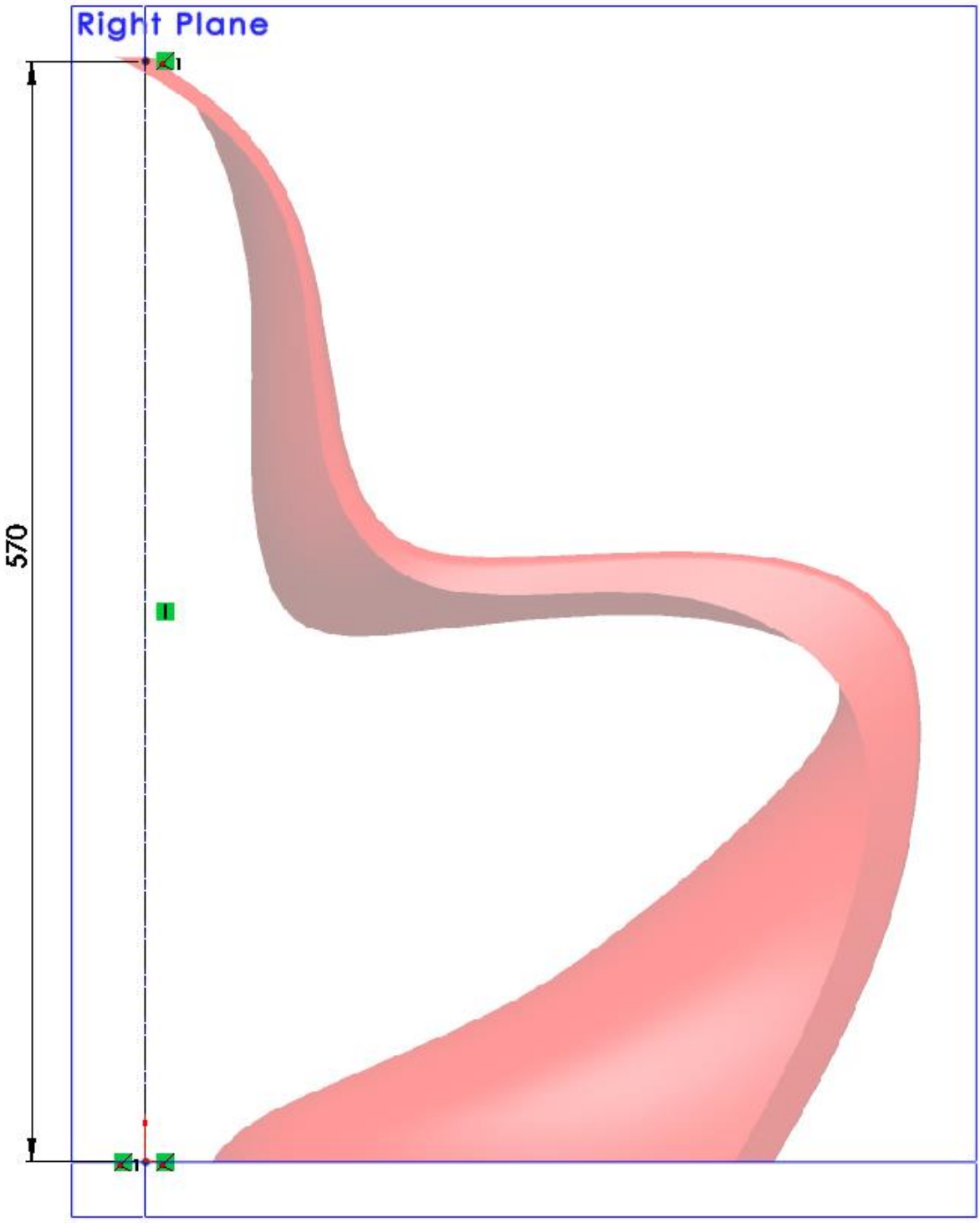
Draw a vertical centerline

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 570 mm by clicking at the dimension button

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



Create another 2D sketch

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

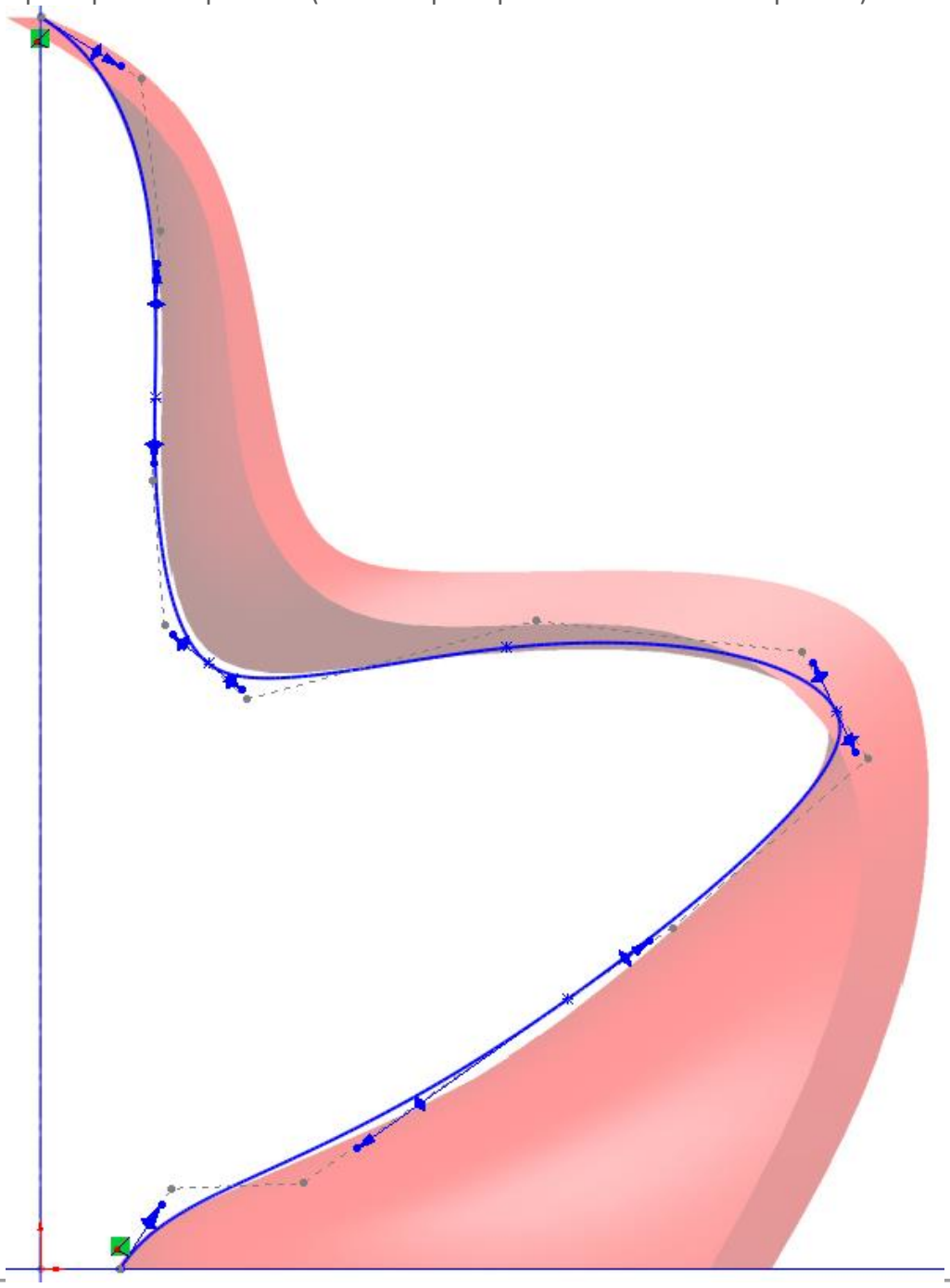
Draw a spline

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

Start the spline at the upper point of the construction line

Try to duplicate the lower curve of the chair as good as possible

Use as little spline points as possible (I used 7 spline points as shown in the picture)



Improve the shape of the curve

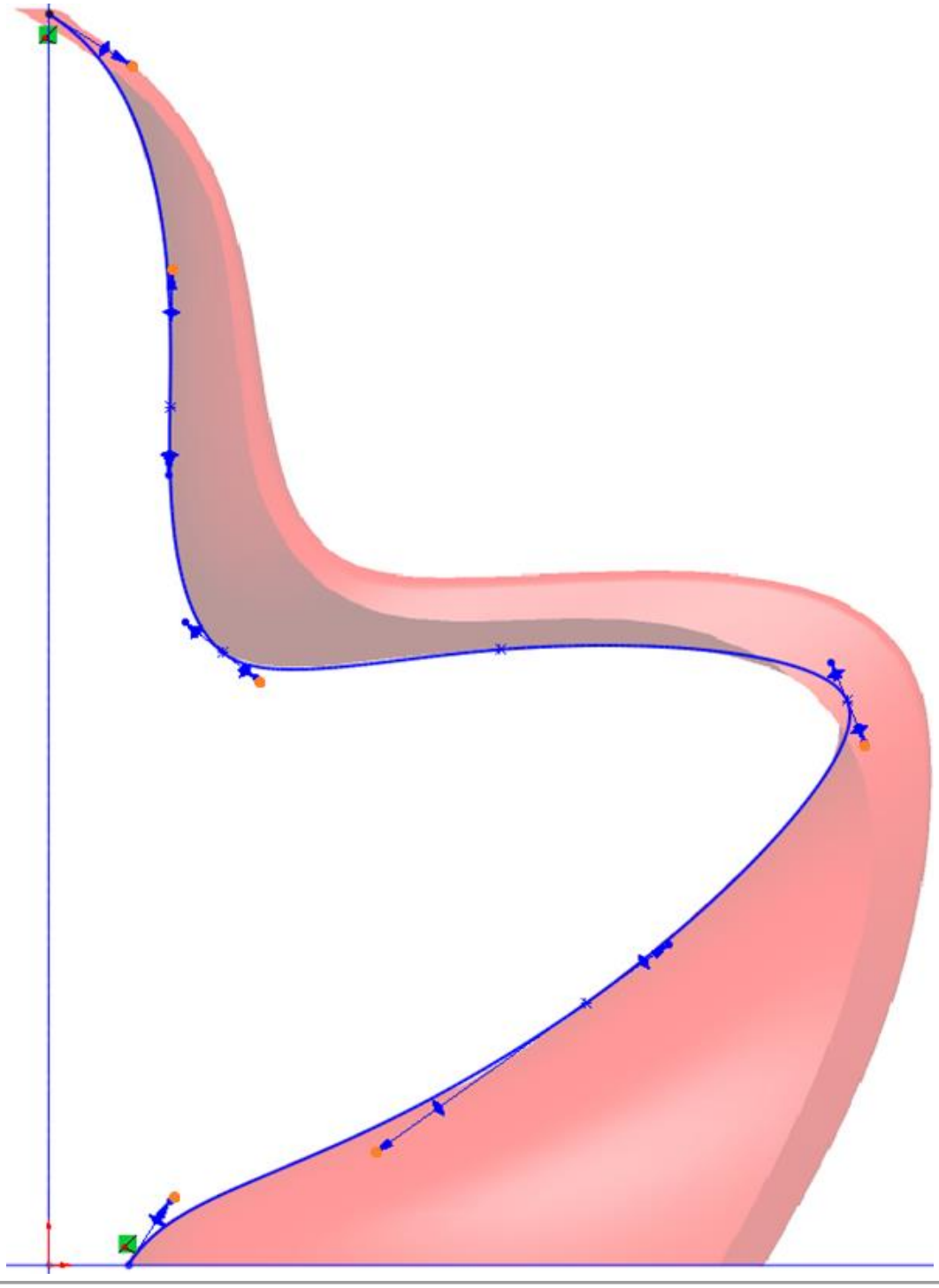
Click and drag the spline points to improve the shape of the curve

Change the direction of a spline point

Click on a spline point which you want to improve

The grey arrow of the Spline point appear

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

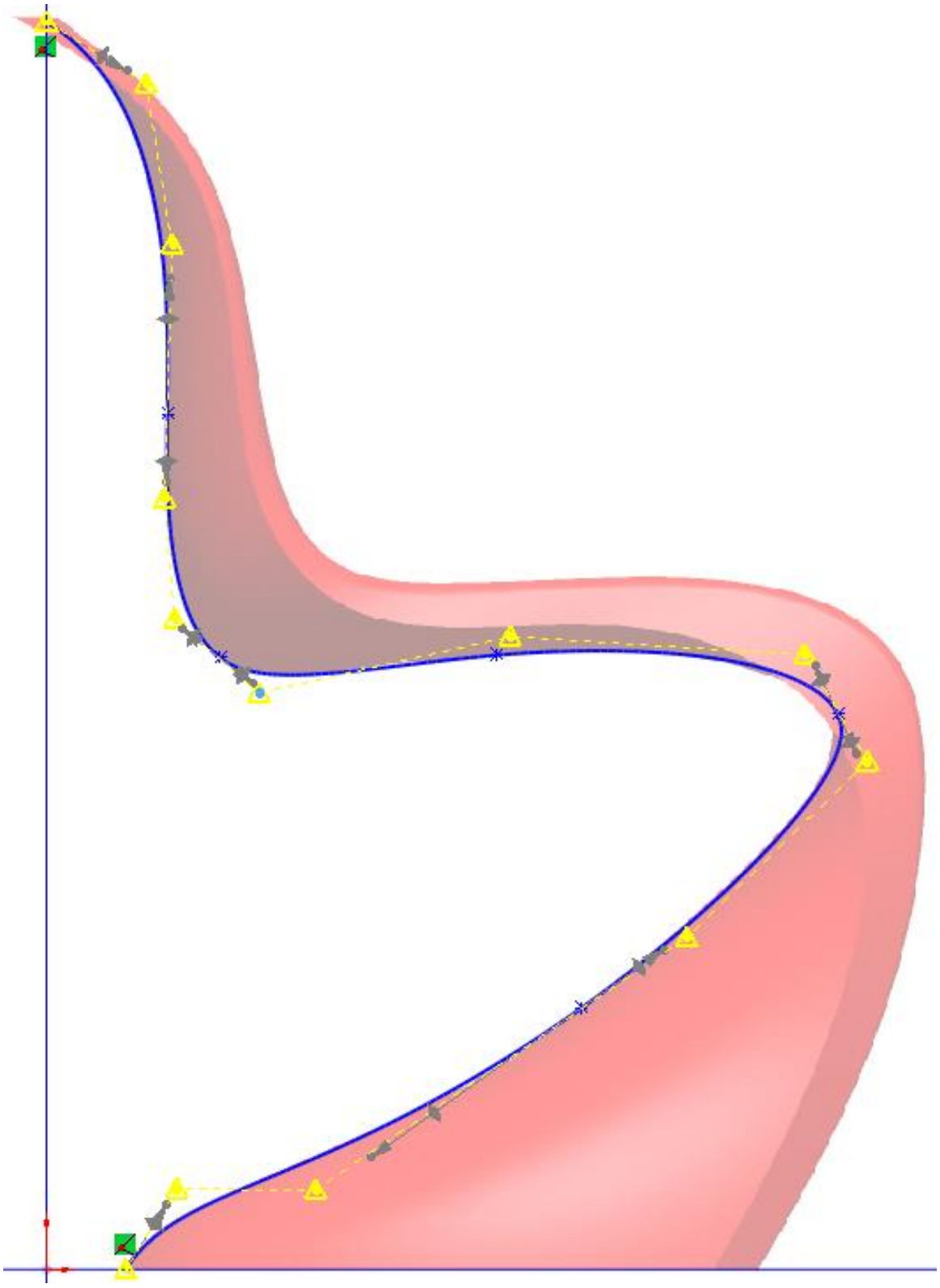


Improve the shape of the curve even more

If you're still not satisfied with the curve you can use the Display Control Polygon option

Click on the **Spline > Right click > Display Control Polygon**

Click and drag one of the grey Polygon points to improve the shape even more

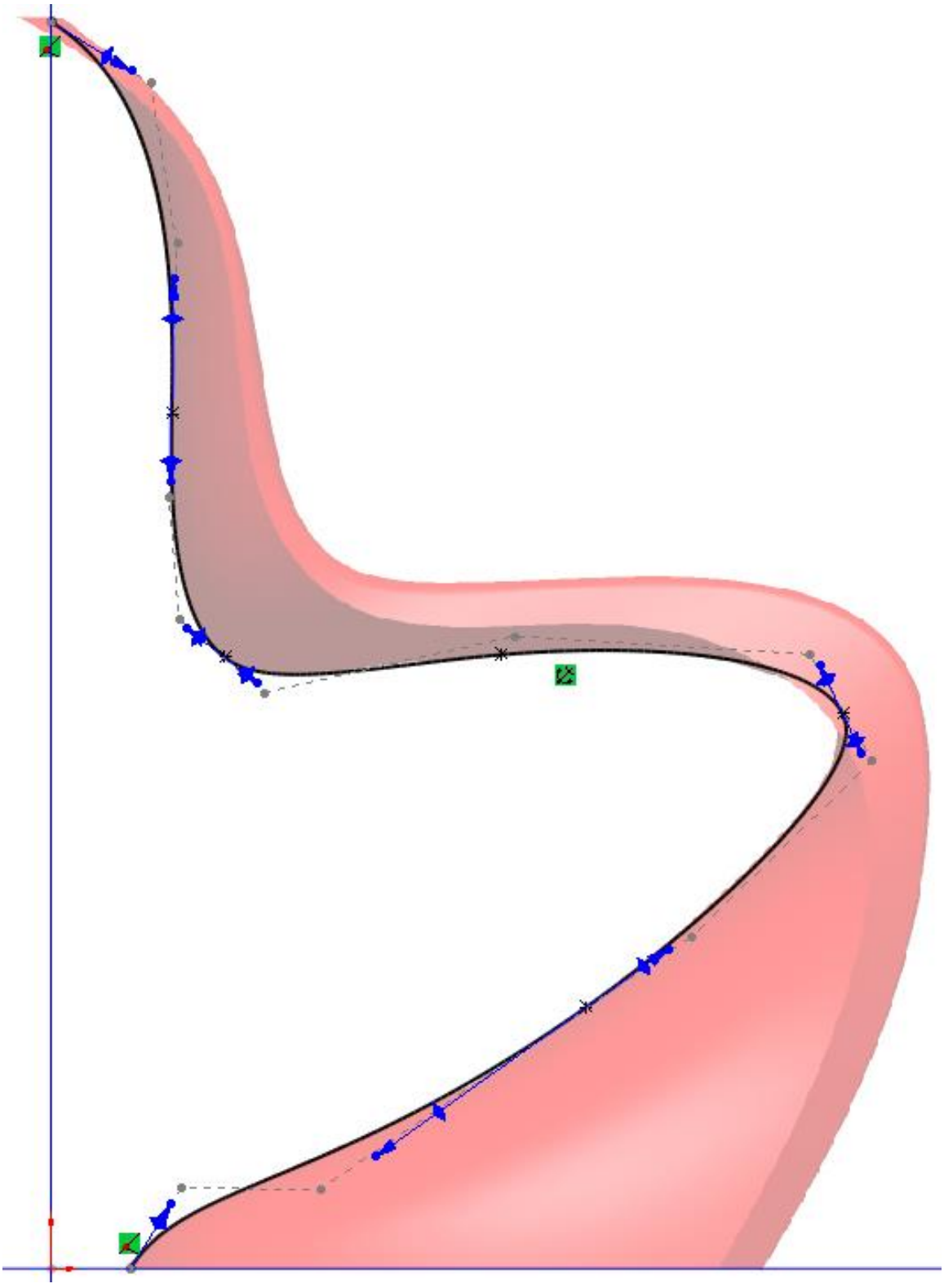


Fix the Spline

When you're satisfied with the curve, click on the Spline and select the **Fix button**

The color of the spline changed to black which means that it's fully defined

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



Create a new plane

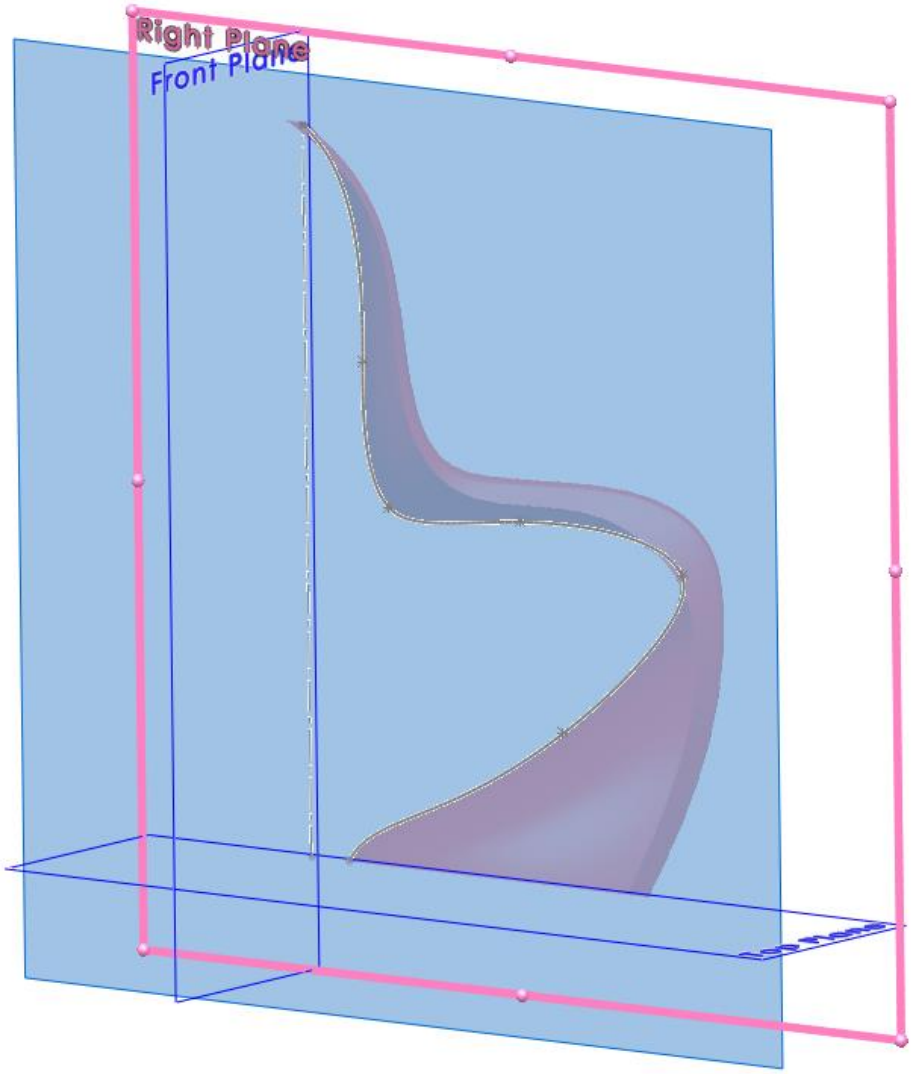
Go to: **Insert > Reference Geometry > Plane** or click at the New Plane icon 

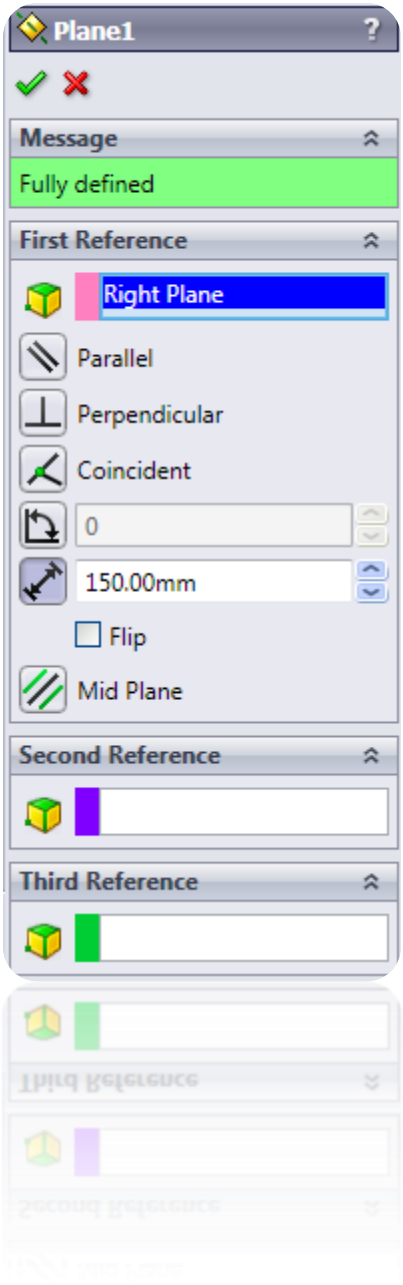
Select the Right Plane

Change the distance into 150 mm as shown in the picture

The new plane appears in blue

Click OK 





Create a 2D sketch on the new Plane1

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

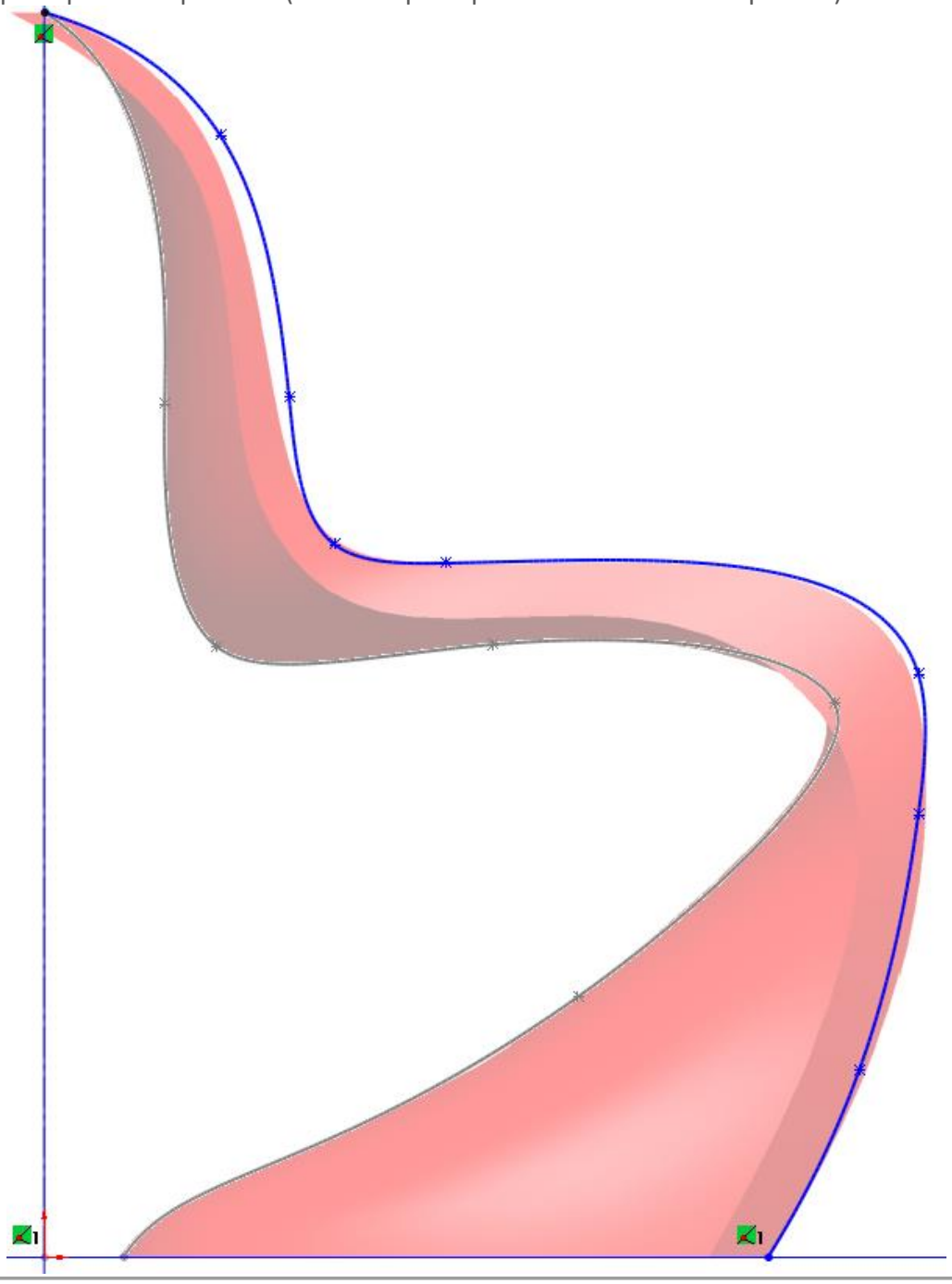
Draw a spline

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

Start the spline at the upper point of the construction line

Try to duplicate the upper curve of the chair as good as possible

Use as little spline points as possible (I used 9 spline points as shown in the picture)



Improve the shape of the curve

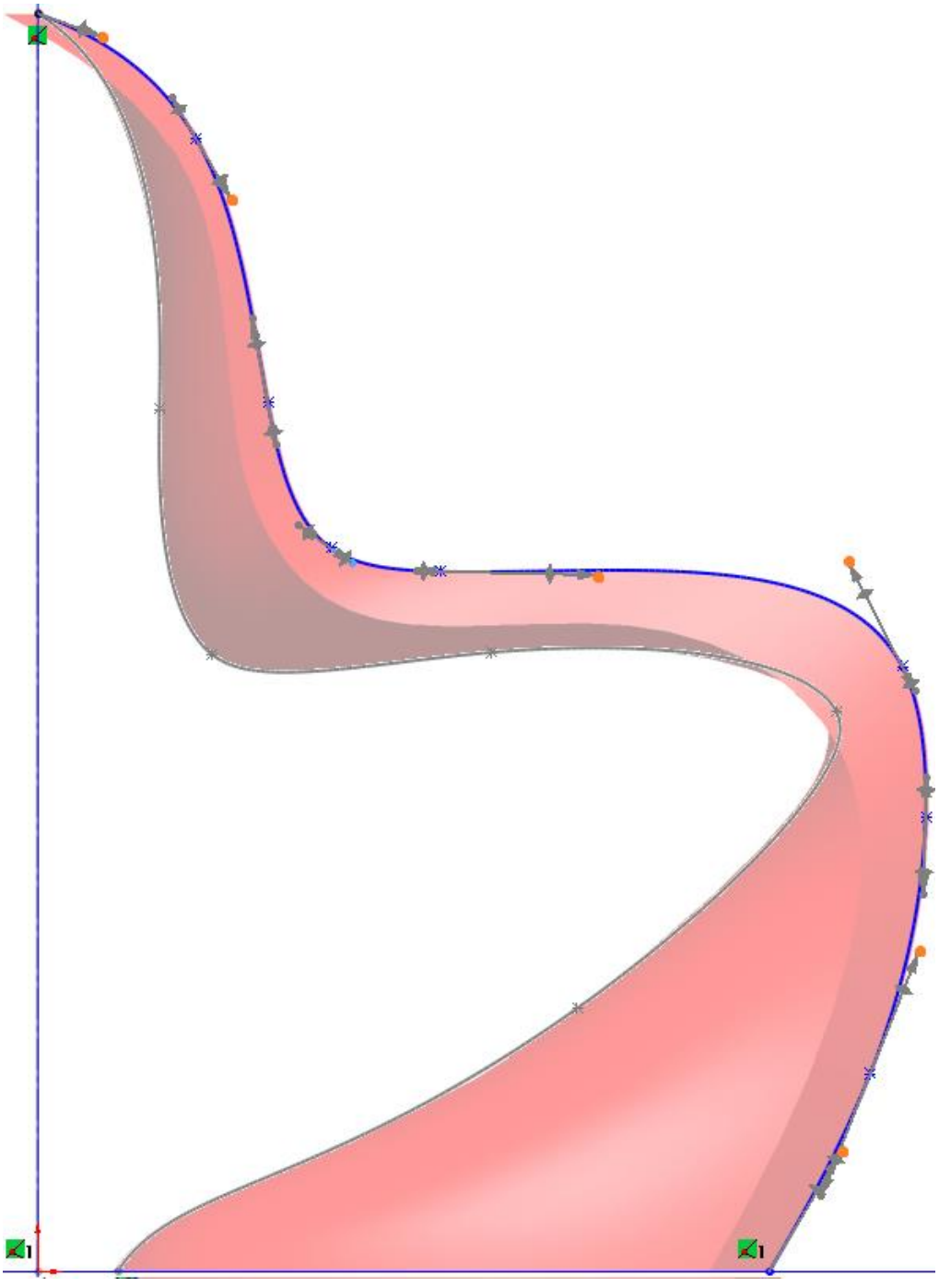
Click and drag the spline points to improve the shape of the curve

Change the direction of a spline point

Click on a spline point which you want to improve

The grey arrow of the Spline point appear

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

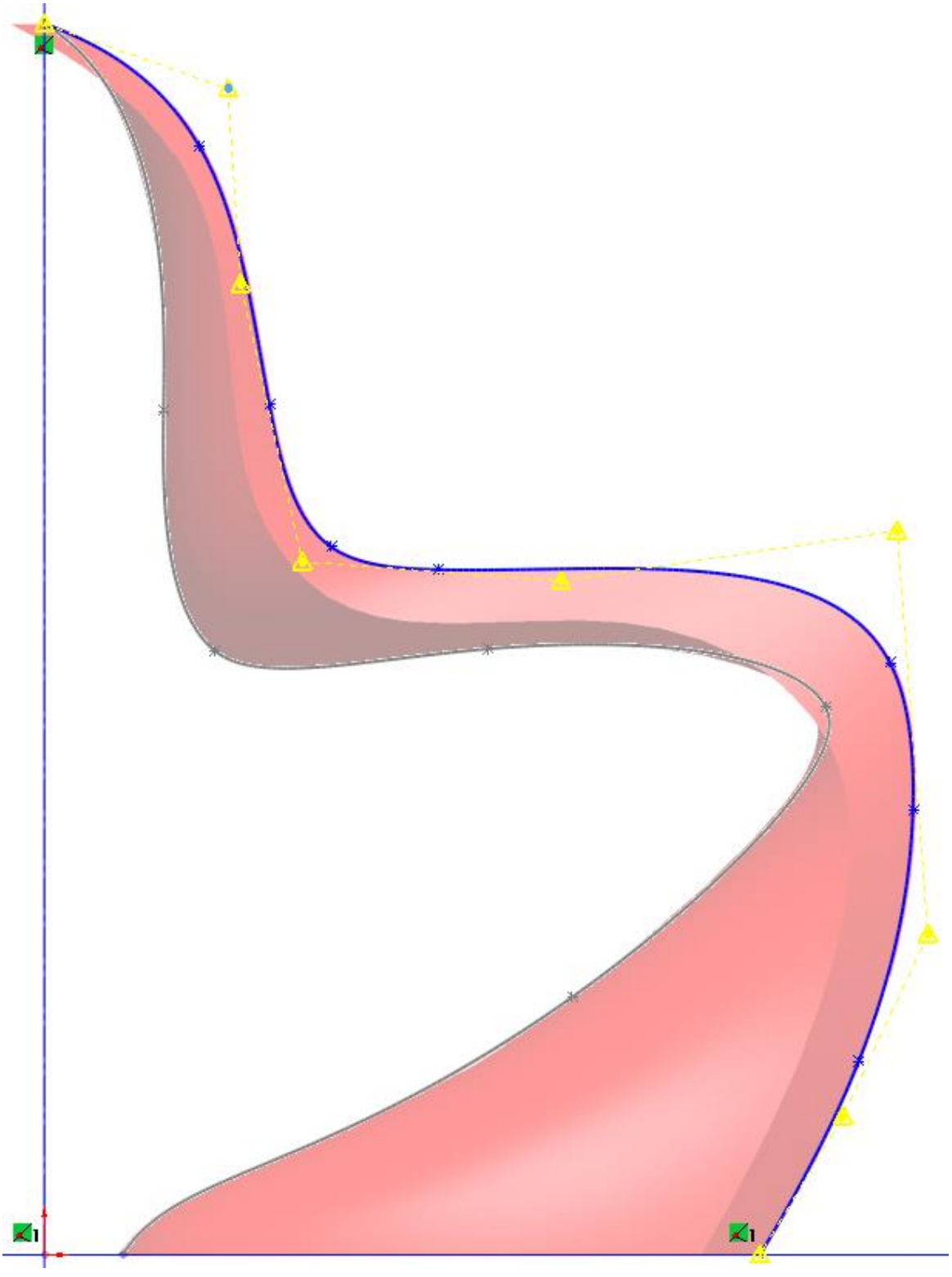


Improve the shape of the curve even more

If you're still not satisfied with the curve you can use the Display Control Polygon option

Click on the **Spline > Right click > Display Control Polygon** 

Click and drag one of the grey Polygon points to improve the shape even more

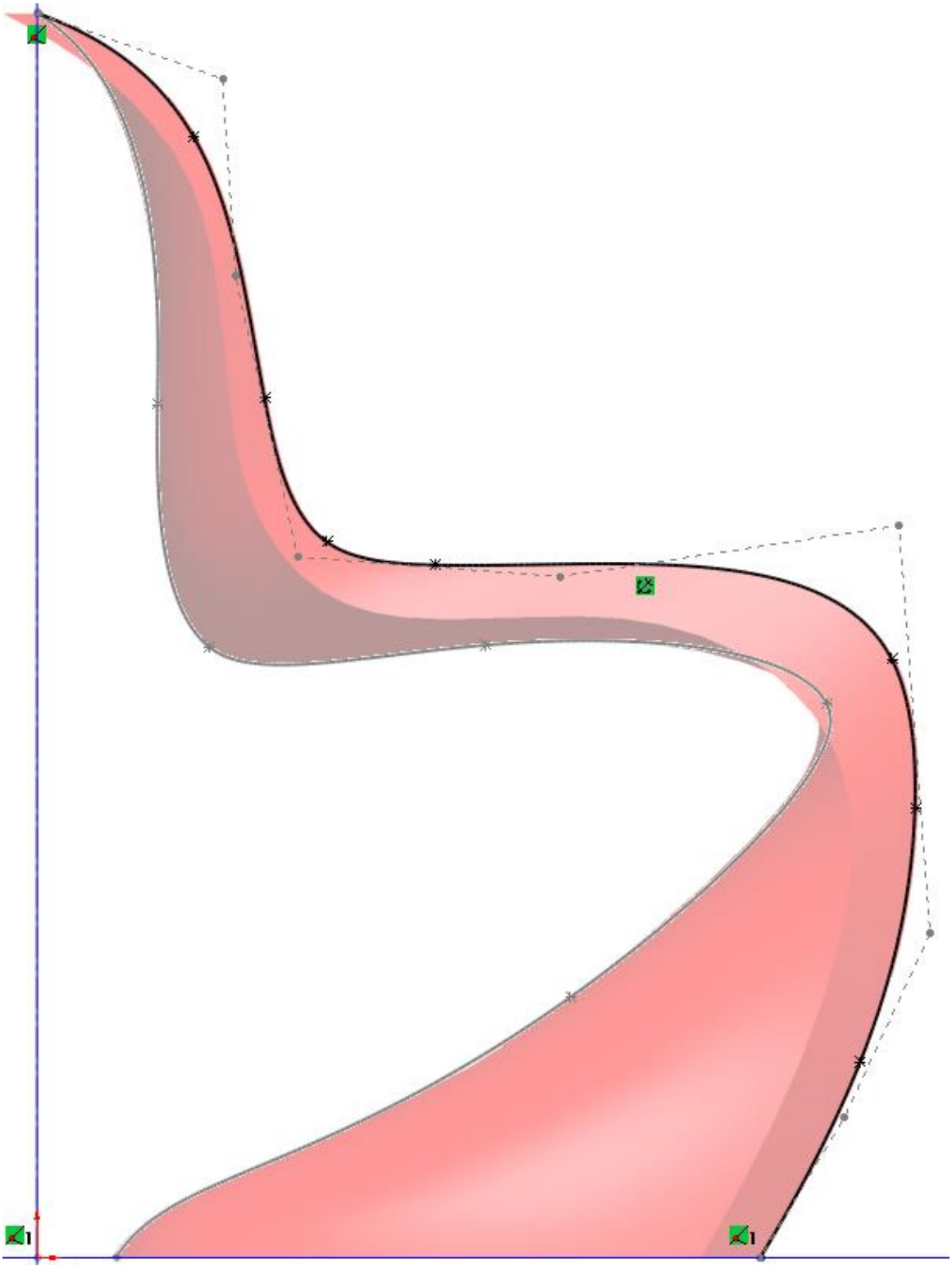


Fix the Spline

When you're satisfied with the curve, click on the Spline and select the **Fix button**

The color of the spline changed to black which means that it's fully defined

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



Create a 2D sketch

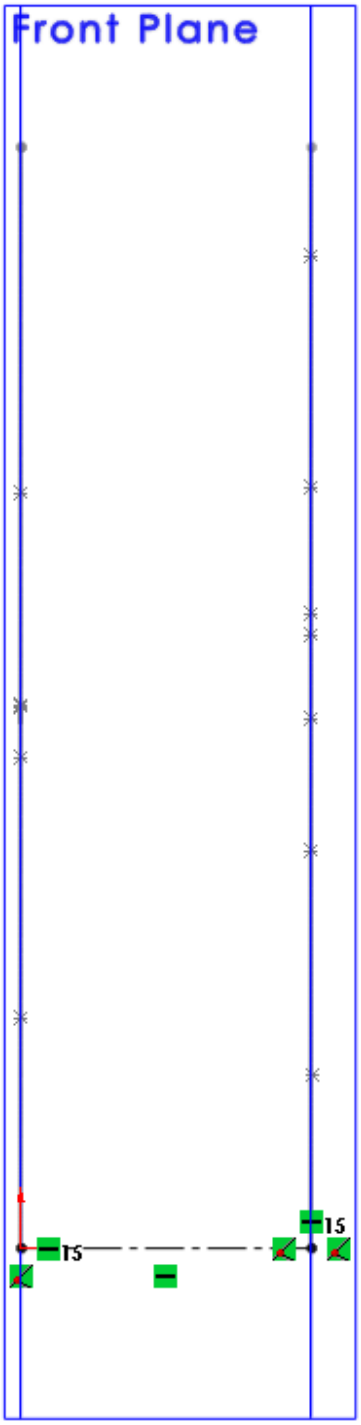
Select the Front Plane and create a sketch by clicking on the 2D Sketch icon

Draw a horizontal centerline

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

Draw a horizontal centerline that starts at the origin.

The centerline ends on Plane1 as shown in the picture

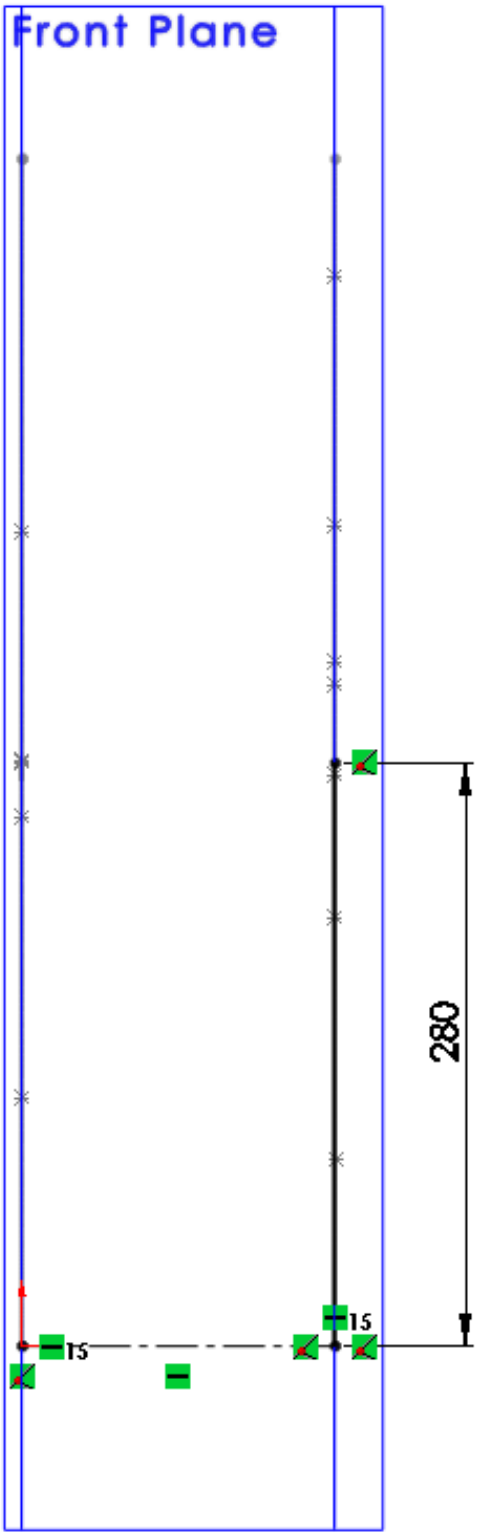


Draw a vertical line

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

Draw a vertical line that starts at the right end of the horizontal construction line

Change the length of the line into 280 mm by clicking at the dimension button



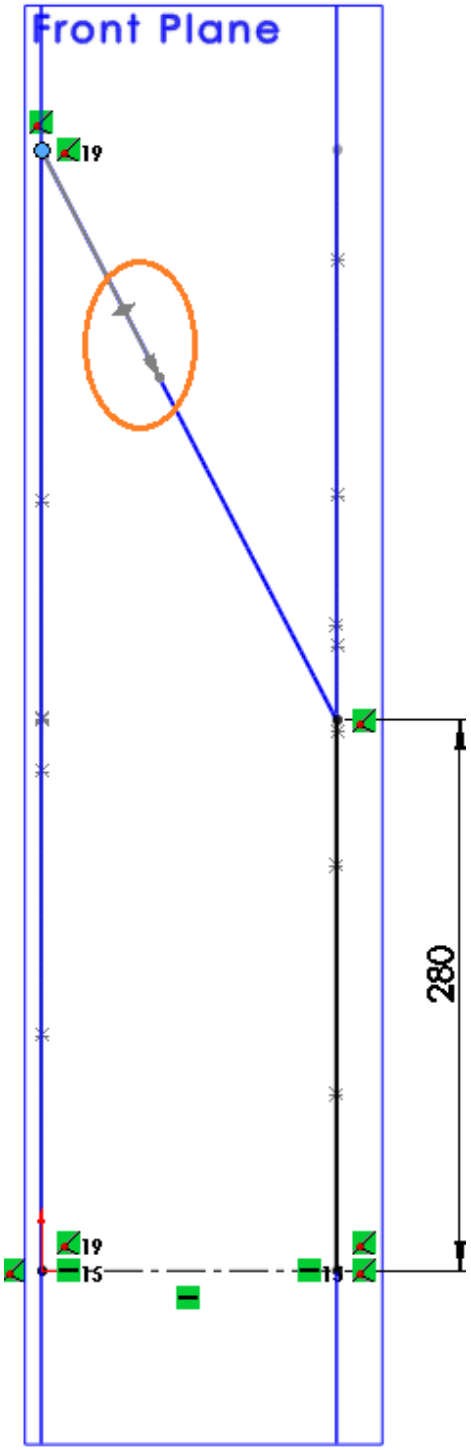
Draw a spline without midpoints

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

Start the spline at the upper point of the vertical line and ending at the top of the centerline of Sketch1 as shown in the picture

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



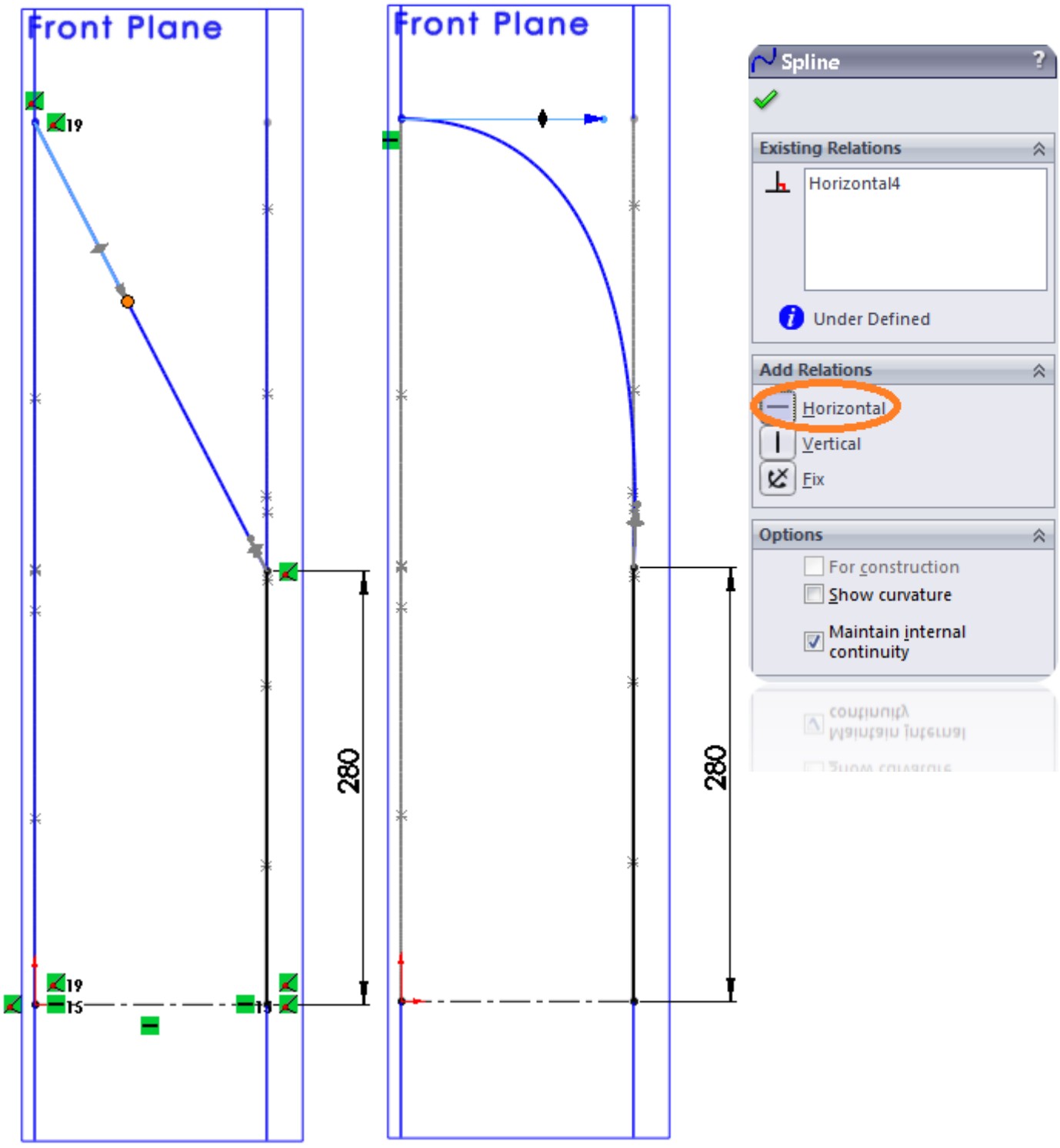
Add a horizontal tangency relation to the end of the spline

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 now perpendicular to the Right Plane

Click OK



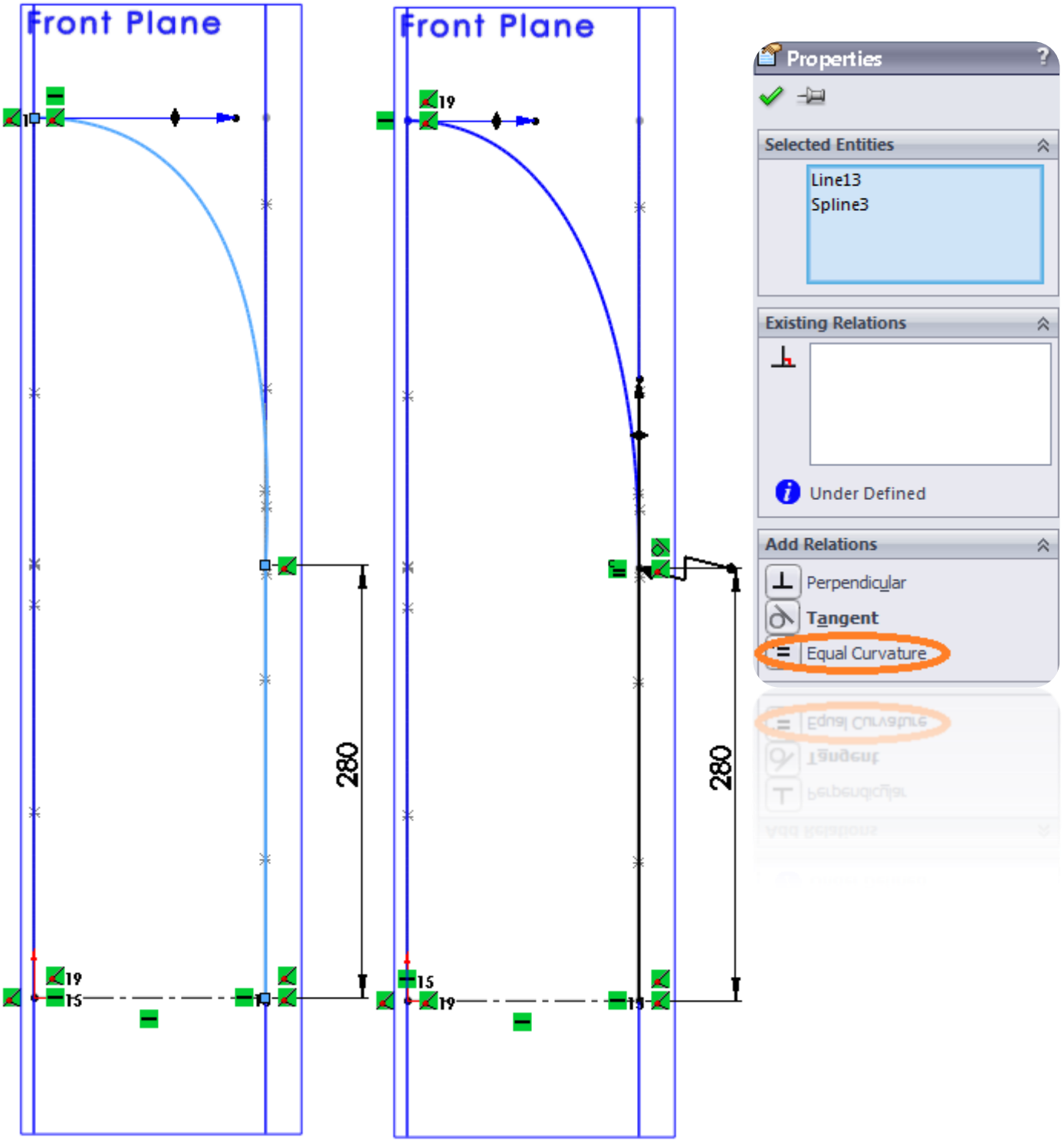
Add a curvature relation to the other end of the spline

Click at the spline, hold the Control button and select the vertical line as well

Select the Curvature relation in the Spline menu bar at the left side 

The transition between the line and spline is now curvature

Click OK 



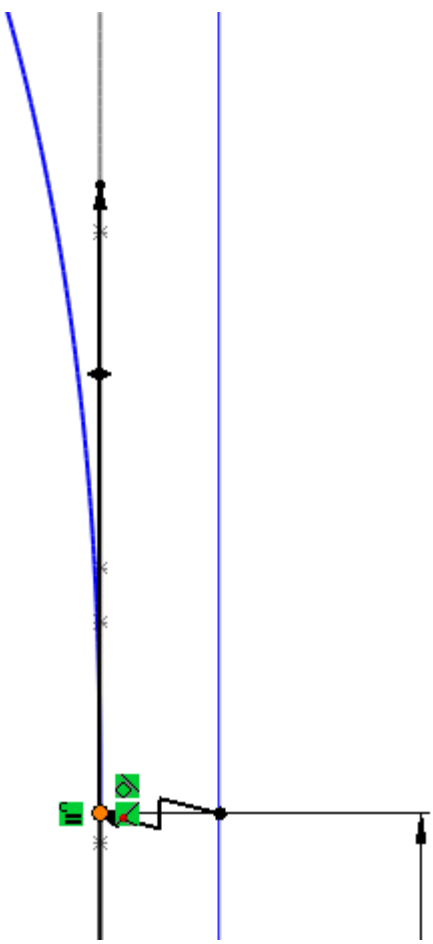
Change the dimension of the curvature relation

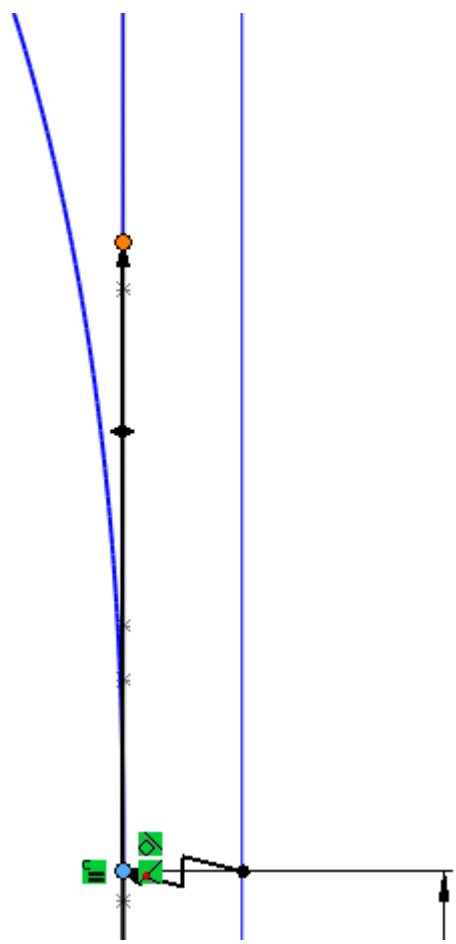
Click at the dimension button 

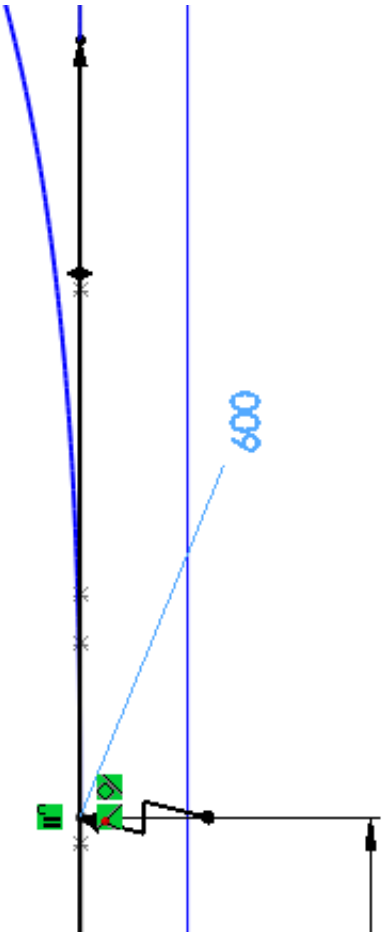
Select the starting point of the spline as shown in the first picture

Select the orange endpoint of the curvature arrow as shown in the second picture

Change the dimension into 600 mm as shown in the third picture







Change the dimension of the tangent relation

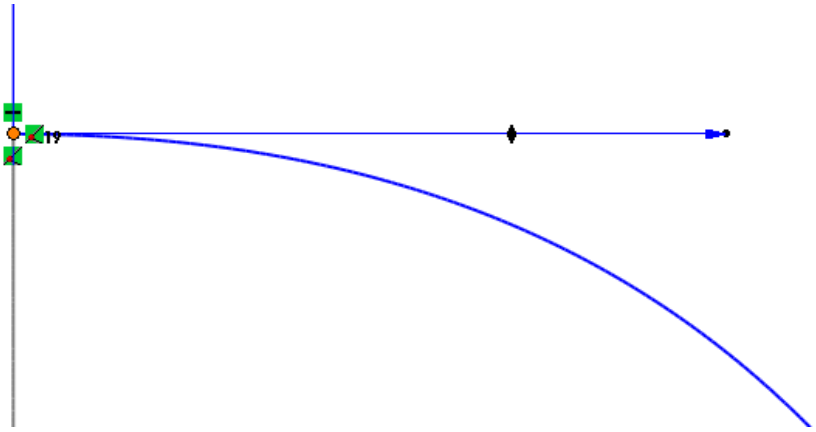
Click at the dimension button 

Select the starting point of the spline as shown in the first picture

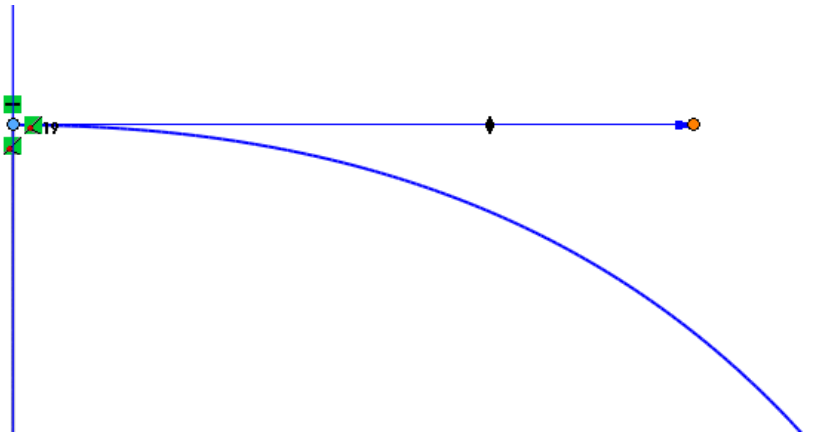
Select the orange endpoint of the tangent arrow as shown in the second picture

Change the dimension into 320 mm as shown in the third picture

1.



2.



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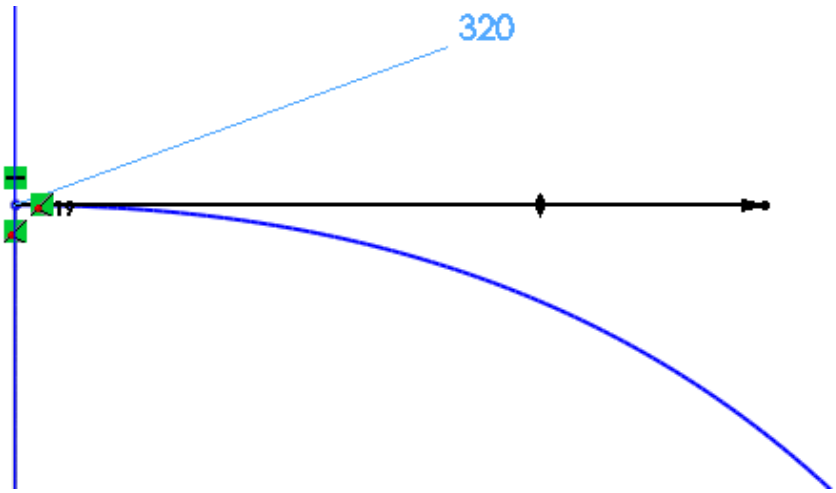
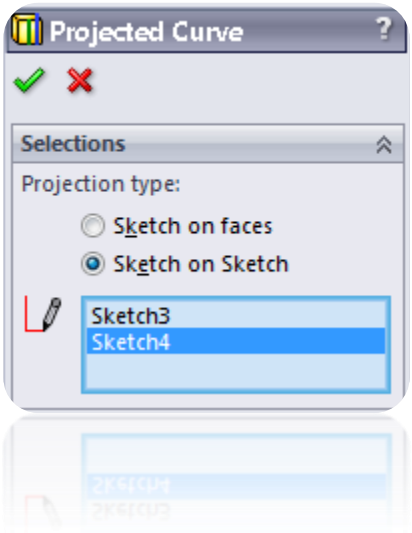


Create a projected curve

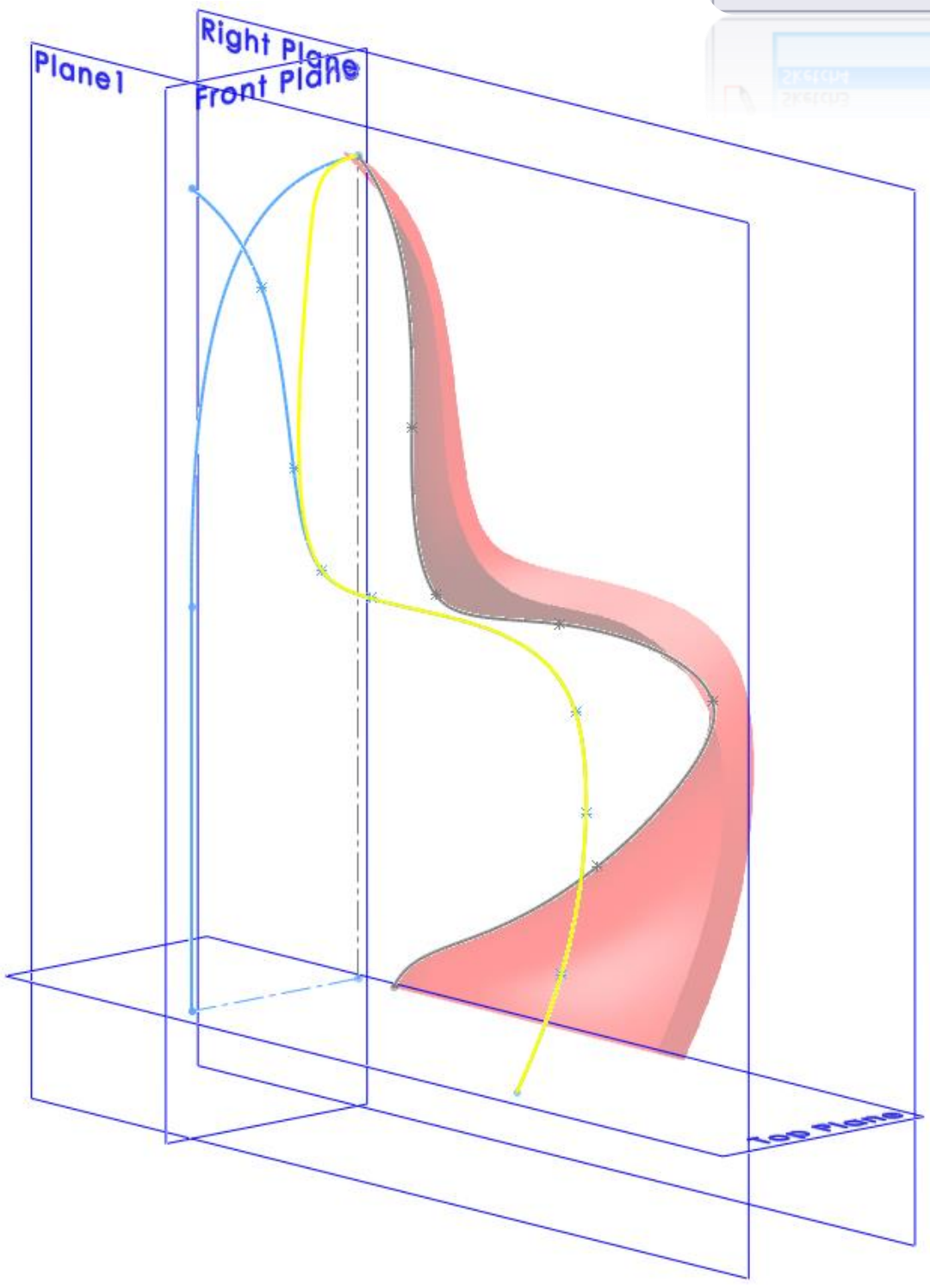


Go to: **Insert > Curve > Projected**

Select the **Sketch on Sketch** option

Select Sketch3 and Sketch4 as shown in the picture

Click OK



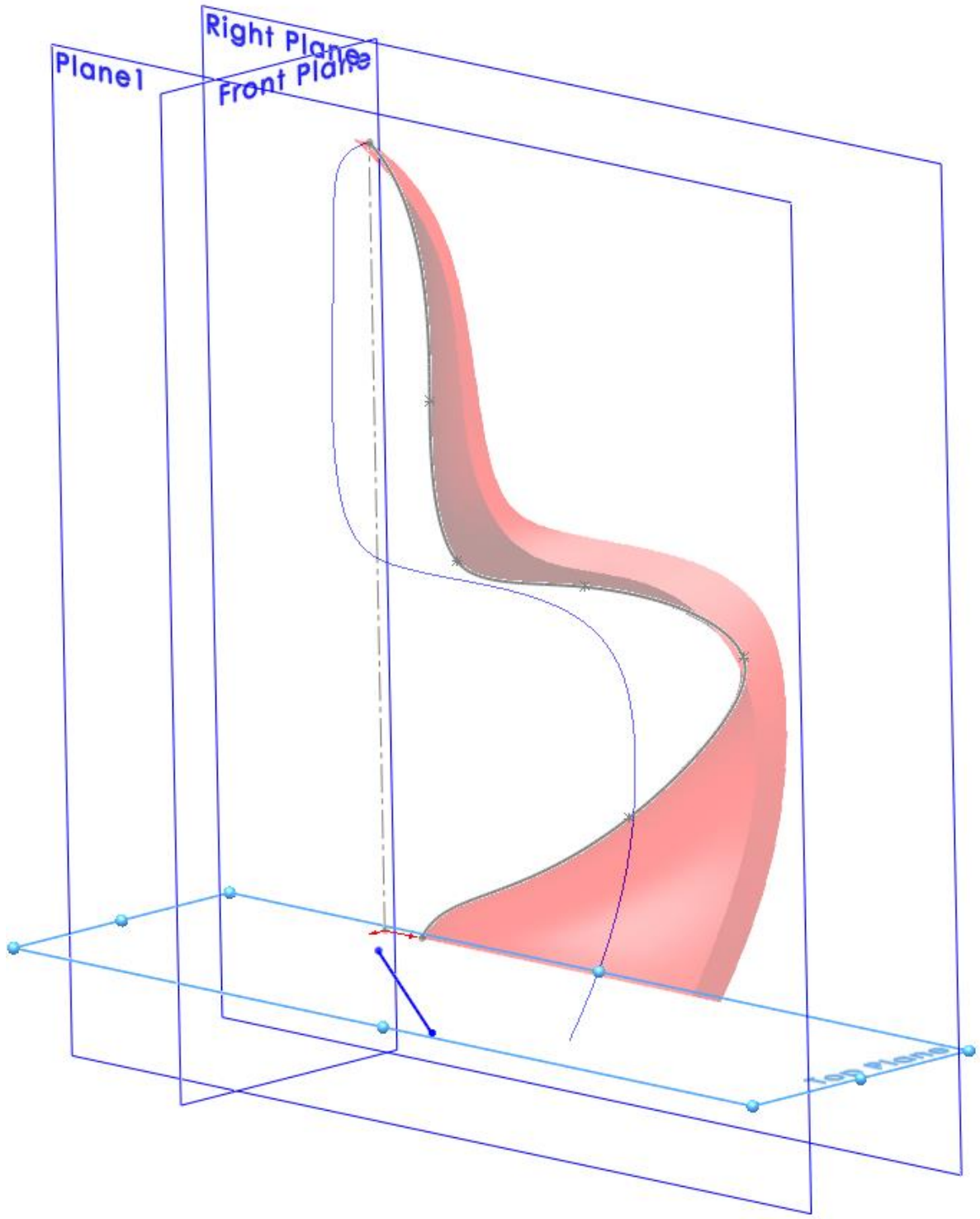
Create a 2D sketch on the Top Plane

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

Draw a spline

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

Draw a spline without any dimensions or connections as shown in the picture



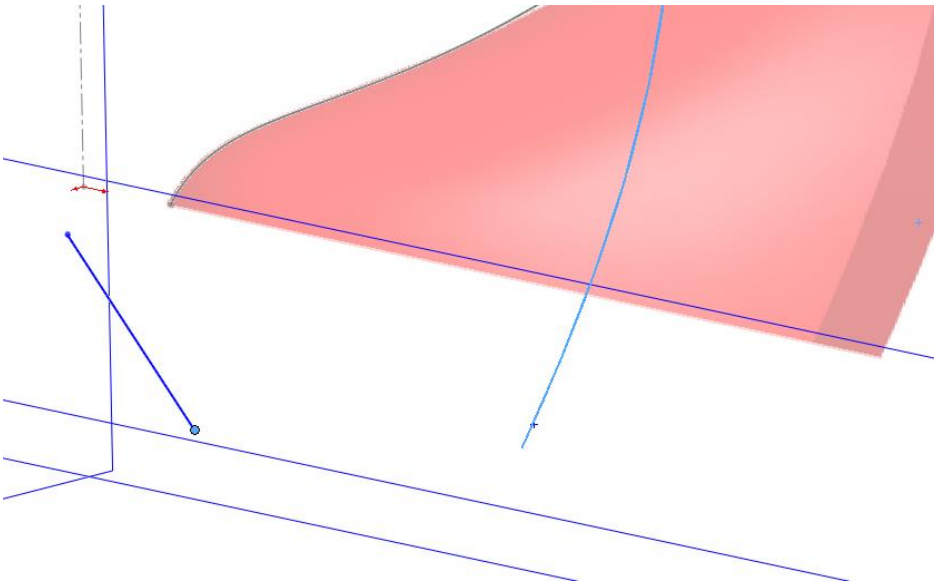
Connect the spline with the new Curve1

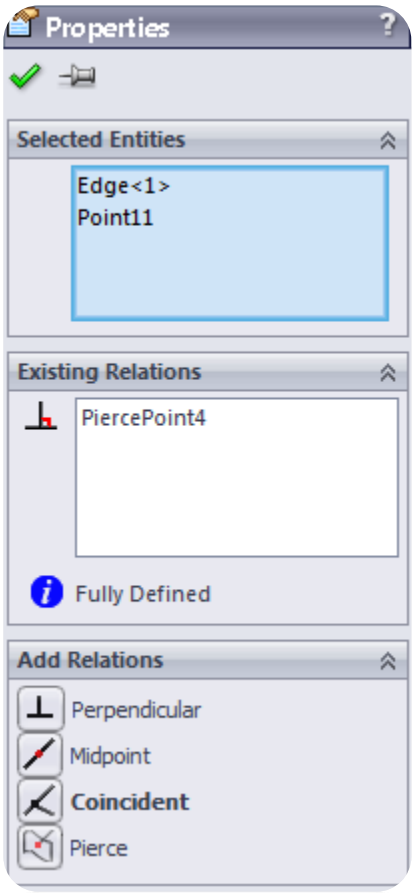
Select a spline point, hold the Control button and select the new Curve1

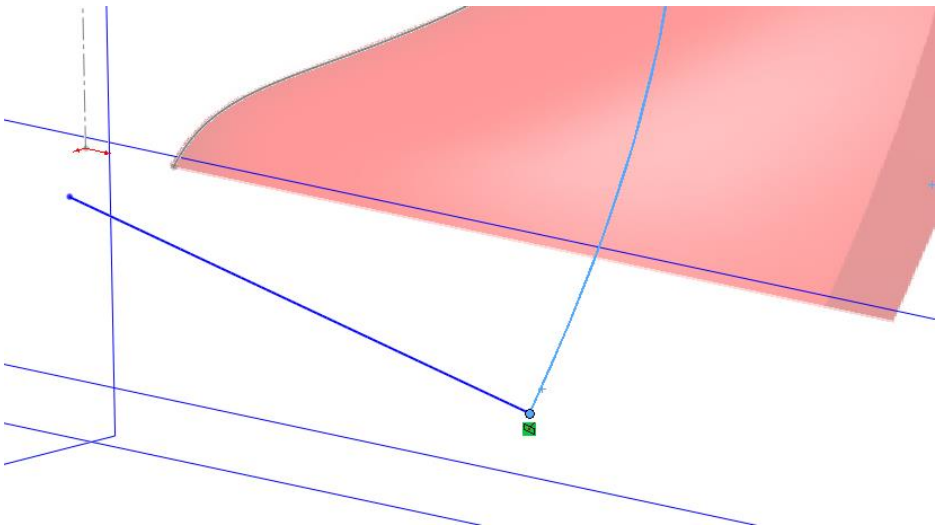
Select the **Pierce relation** in the Add relation menu bar at the left side 

The spline and Curve1 are now connected

Click OK 







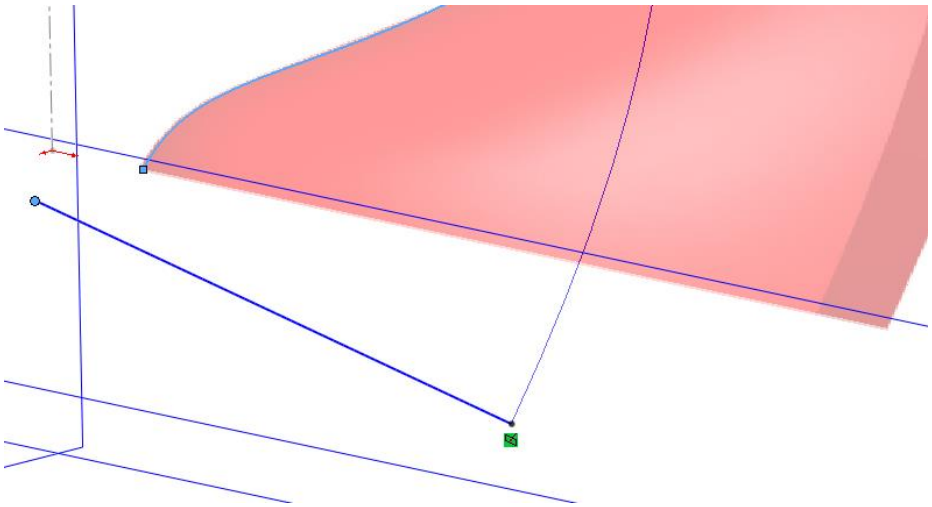
Connect the other end of the spline with Sketch2

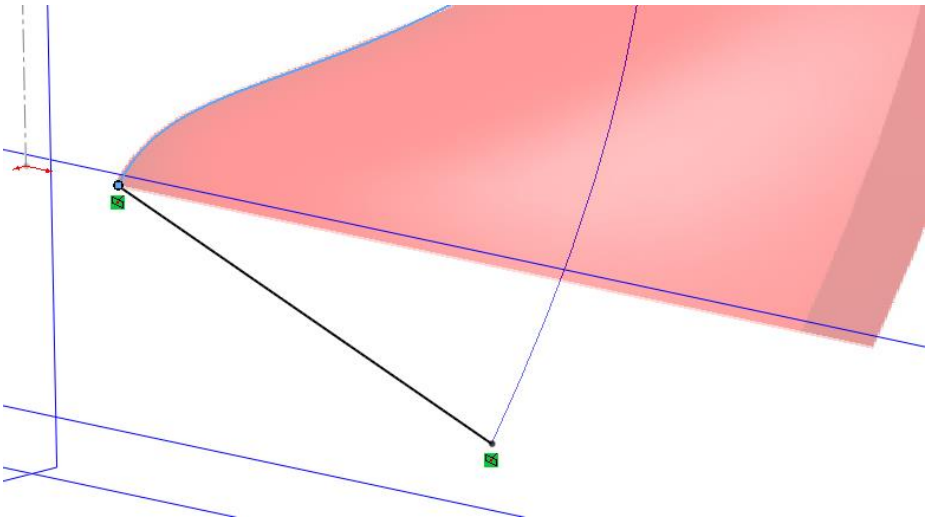
Select the other spline point, hold the Control button and select Sketch2

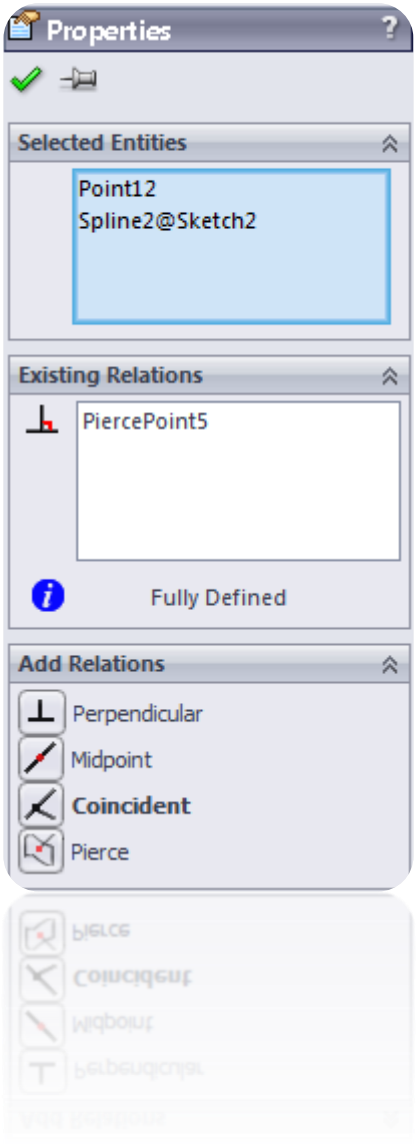
Select the **Pierce relation** in the Add relation menu bar at the left side 

The spline and Curve1 are now connected

Click OK 







Add a perpendicular relation to the end of the spline

Click at the cyan end point of the Spline as shown in the picture

The arrow of the Spline appears in grey

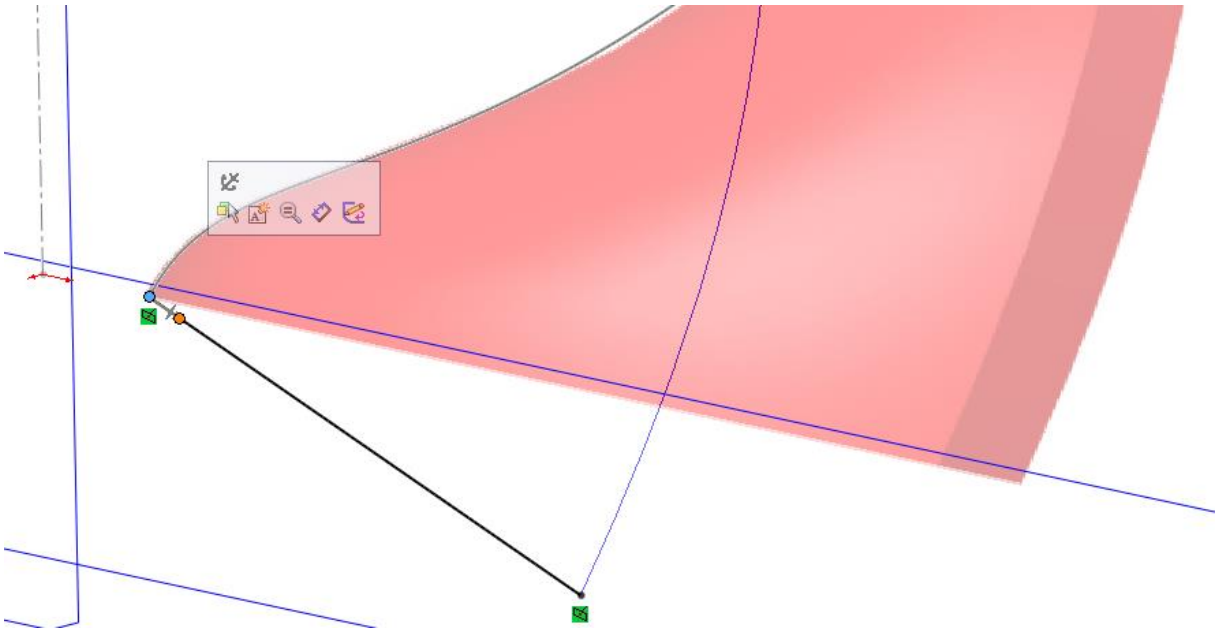
Click at the orange dot of the arrow as shown in the picture

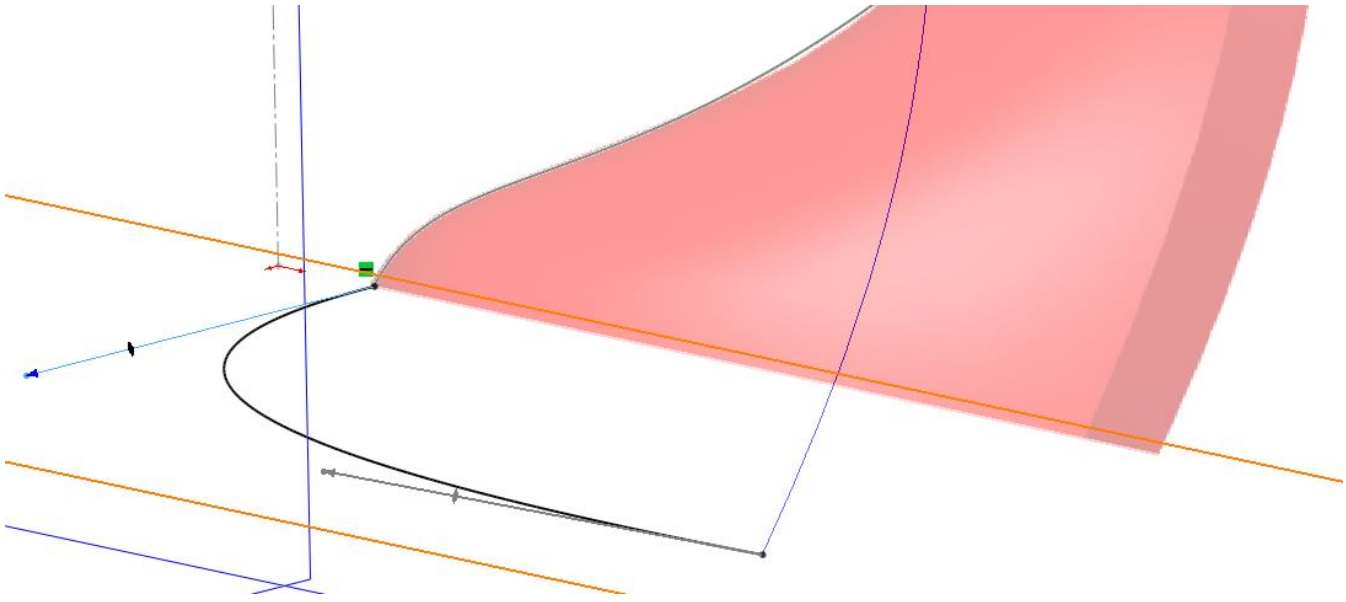
Select the Horizontal relation in the Spline menu bar at the left side

The endpoint of the spline is now perpendicular to the Right Plane

Click OK `

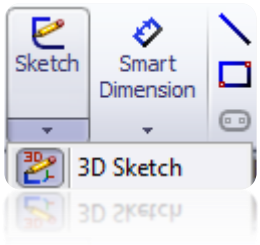
Click at the Sketch button in the upper right corner close the 2D Sketch





Create a 3D sketch

Click at the dropdown menu under the 2D Sketch icon



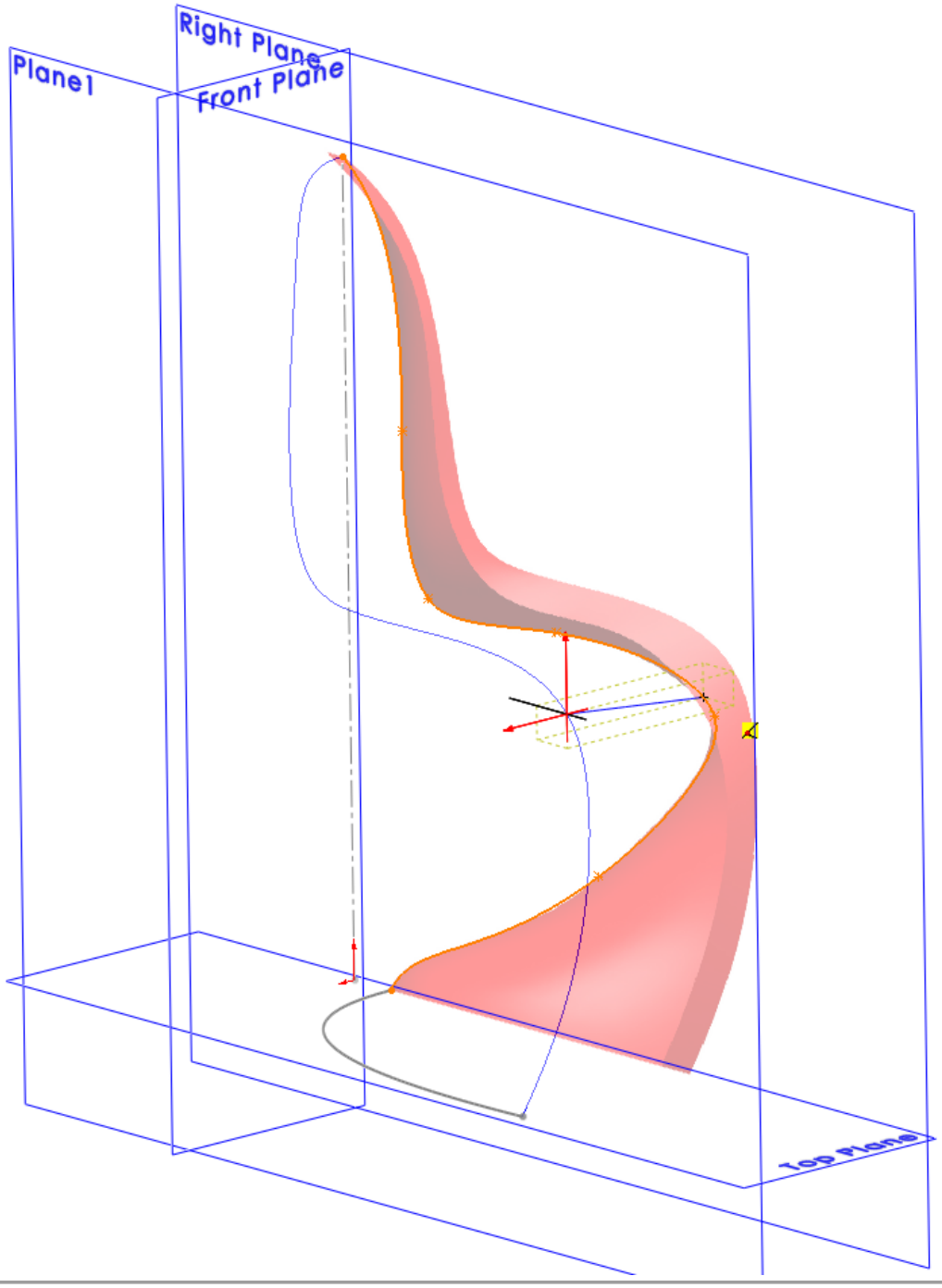
Select the 3D Sketch option 

Draw a spline

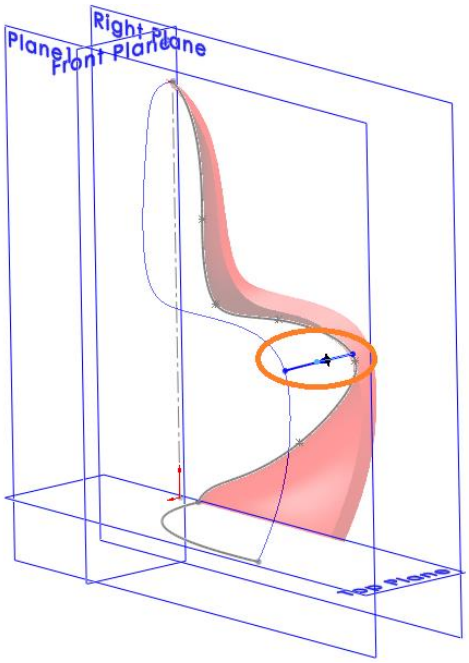
Go to **Tools > Sketch Entities > Spline** or click at the Spline icon 

Draw a 3d spline without any midpoints on the global position as shown in the picture

Connect the endpoints of the spline with Curve1 and Sketch2 



Add a perpendicular relation to the end of the spline



Click at the cyan end point of the Spline as shown in the picture

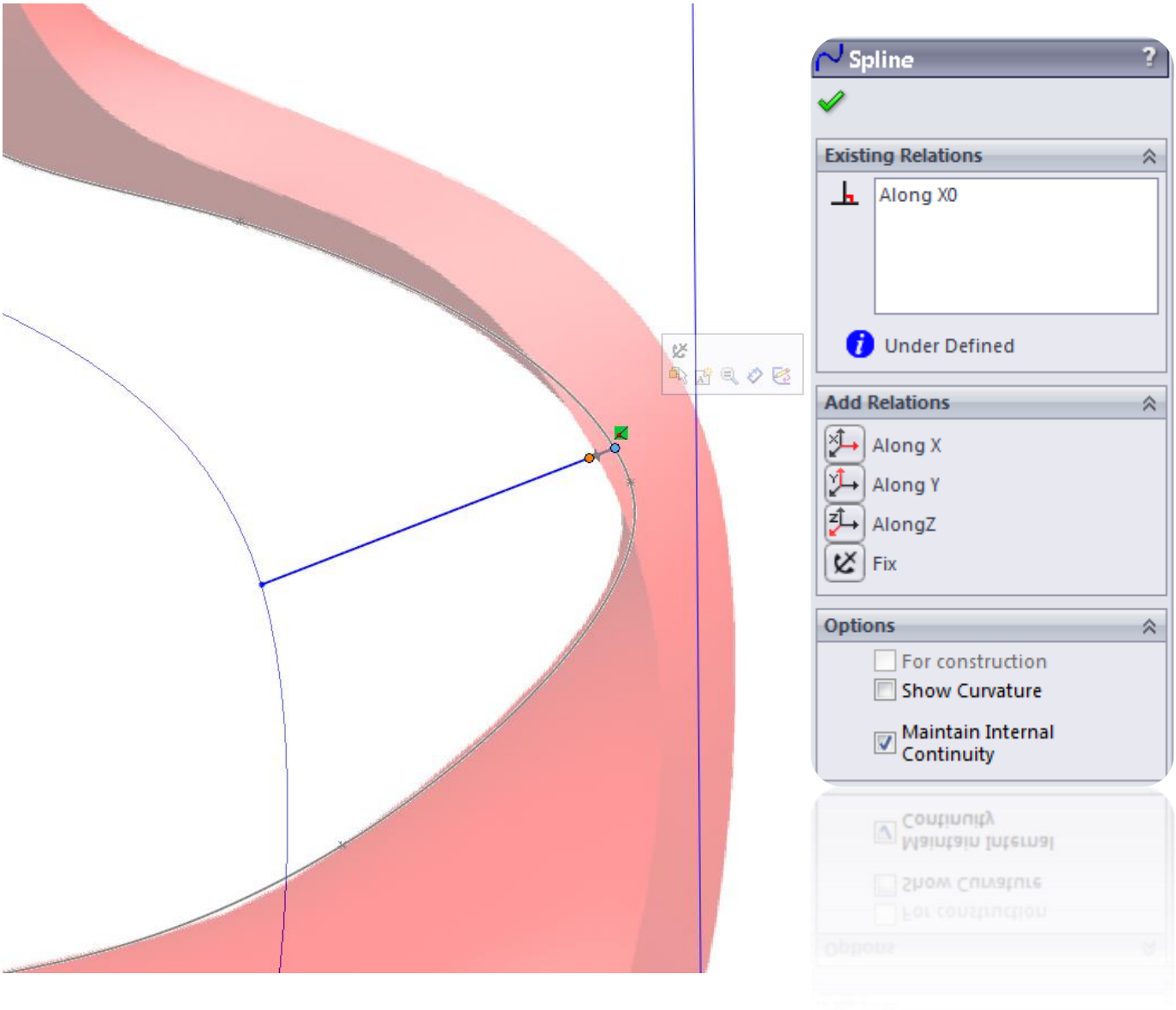
The arrow of the Spline appears in grey

Click at the orange dot of the arrow as shown in the picture

Select the **Along X** relation in the Spline menu bar at the left side

The endpoint of the spline is now perpendicular to the Right Plane

Click OK

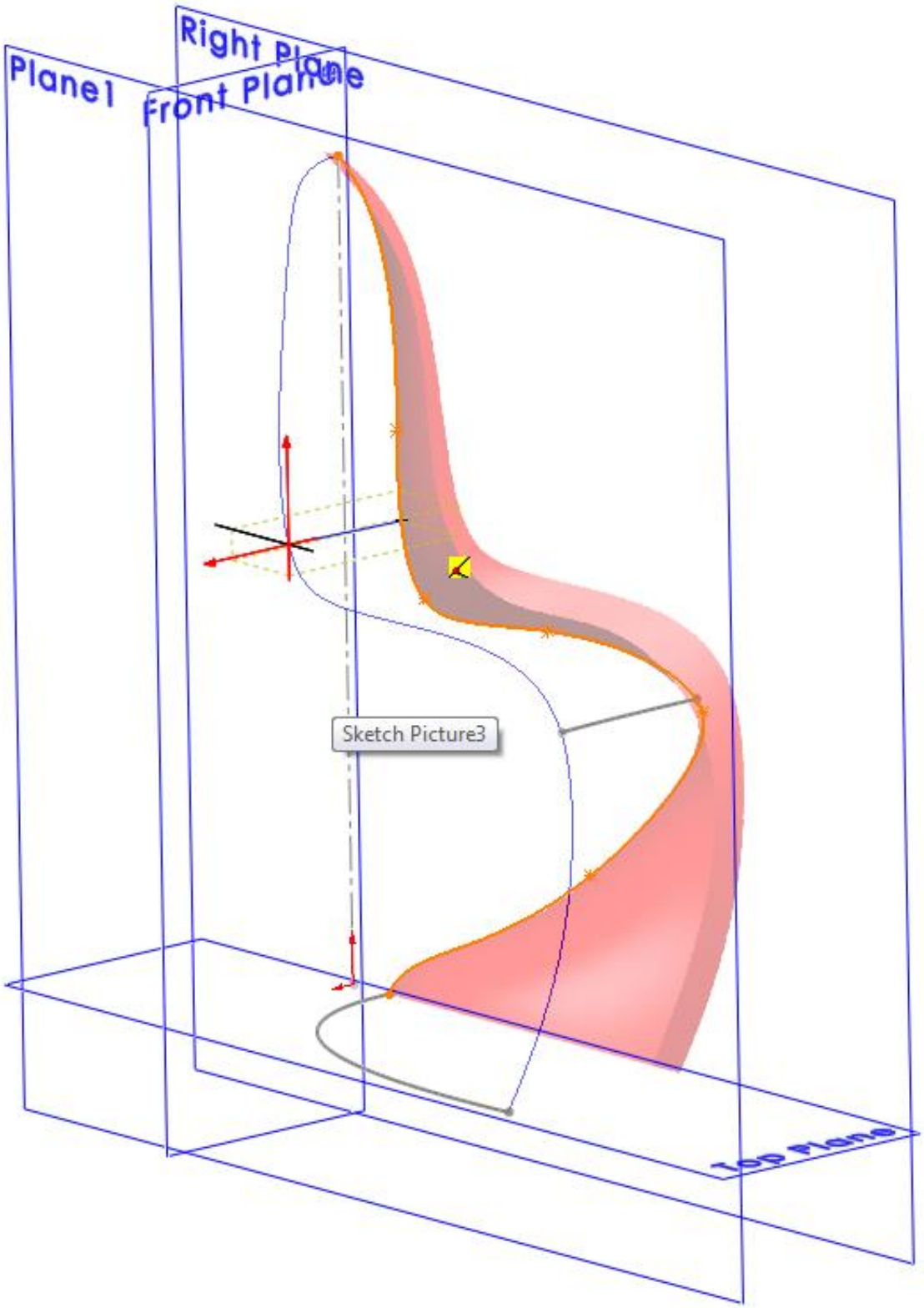


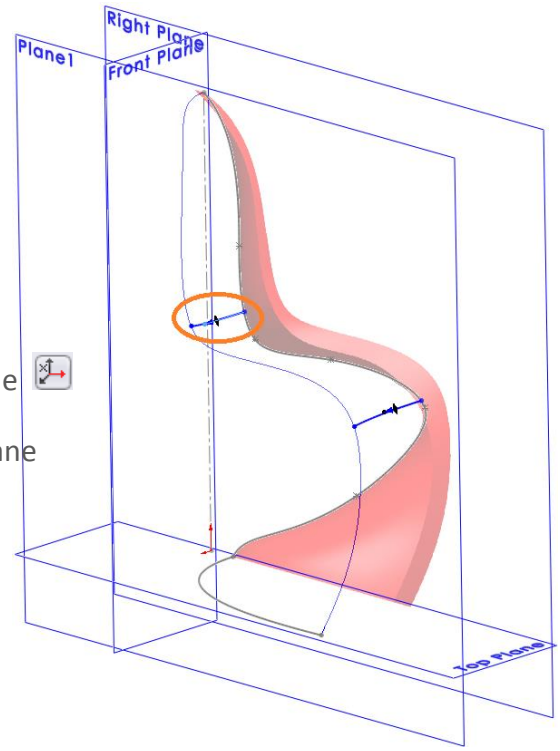
Draw another spline

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

Draw a 3d spline without any midpoints on the global position as shown in the picture

Connect the endpoints of the spline with Curve1 and Sketch2





Add a perpendicular relation to the end of the spline

Click at the cyan end point of the Spline as shown in the picture

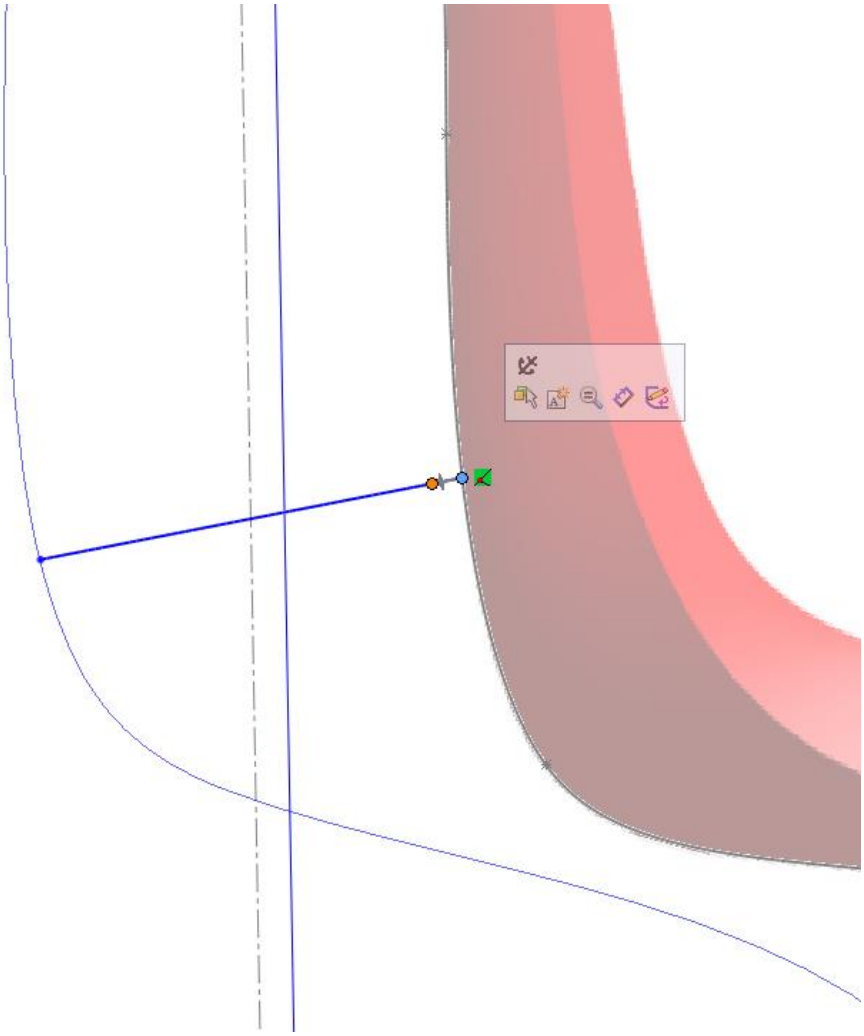
The arrow of the Spline appears in grey

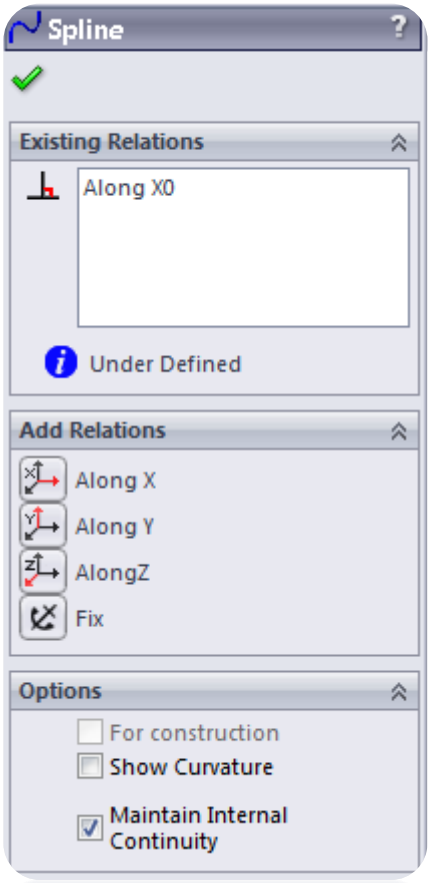
Click at the orange dot of the arrow as shown in the picture

Select the **Along X** relation in the Spline menu bar at the left side

The endpoint of the spline is now perpendicular to the Right Plane

Click OK



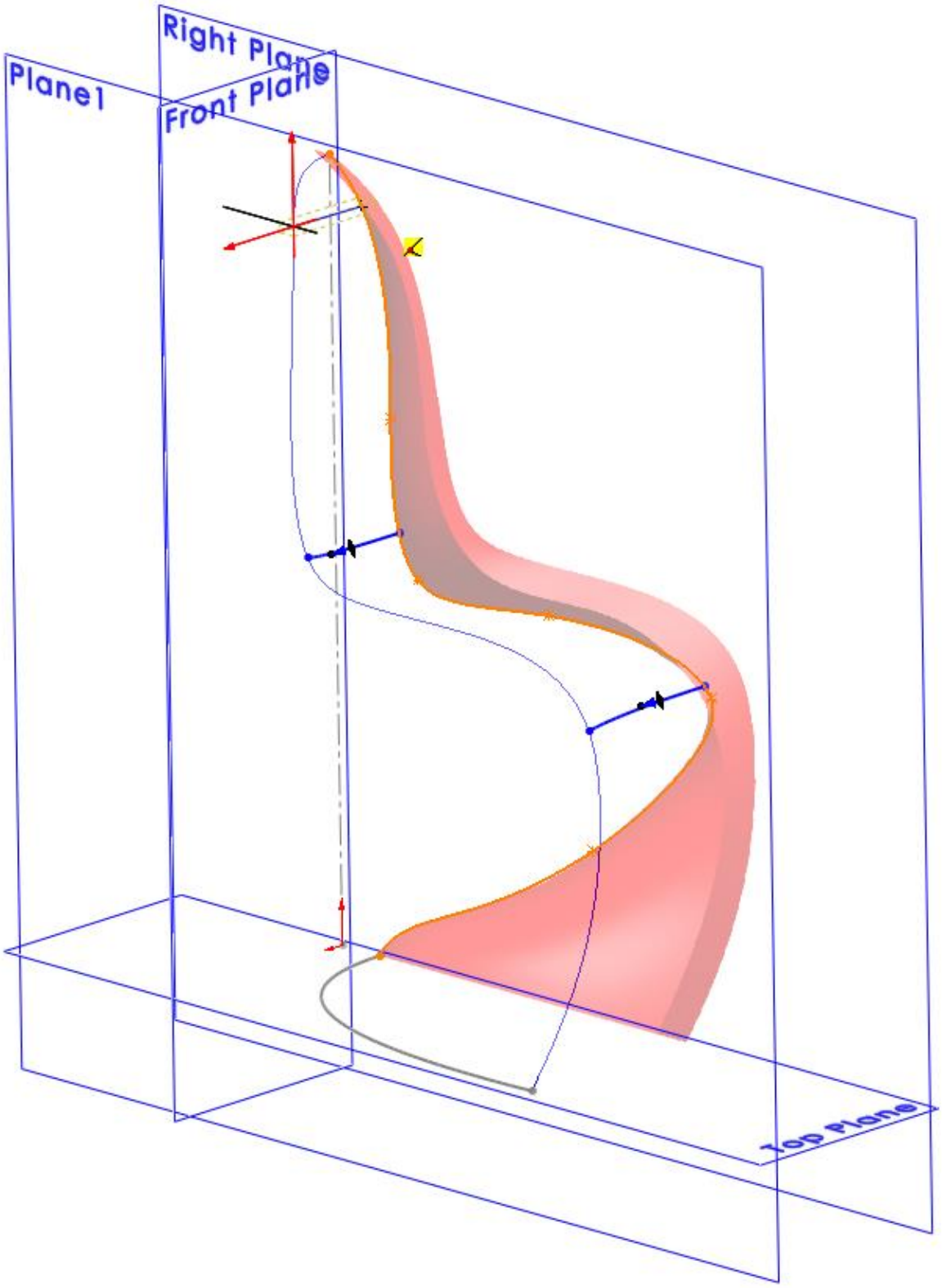


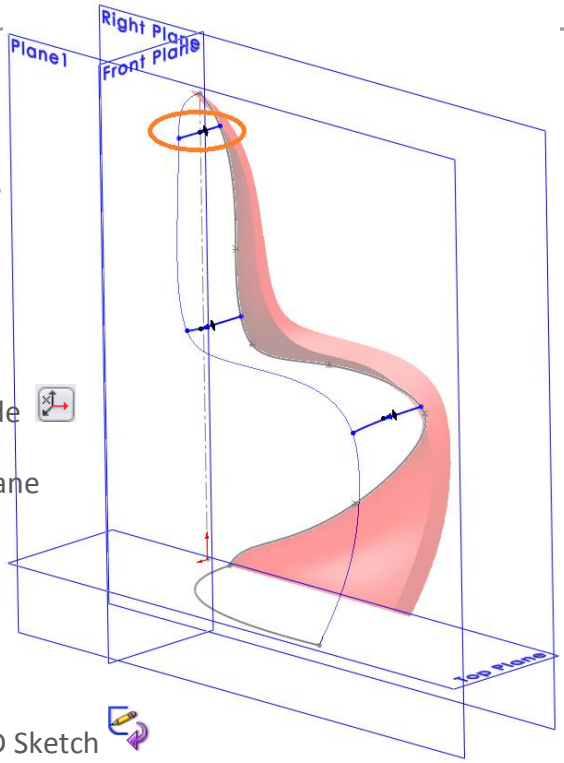
Draw another spline

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

Draw a 3d spline without any midpoints on the global position as shown in the picture

Connect the endpoints of the spline with Curve1 and Sketch2





Add a perpendicular relation to the end of the spline

Click at the cyan end point of the Spline as shown in the picture

The arrow of the Spline appears in grey

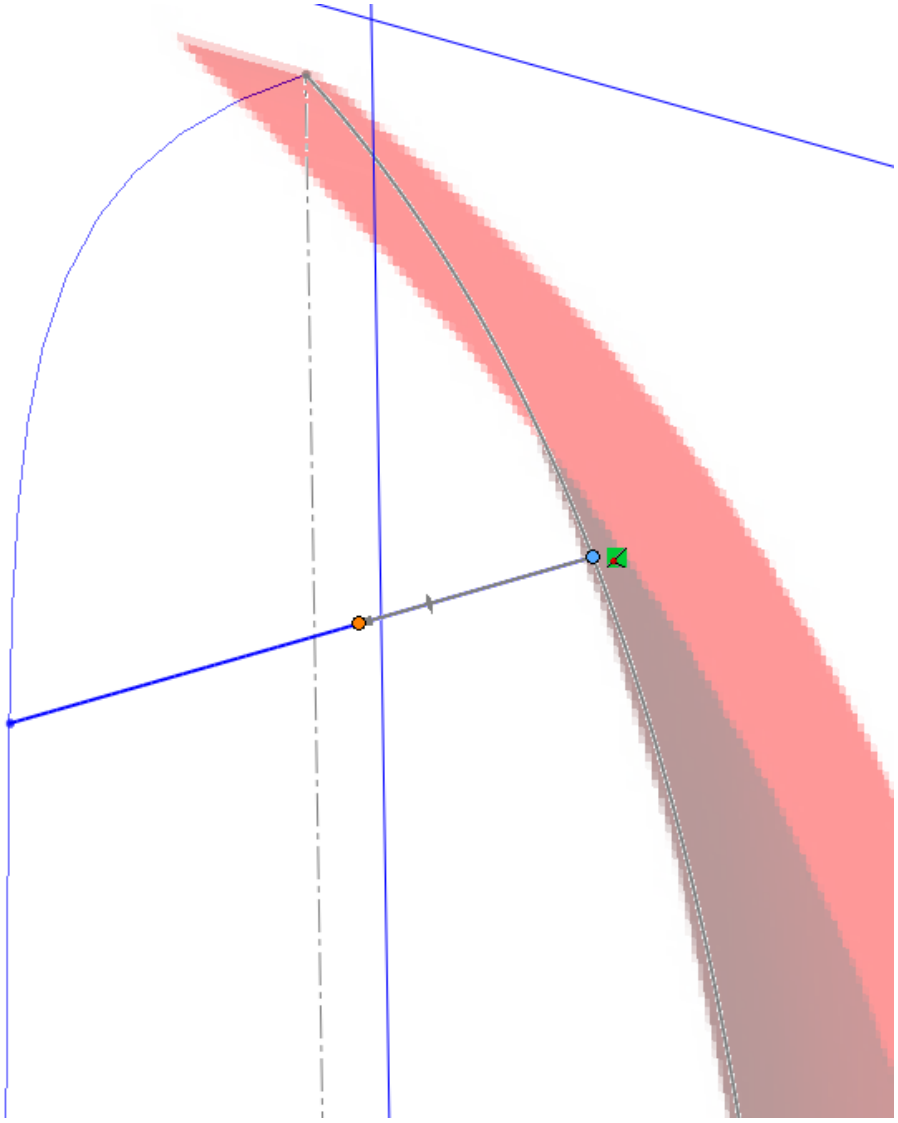
Click at the orange dot of the arrow as shown in the picture

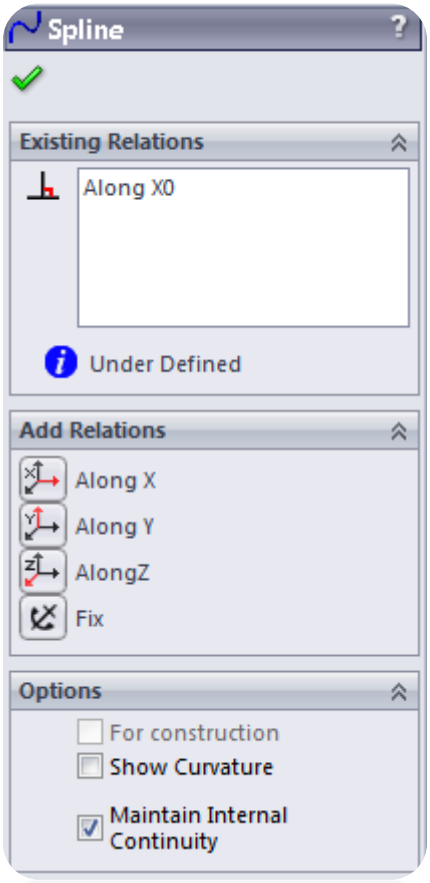
Select the **Along X** relation in the Spline menu bar at the left side

The endpoint of the spline is now perpendicular to the Right Plane

Click OK

Click at the Sketch button in the upper right corner close the 3D Sketch





Create a Surface Loft

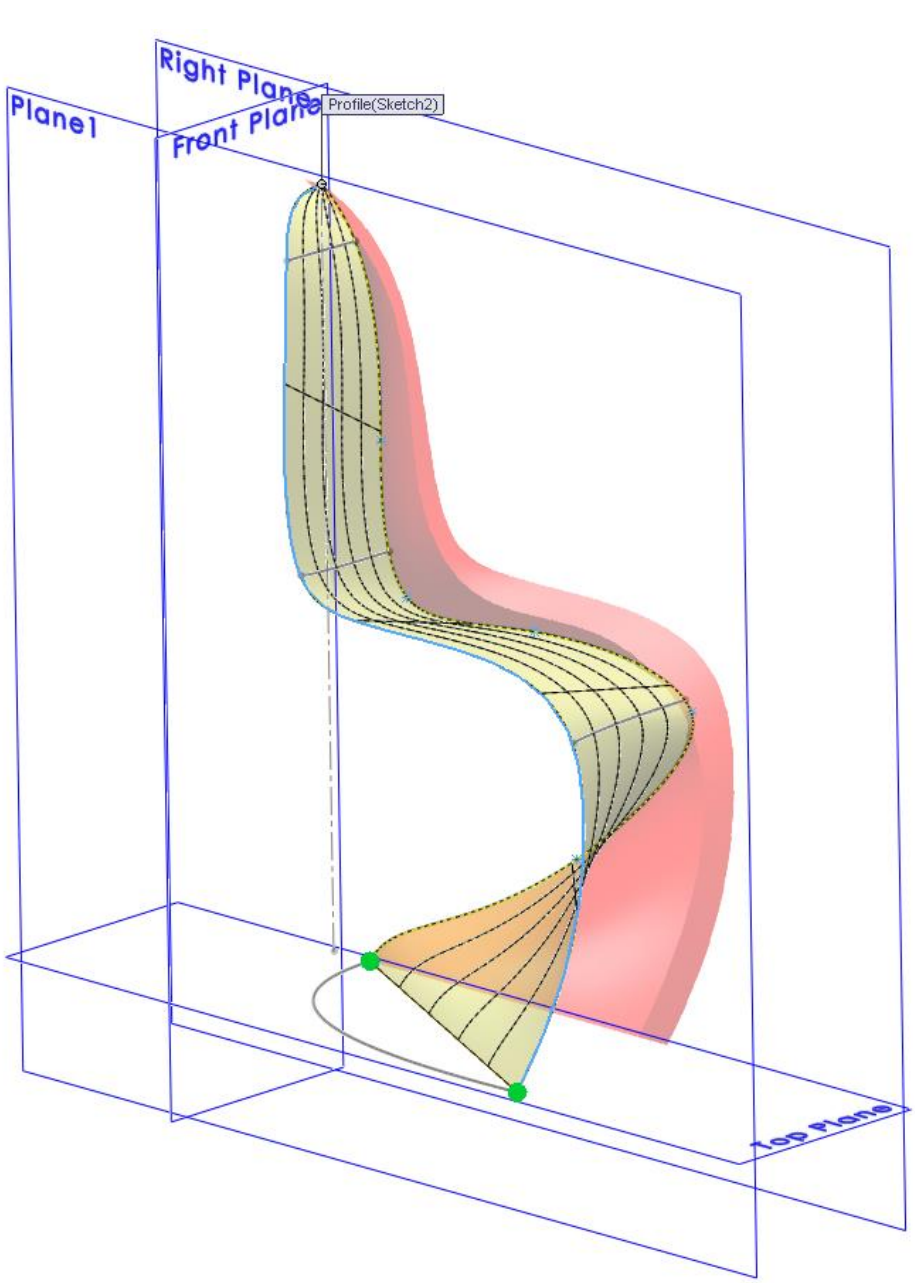
Go to **Insert > Surface > Loft** or click at the Surface icon

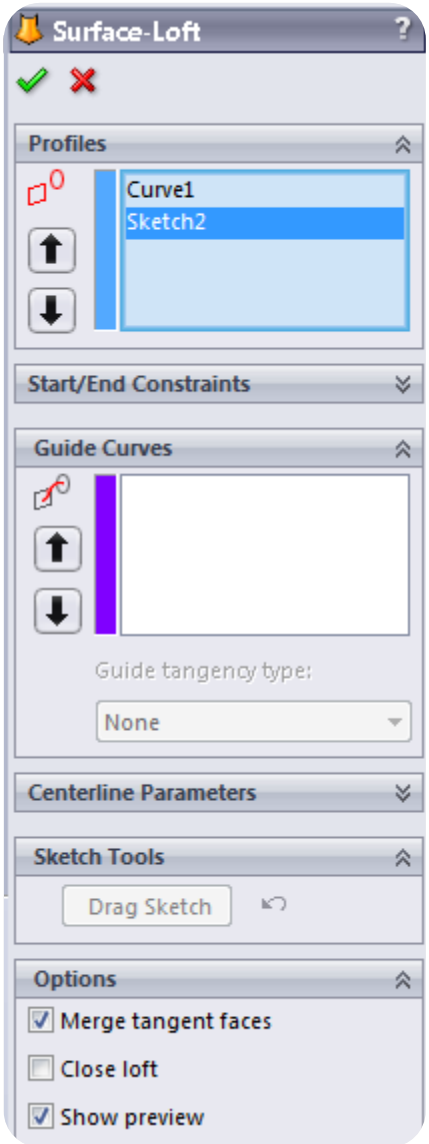
Click in the Profiles box

Select Curve1 and Sketch 2 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





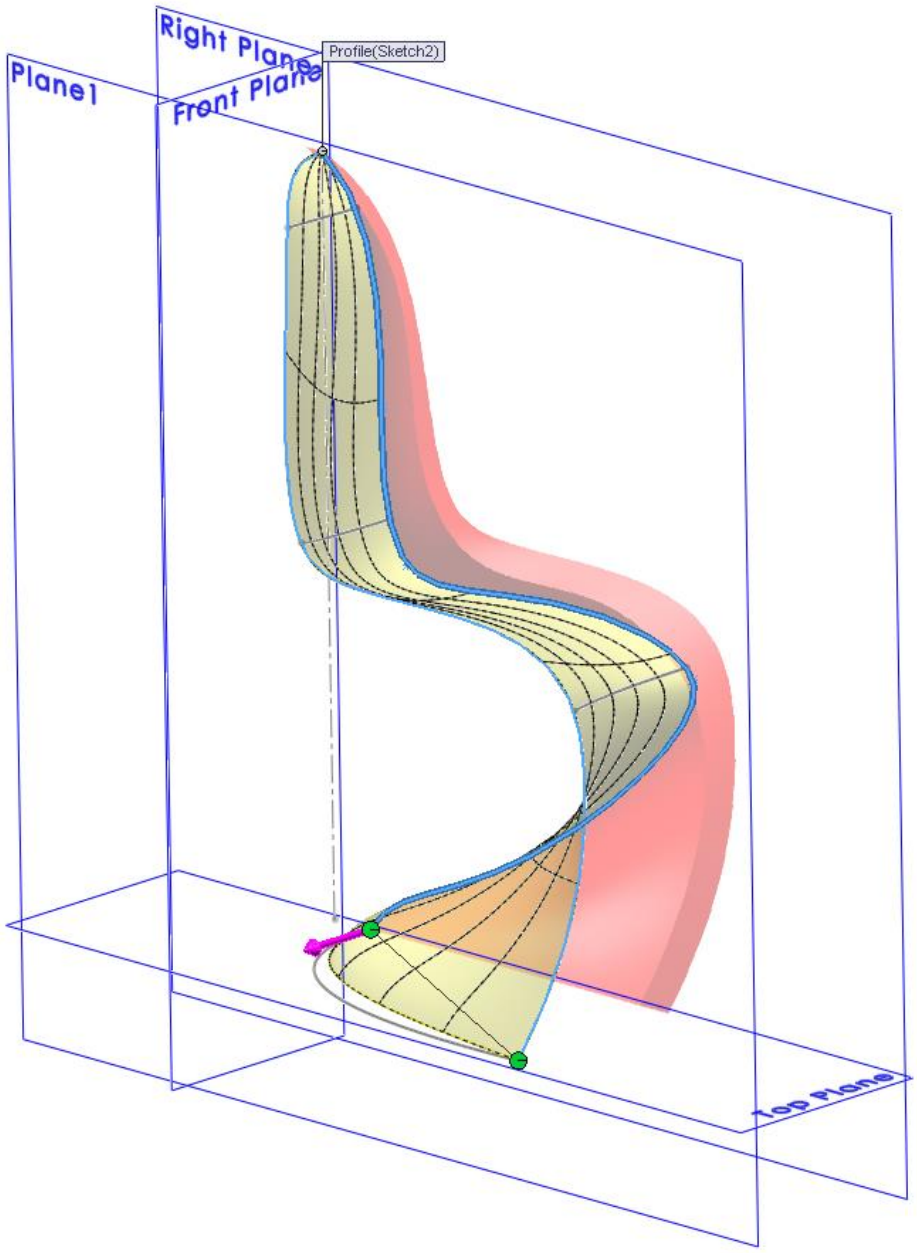
Make the loft surface perpendicular to the Right Plane

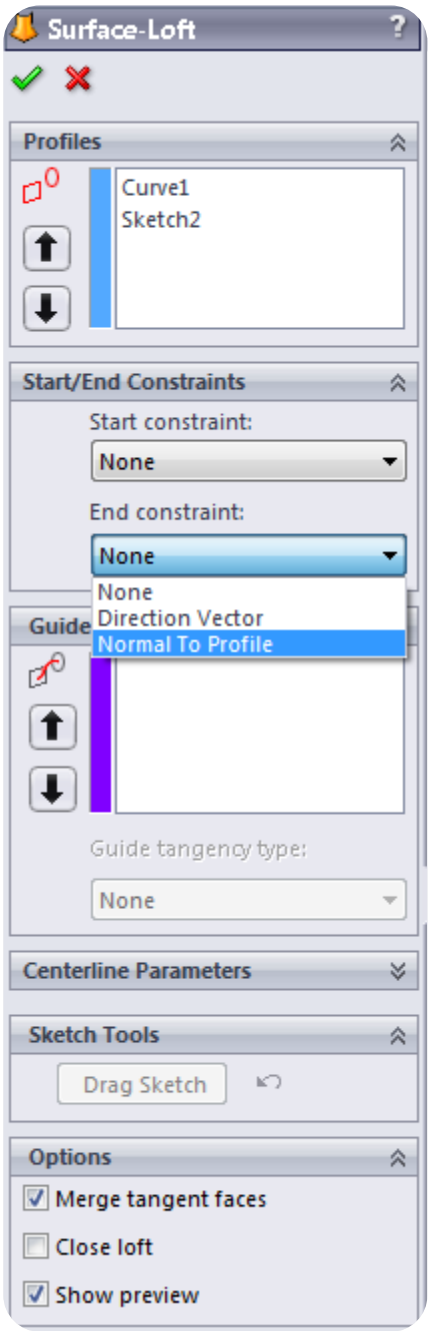
Click on Sketch2 in the Profiles box

Click at the arrow of the dropdown menu called **"Start/End Constraints"**

Click at the None button under **"End constraint"**

Select the **"Normal To Profile"** option as shown in the picture



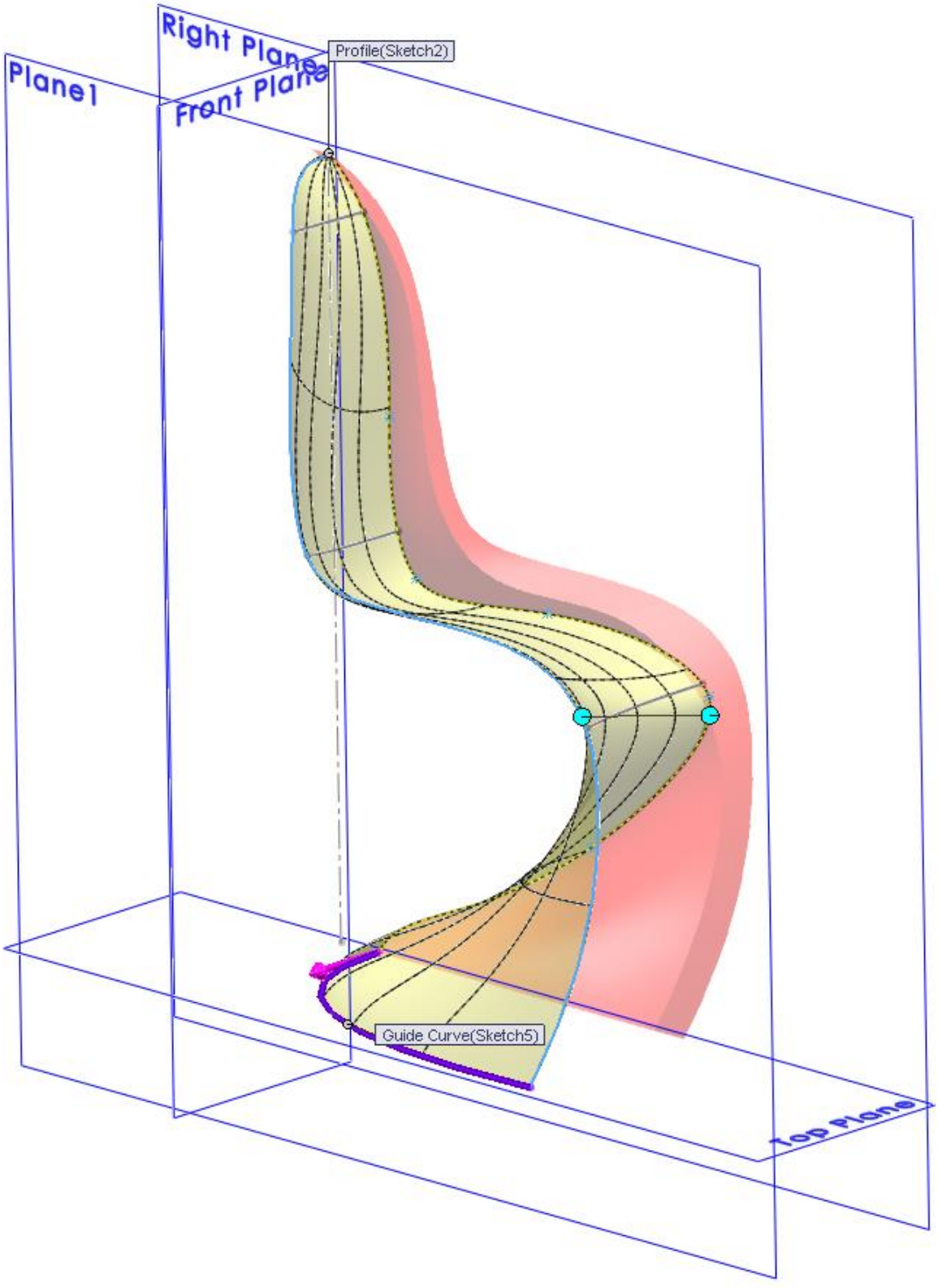


Add three Guide Curves to control the shape of the Surface Loft

Click in the Guide Curves box 

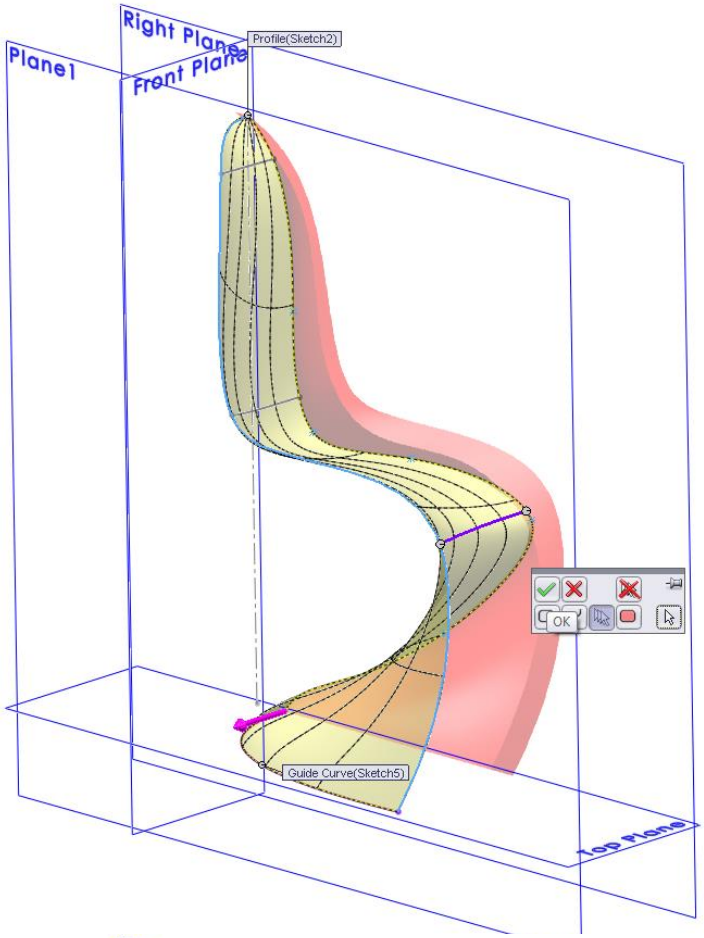
Select Sketch5 as shown in the picture

Guide curves influence: To Next Guide



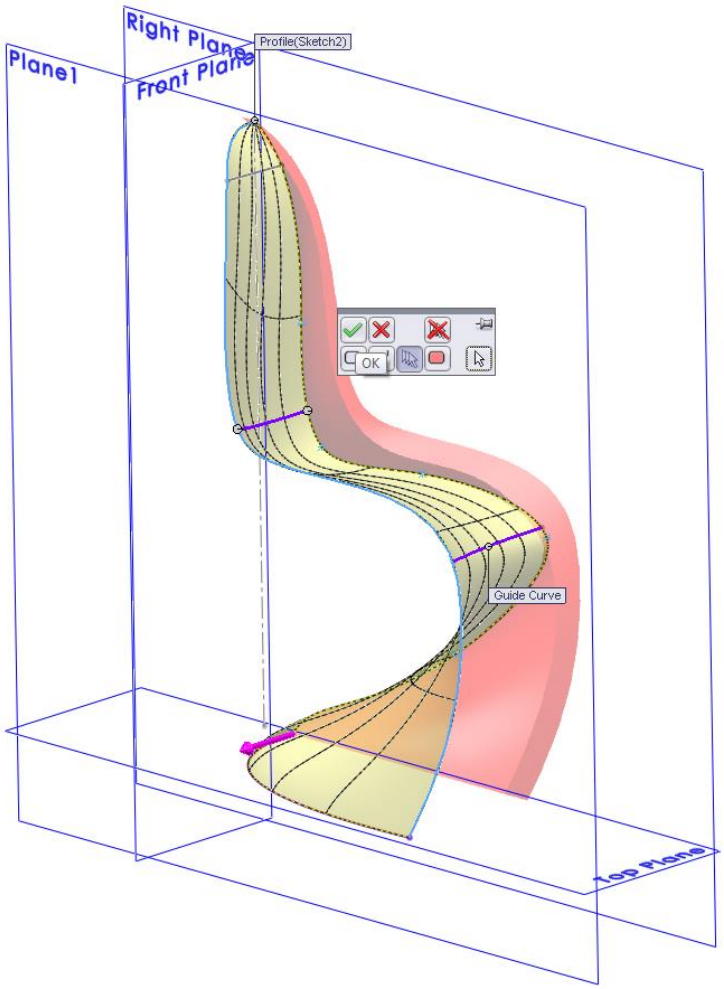
Click at one of the splines of the 3DSketch

Click OK to make a Guideline of it



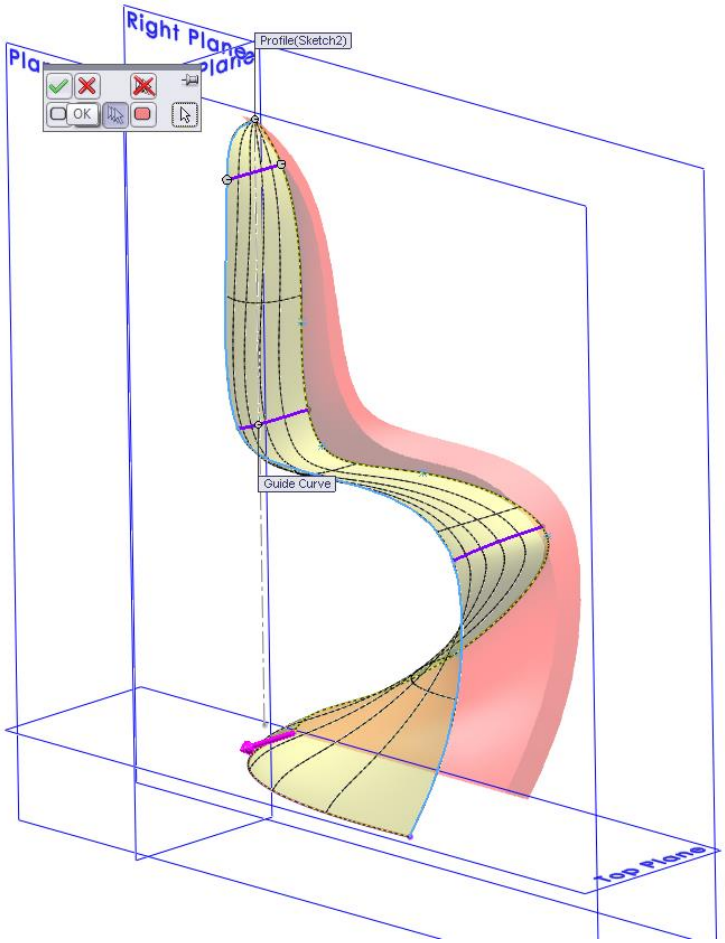
Select the second spline of the 3DSketch

Click OK to make a Guideline of it

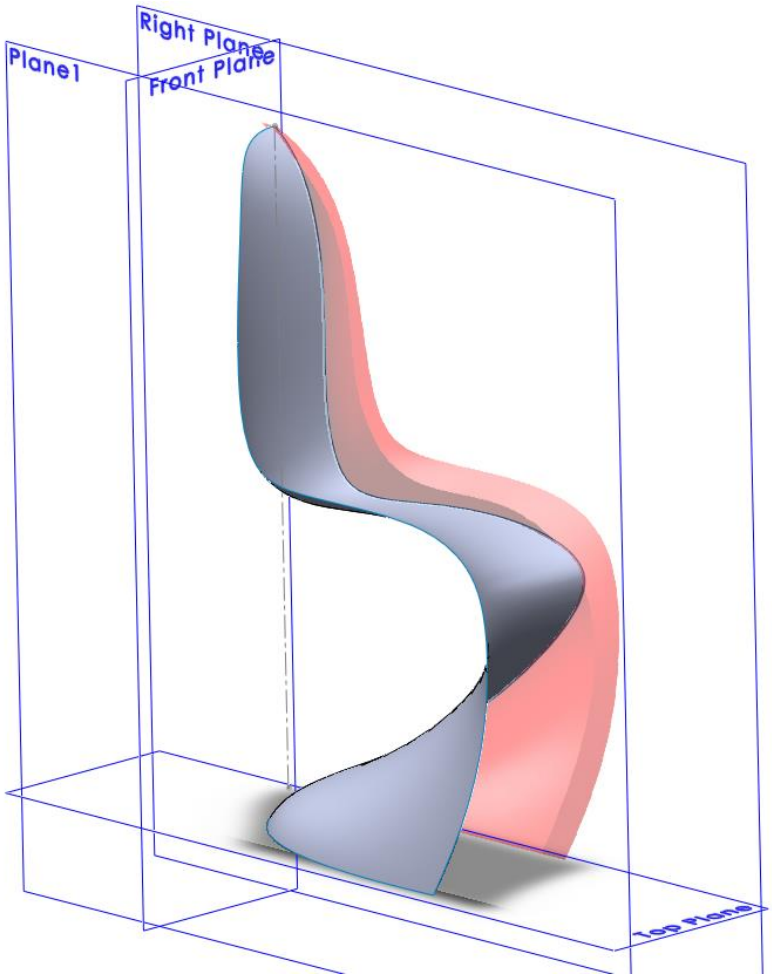


Select the third spline of the 3DSketch

Click OK to make a Guideline of it



Click OK to finish the Surface Loft



Create a 2D sketch on the Right Plane

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

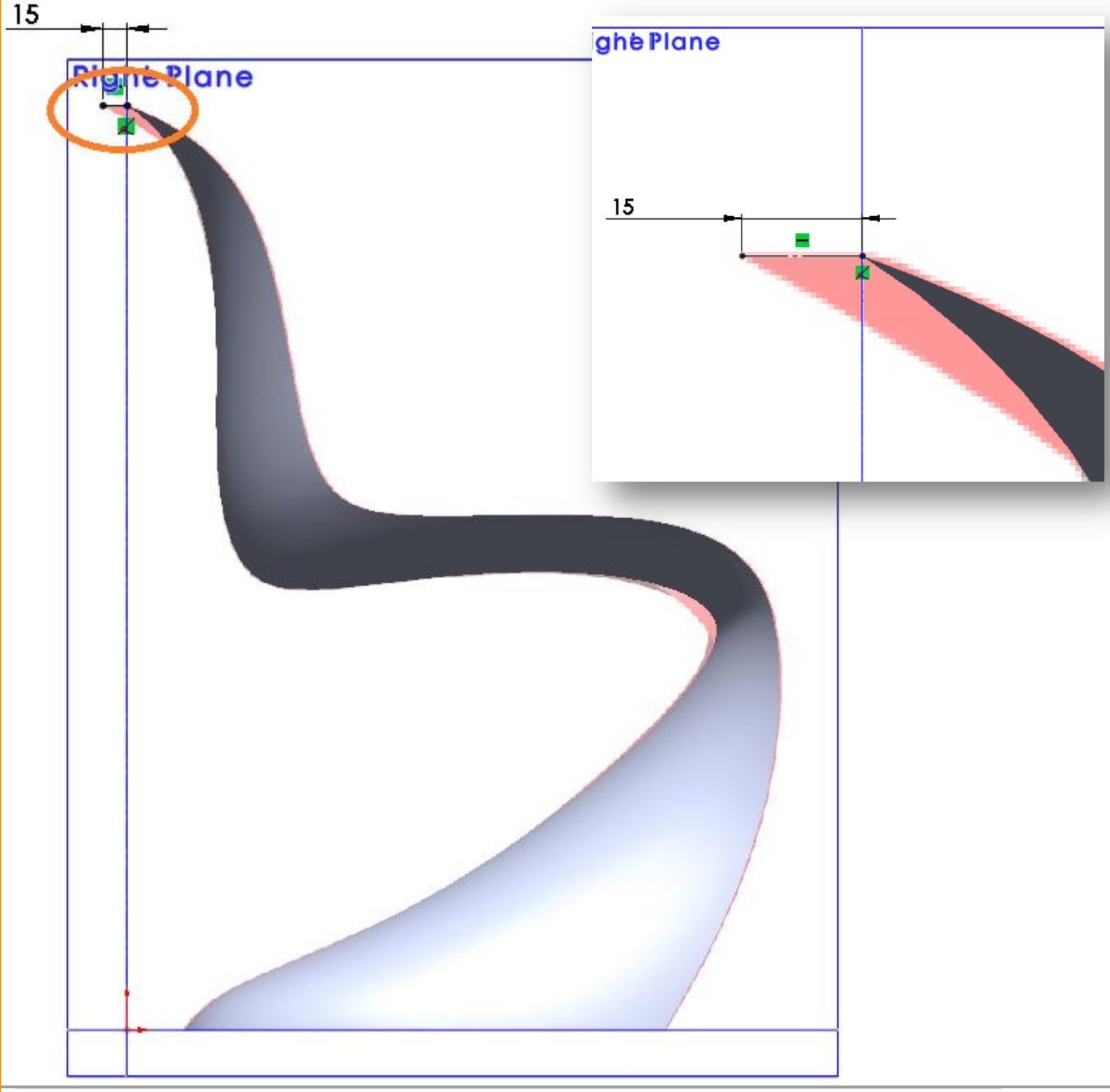
Draw a horizontal centerline

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

Draw a horizontal centerline that starts at the top point of Surface Loft1

Change the length of the line into 15 mm by clicking at the dimension button

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

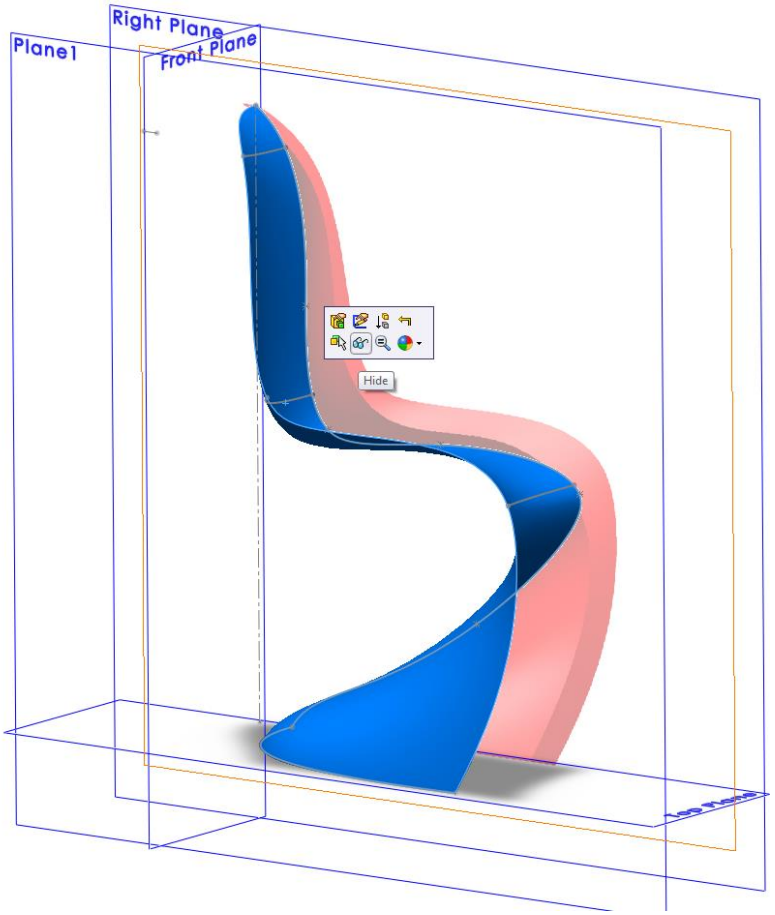


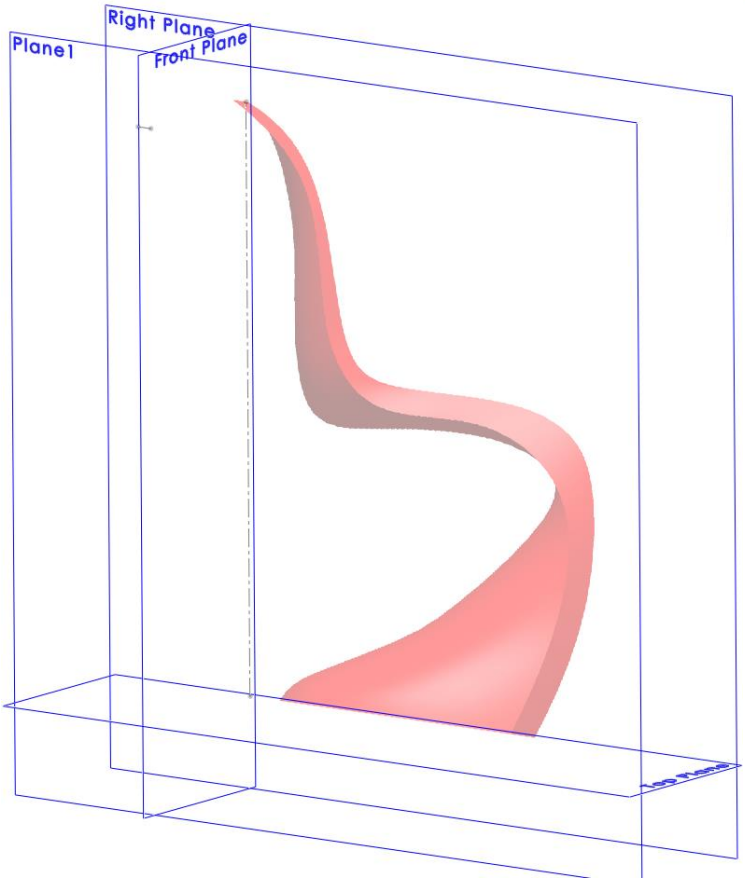
Hide Surface Loft1

To view the reference picture it's necessary to hide the Surface Loft temporary

Click on the Surface Loft

Click on the Glasses to Hide the body 





Re-open Sketch6

Click in the feature tree on Sketch6 and click at the 2D Sketch icon

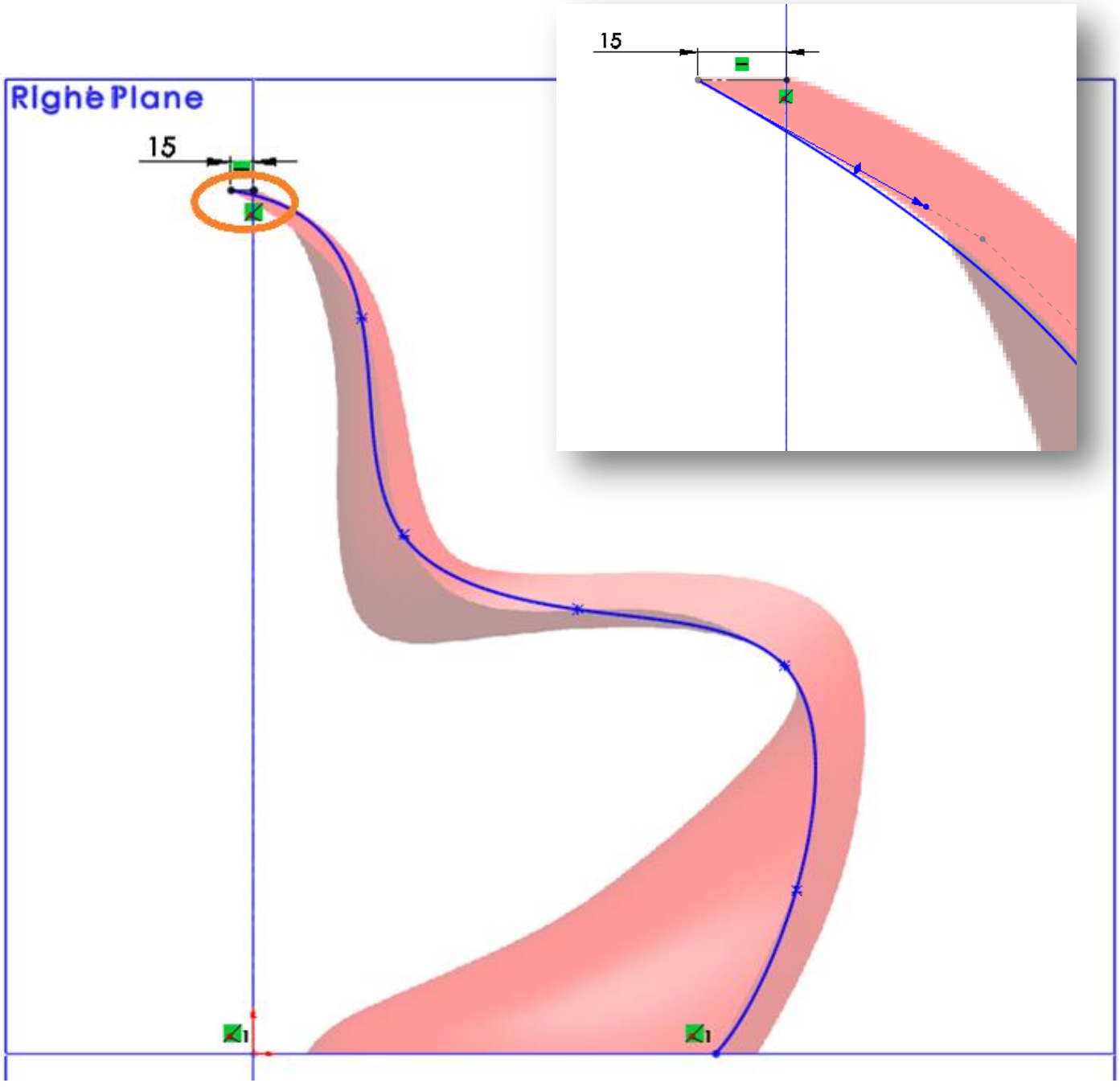
Draw a spline

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

Start the spline at the left endpoint of the construction line

Try to duplicate the edge curve of the chair as good as possible

Use as little spline points as possible (I used 7 spline points as shown in the picture)



Improve the shape of the curve

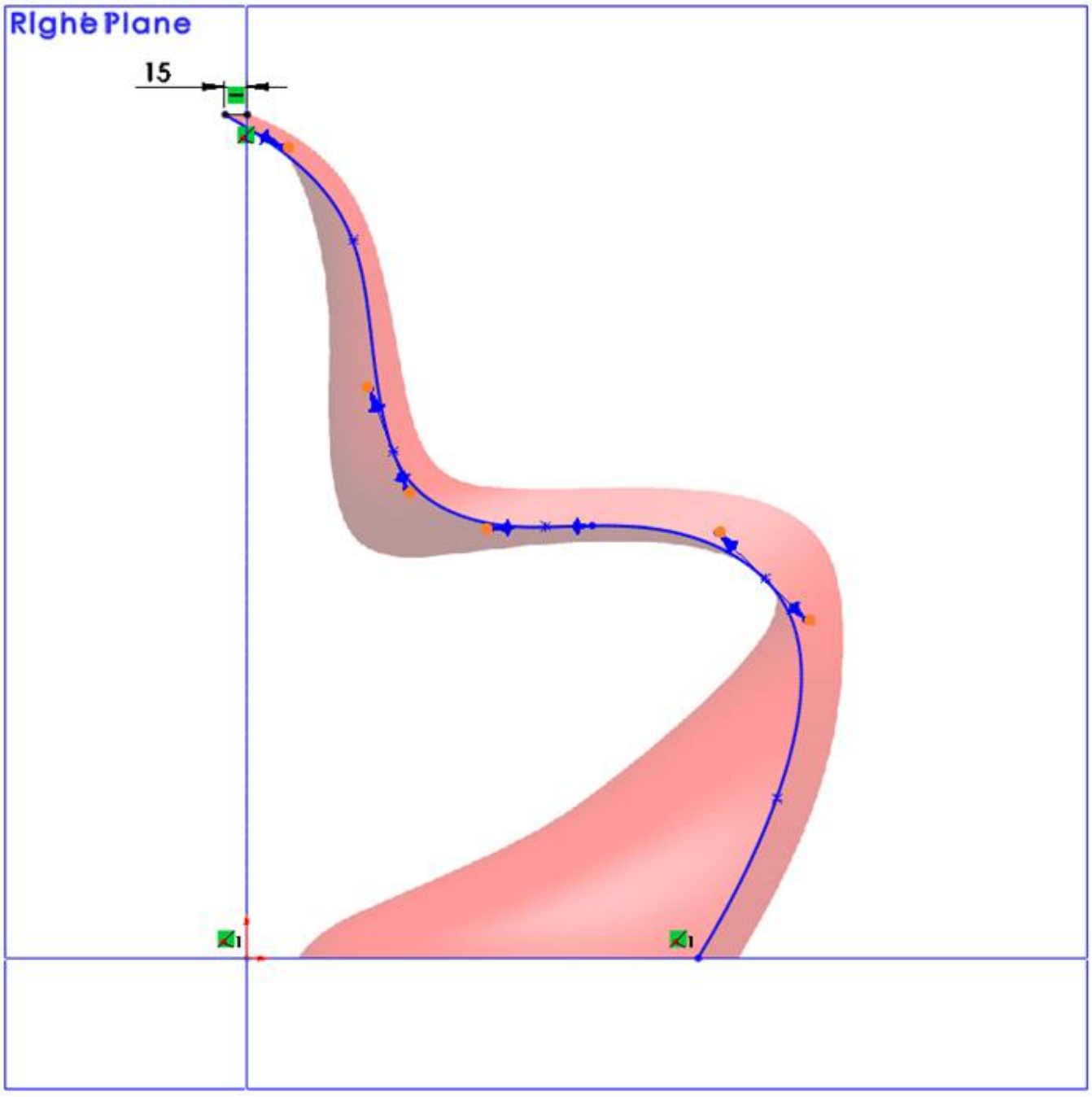
Click and drag the spline points to improve the shape of the curve

Change the direction of a spline point

Click on a spline point which you want to improve

The grey arrow of the Spline point appear

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



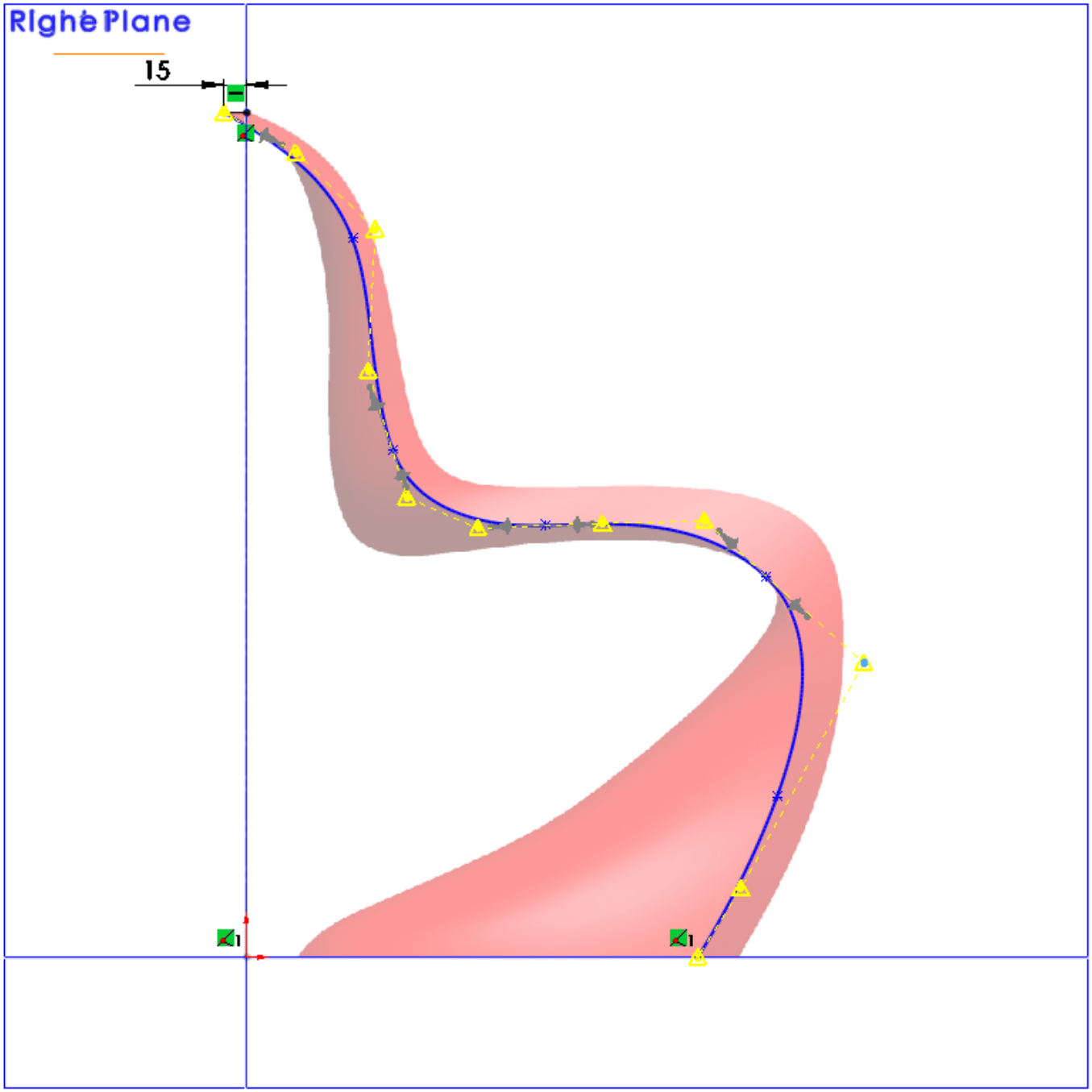
Improve the shape of the curve even more

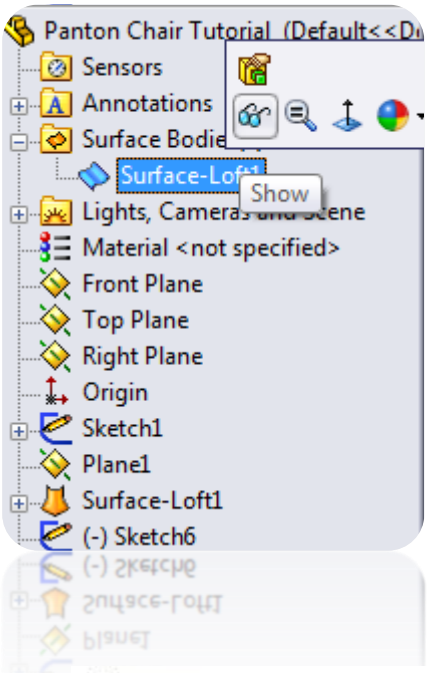
If you're still not satisfied with the curve you can use the Display Control Polygon option

Click on the **Spline > Right click > Display Control Polygon**

Click and drag one of the grey Polygon points to improve the shape even more

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



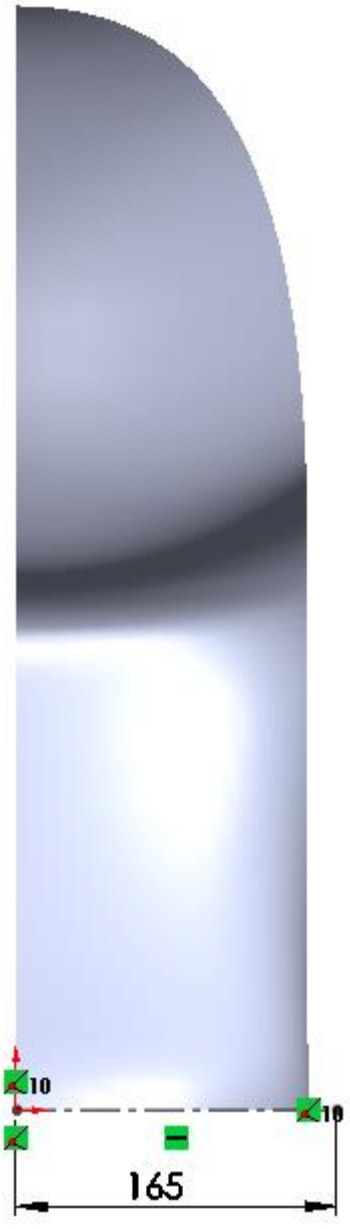


Unhide Surface Loft1

Click in the feature tree on the + before the **Surface Bodies** map

Click on **Surface-Loft1** as shown in the picture

Click on the **Glasses** to Show the body



Create another 2D sketch

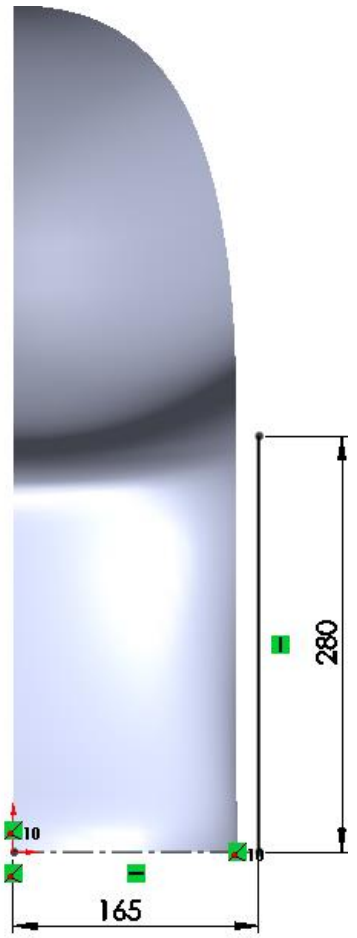
Select the Front Plane and create a sketch by clicking on the 2D Sketch icon

Draw a horizontal centerline

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

Draw a horizontal centerline that starts at the origin.

Change the length of the line into 165 mm by clicking at the dimension butt

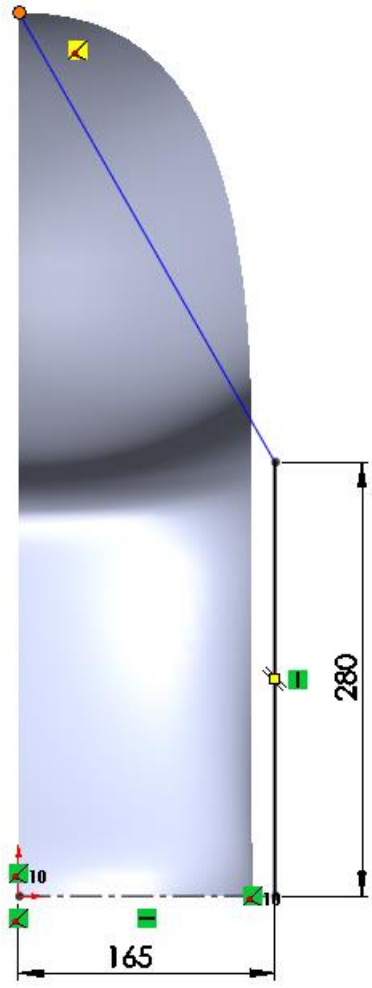


Draw a vertical line

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

Draw a vertical line that starts at the right end of the horizontal construction line

Change the length of the line into 280 mm by clicking at the dimension button

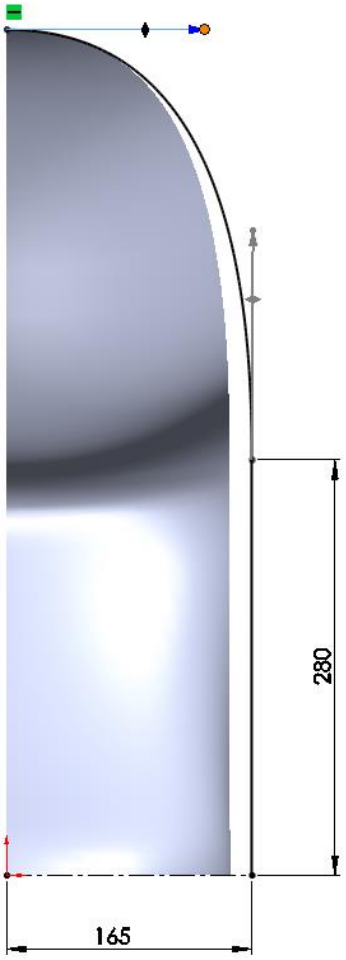


Draw a spline without midpoints

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

Start the spline at the upper point of the vertical line and ending at the top of the centerline of Surface Loft as shown in the picture

Right mouse button > Select



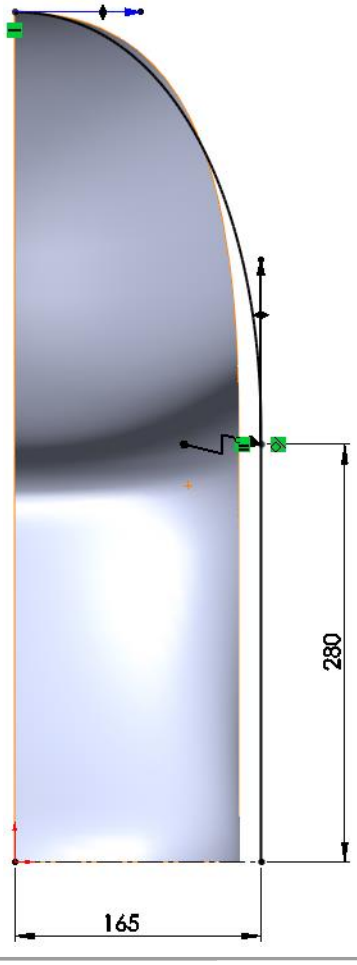
Add a horizontal tangency relation to the end of the spline

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 now perpendicular to the Right Plane

Click OK



Add a curvature relation to the other end of the spline

Click at the spline, hold the Control button and select the vertical line as well

Select the Curvature relation in the Spline menu bar at the left side

The transition between the line and spline is now curvature

Click OK

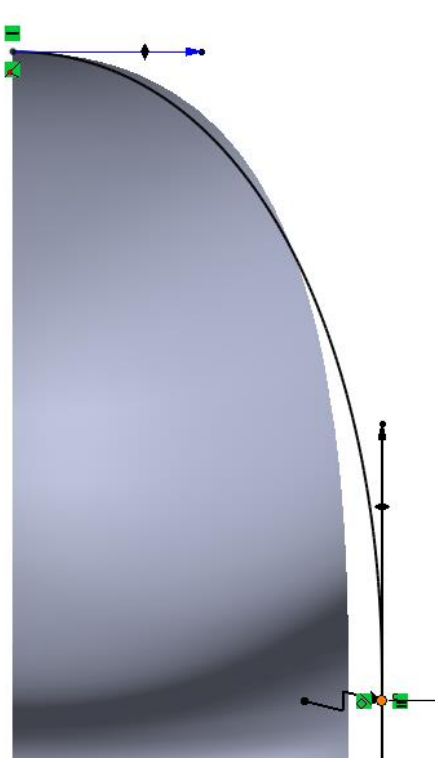
Change the dimension of the curvature relation

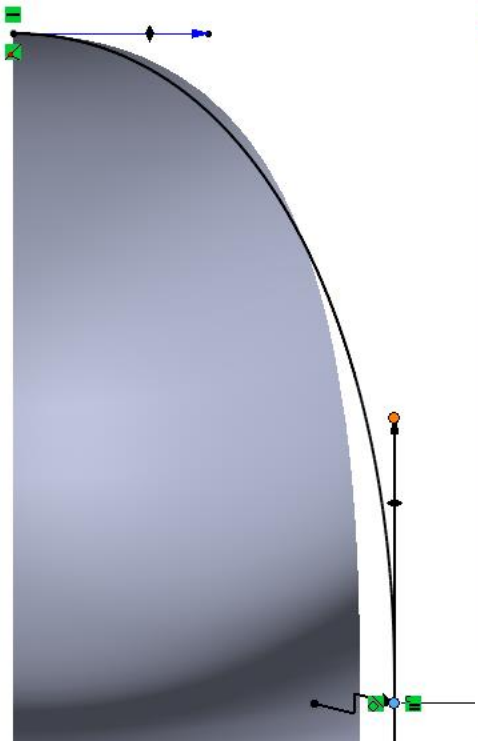
Click at the dimension button 

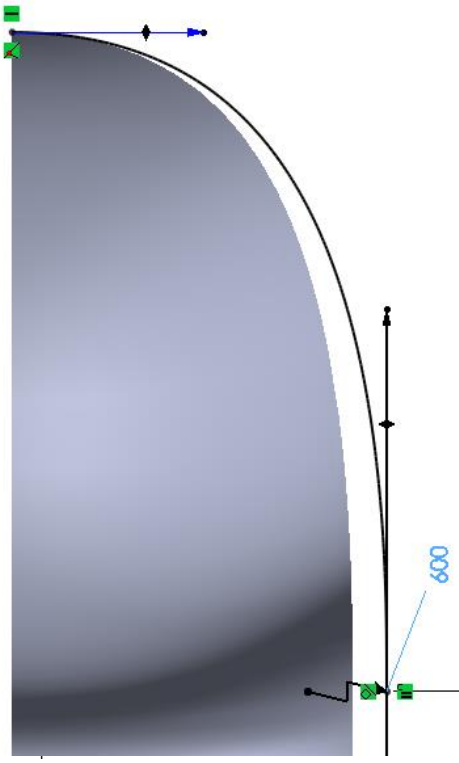
Select the starting point of the spline as shown in the first picture

Select the orange endpoint of the curvature arrow as shown in the second picture

Change the dimension into 600 mm as shown in the third picture







Change the dimension of the tangent relation

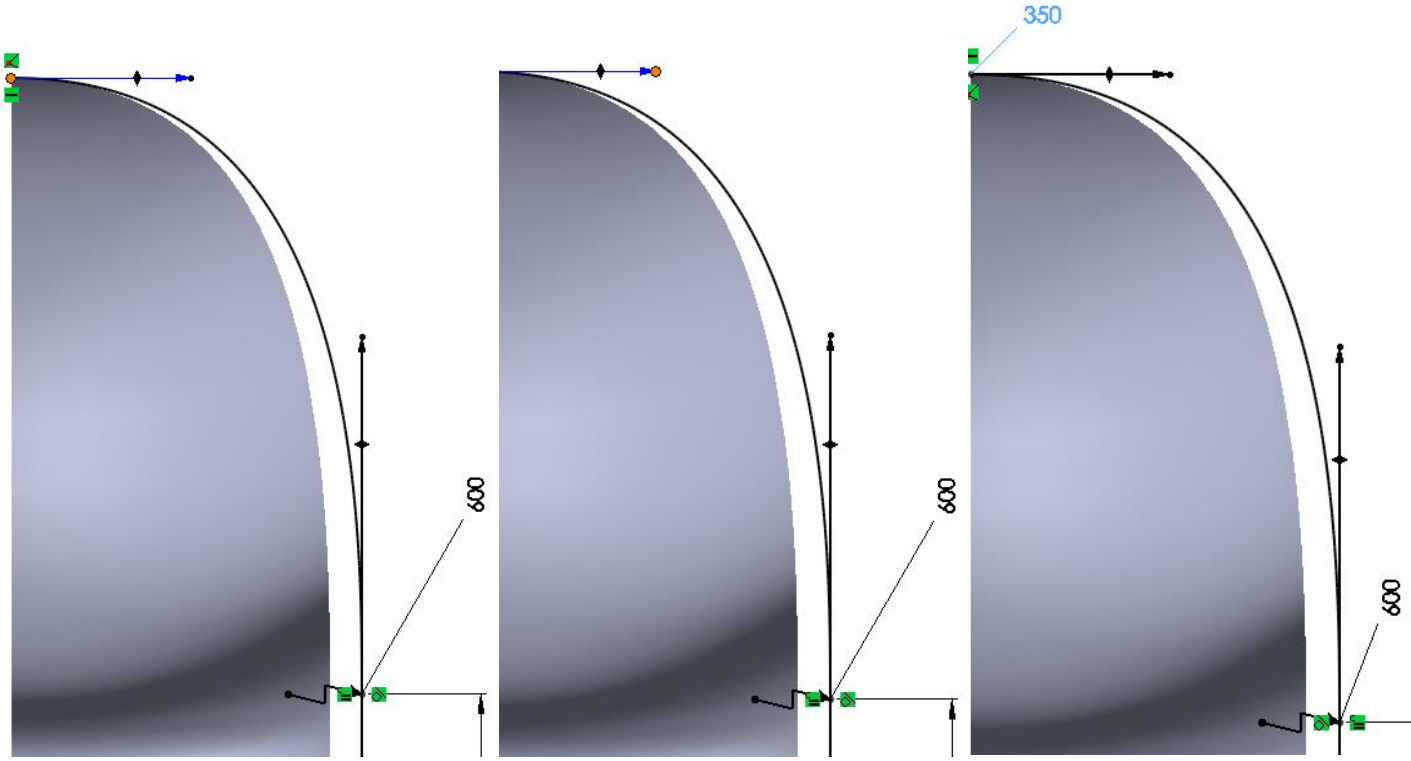
Click at the dimension button 

Select the starting point of the spline as shown in the first picture

Select the orange endpoint of the tangent arrow as shown in the second picture

Change the dimension into 350 mm as shown in the third picture

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



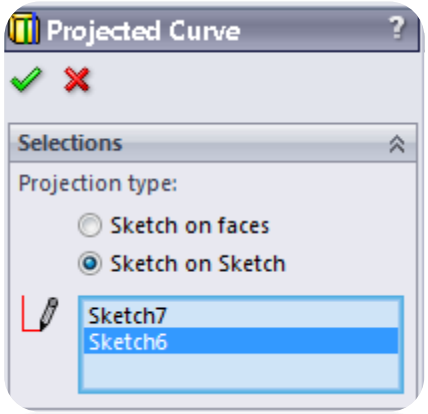
Create a projected curve

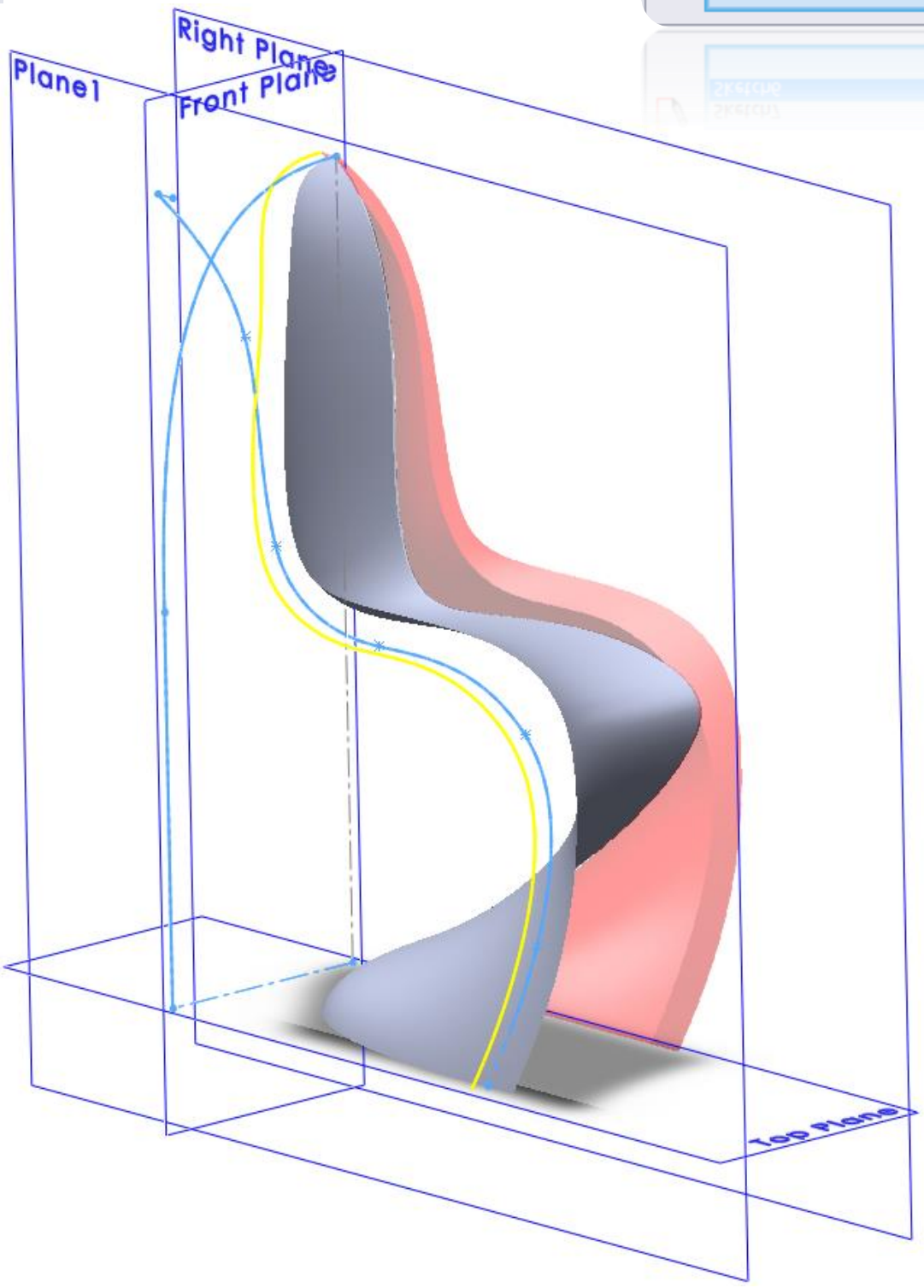
Go to: **Insert > Curve > Projected**

Select the **Sketch on Sketch** option

Select Sketch6 and Sketch7 as shown in the picture

Click OK



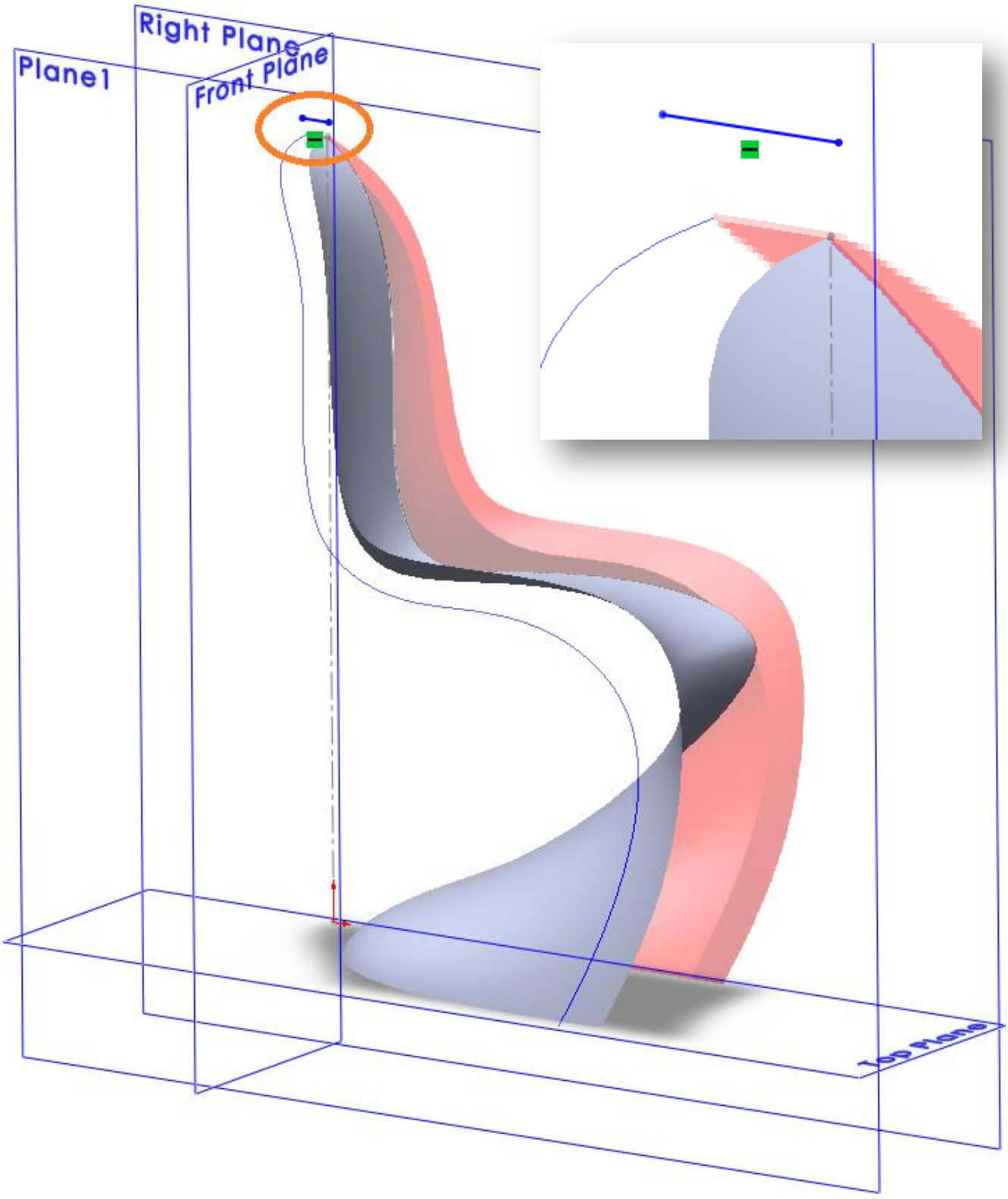


Create a 2D sketch on the Right Plane

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

Draw a line

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



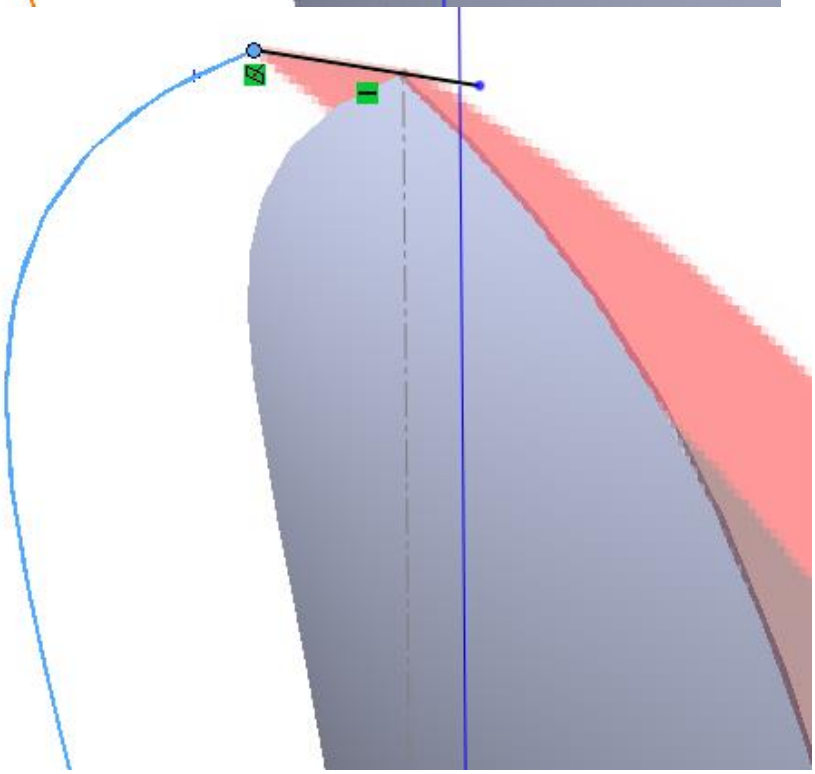
Connect the line with the new Curve2

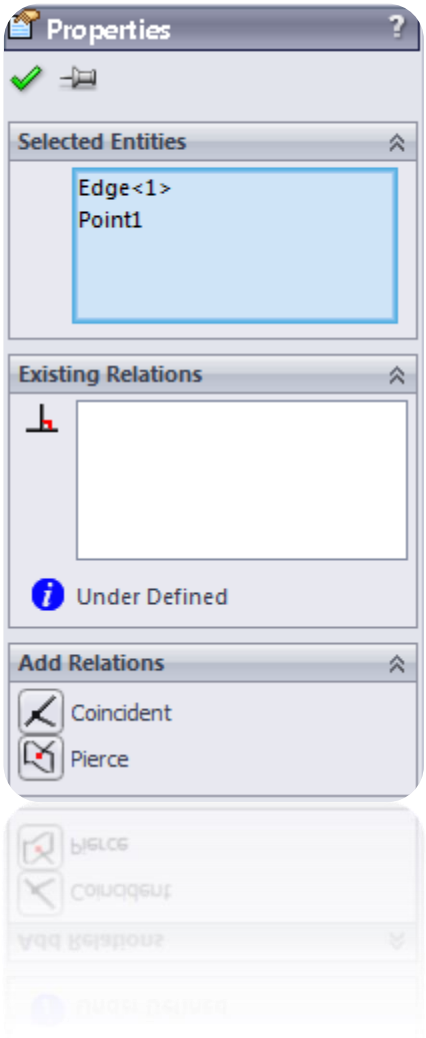
Select a line point, hold the Control button and select the new Curve2

Select the **Pierce relation** in the Add relation menu bar at the left side 

The line and Curve2 are now connected

Click OK 





Connect the other end of the line with the Surface Loft

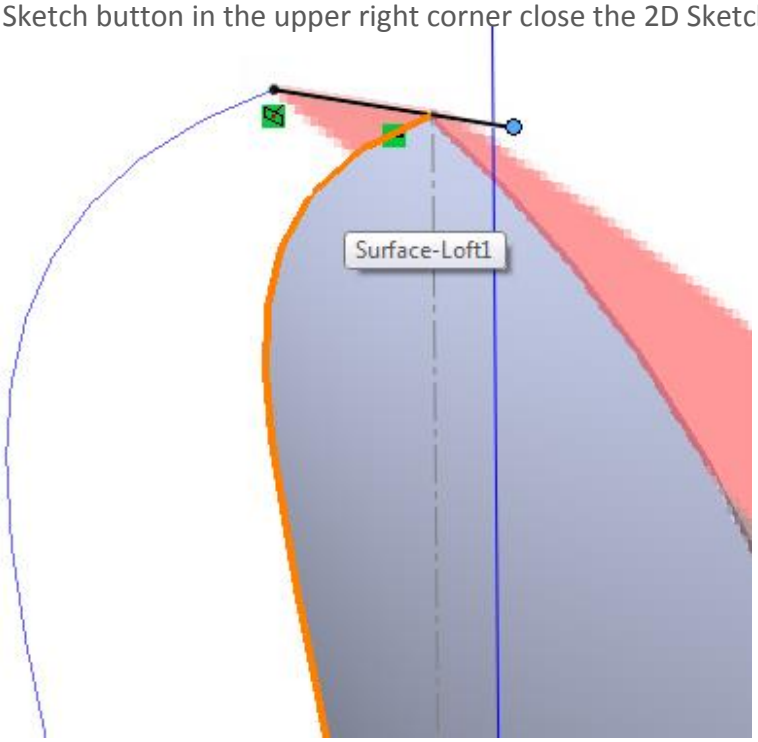
Select the other line point, hold the Control button and select the edge of Surface Loft 1

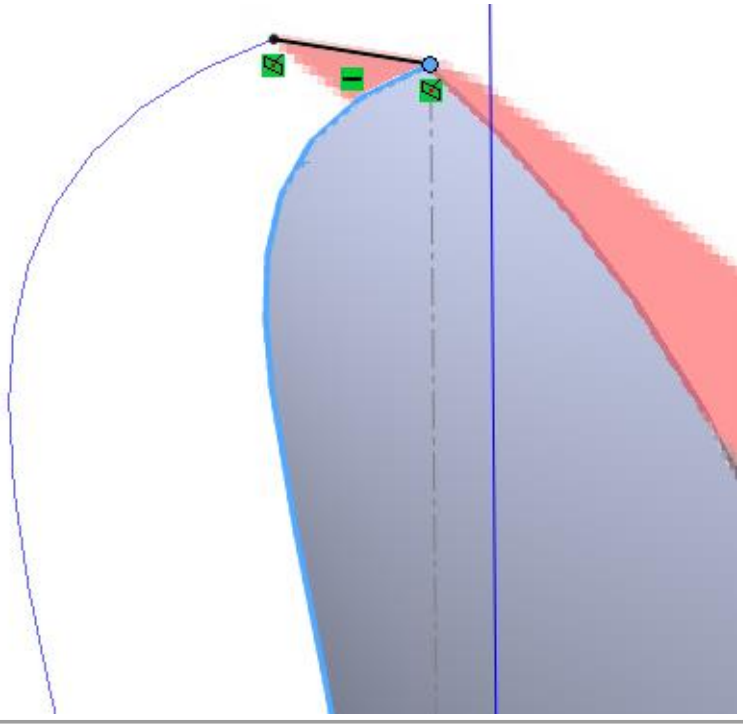
Select the **Pierce relation** in the Add relation menu bar at the left side 

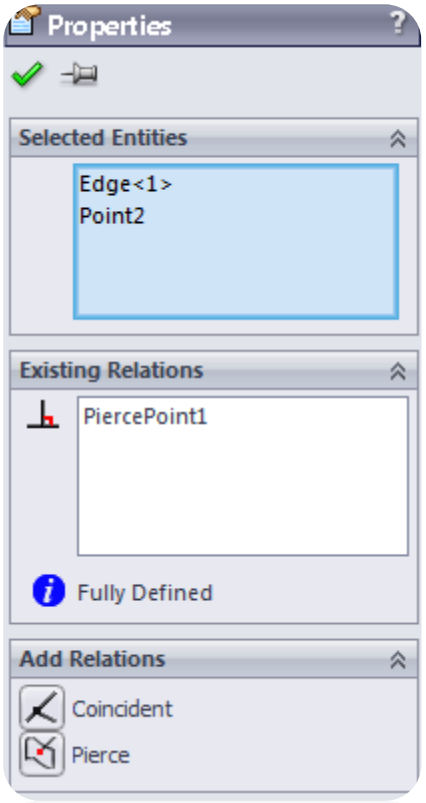
The line and Surface Loft 1 are now connected

Click OK 

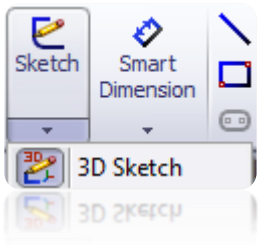
Click at the Sketch button in the upper right corner close the 2D Sketch 







Create a 3D sketch



Click at the dropdown menu under the 2D Sketch icon

Select the 3D Sketch option

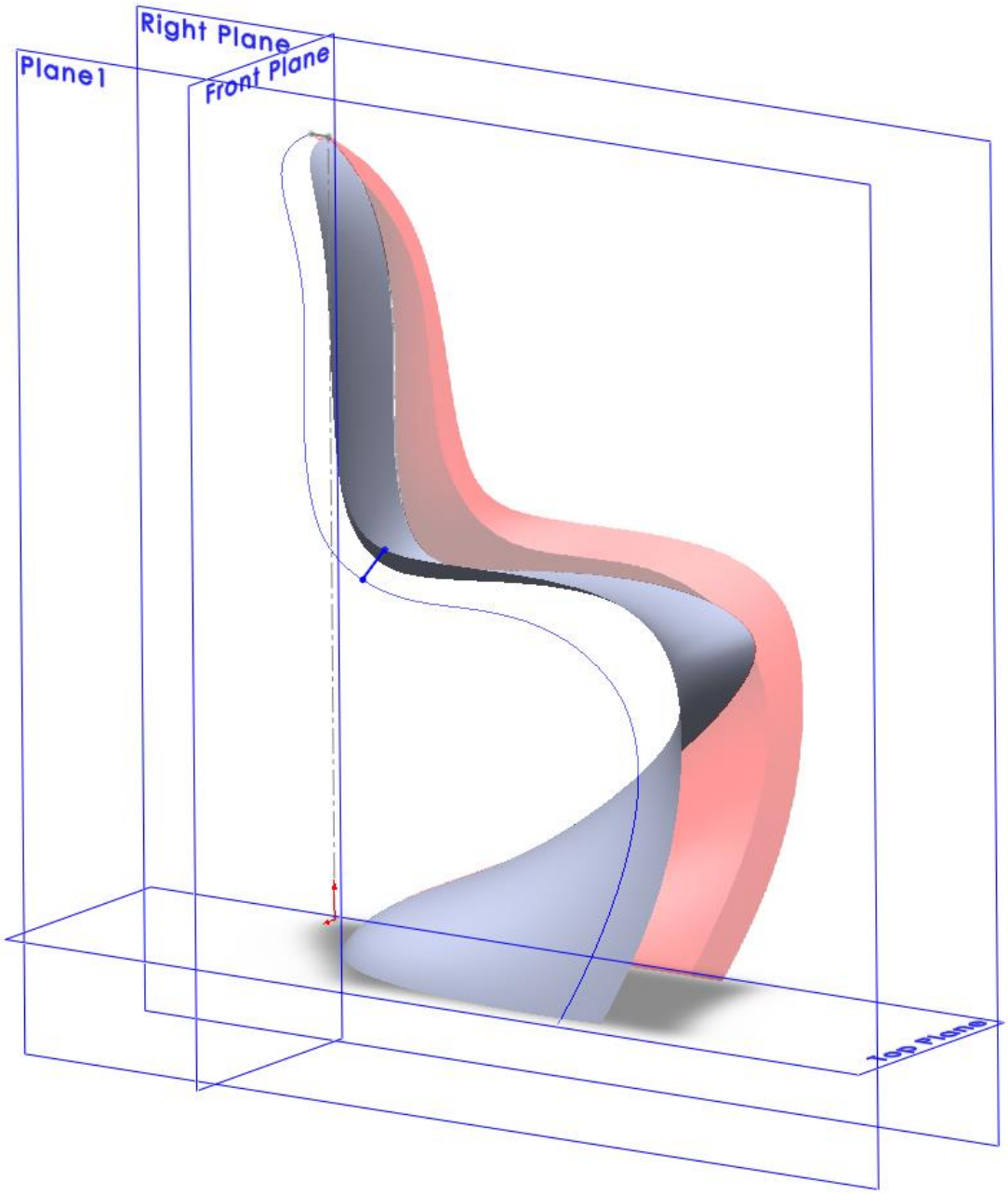
Draw a spline

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

Draw a 3d spline without any midpoints on the global position as shown in the picture

Connect the endpoints of the spline with Curve2 and Surface-Loft1

Click at the Sketch button in the upper right corner close the 3D Sketch



Create a Surface Loft

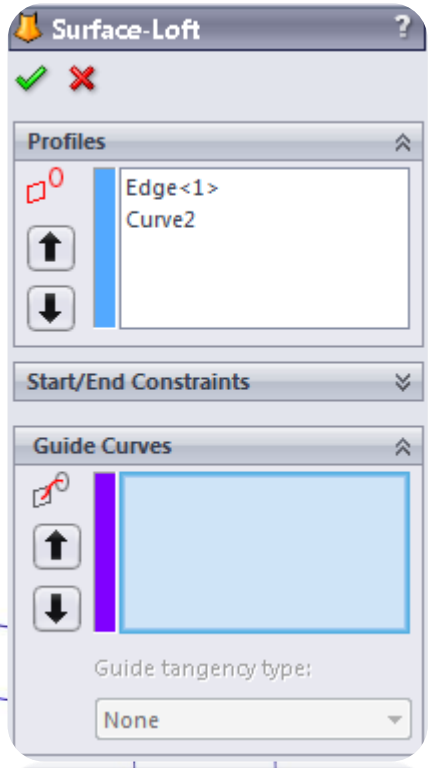
Go to **Insert > Surface > Loft** or click at the Surface icon

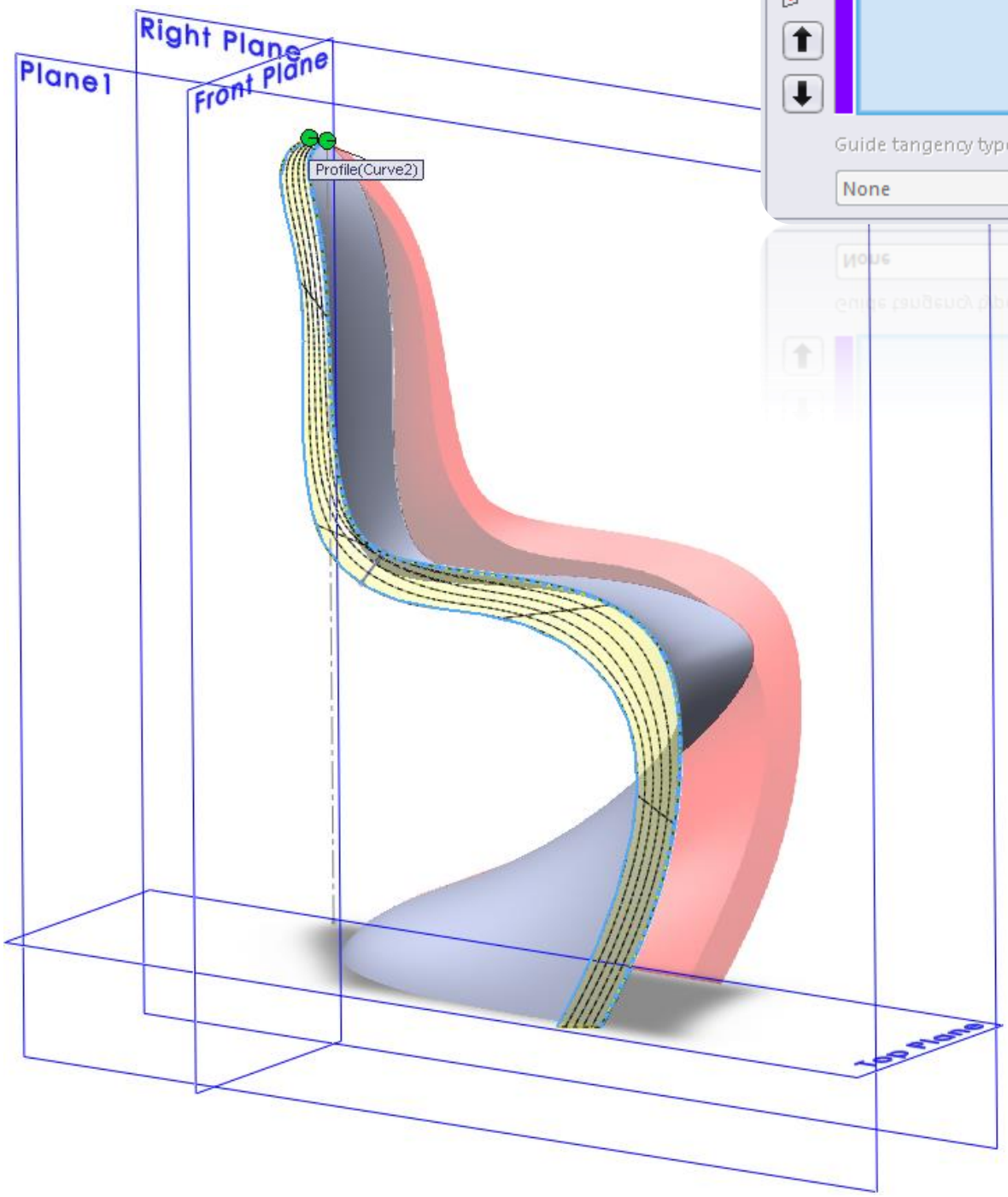
Click in the Profiles box

Select the edge of the surface loft and Curve2 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





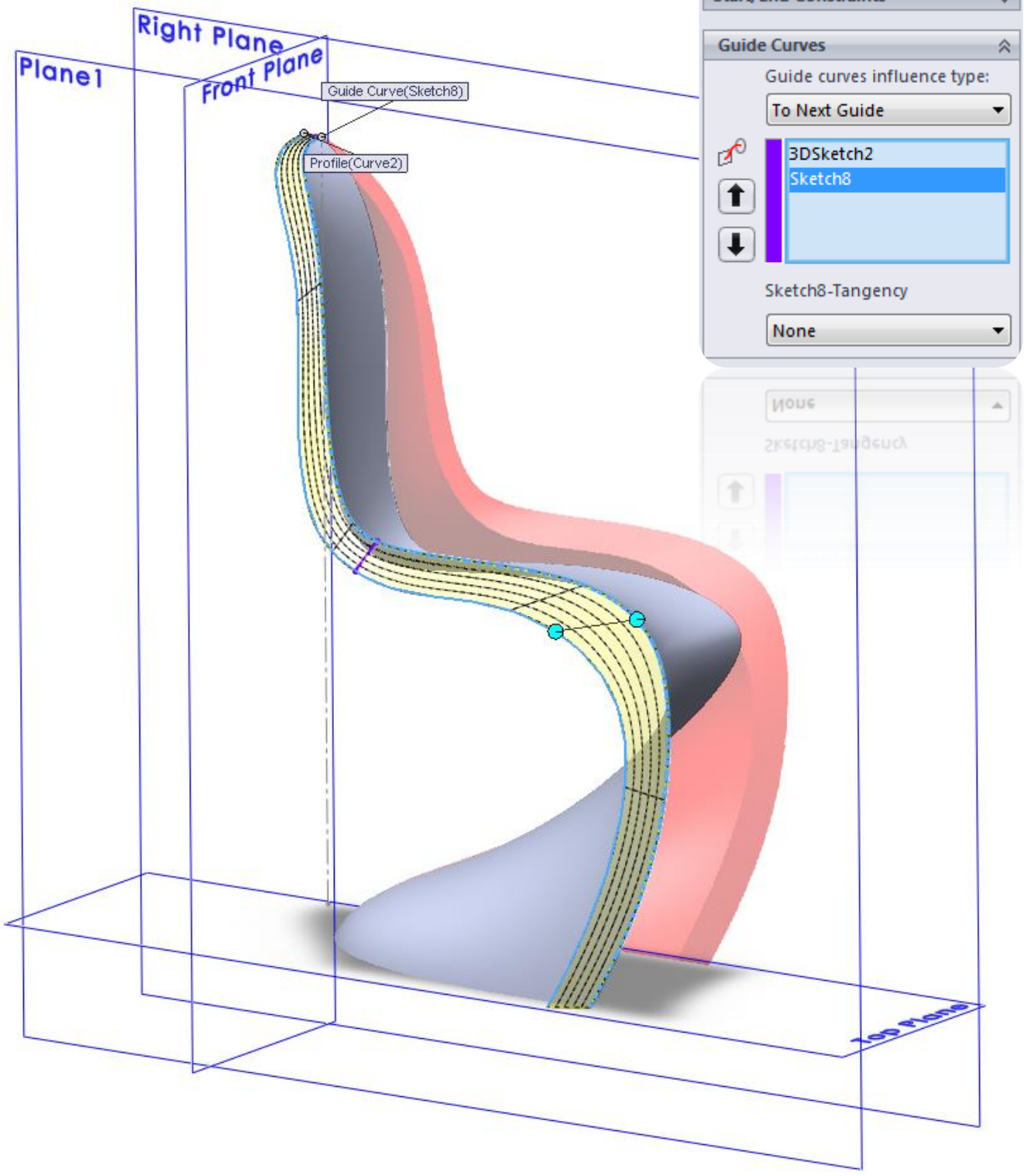
Add two Guide Curves to control the shape of the Surface Loft

Click in the Guide Curves box

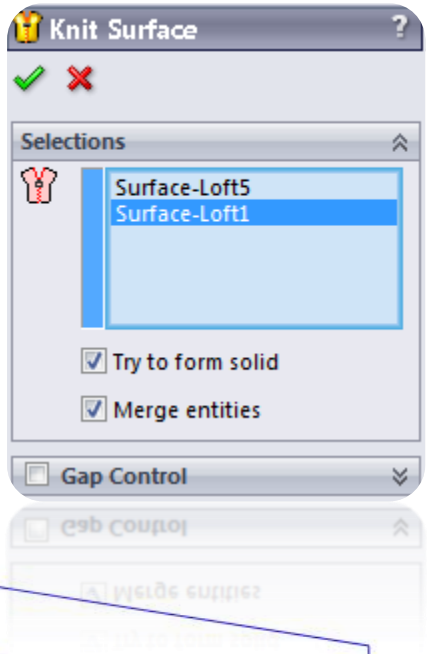
Select Sketch5 as shown in the picture

Guide curves influence: To Next Guide

Click OK



Knit the 2 surfaces and create a solid body



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

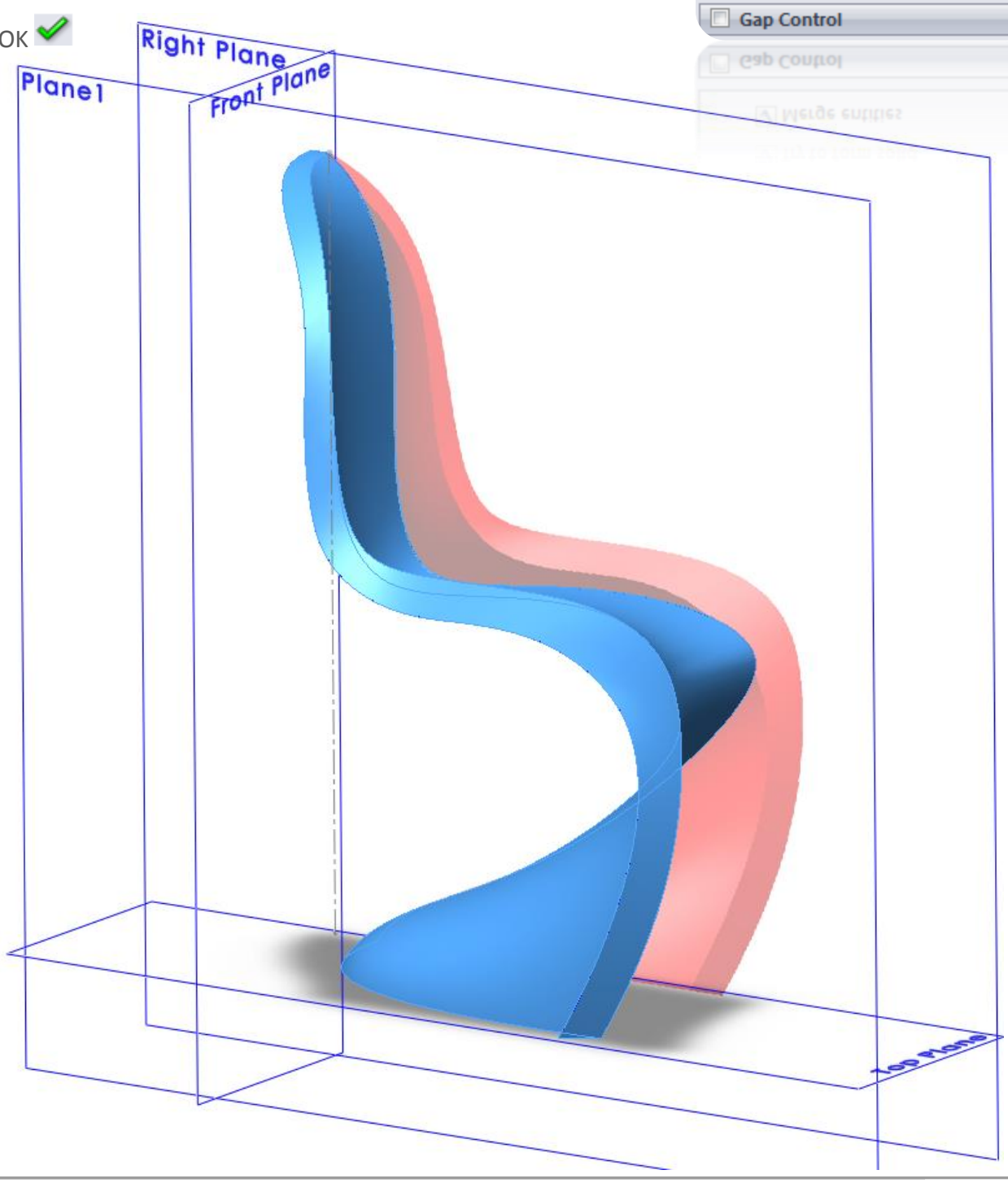
Click in the Selections box and select the 2 Surface Lofts

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

Select the **"Merge entities"** option

Deselect the **''Gap Control''** option

Click OK



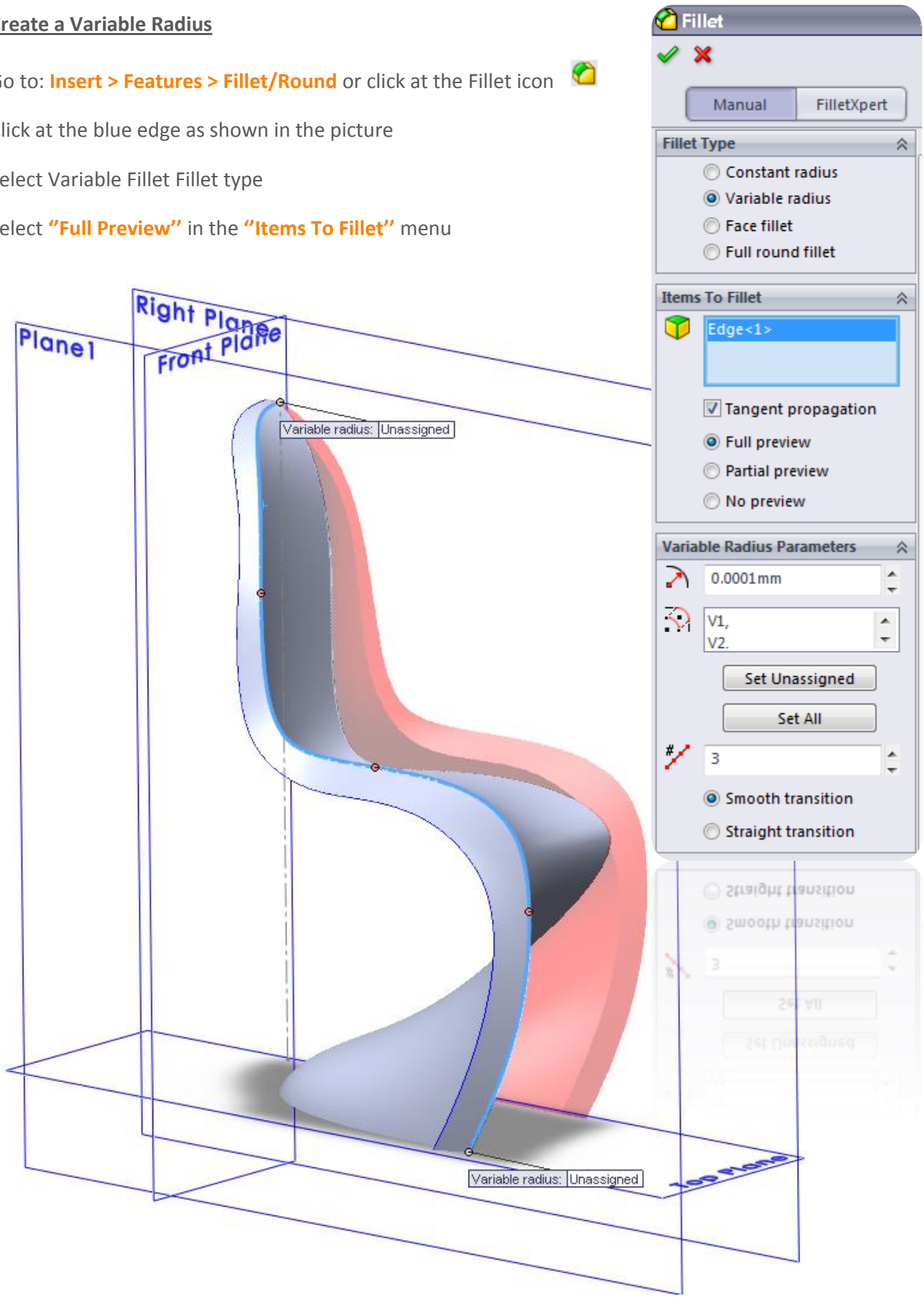
Create a Variable Radius

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

Click at the blue edge as shown in the picture

Select Variable Fillet Fillet type

Select **''Full Preview''** in the **''Items To Fillet''** menu



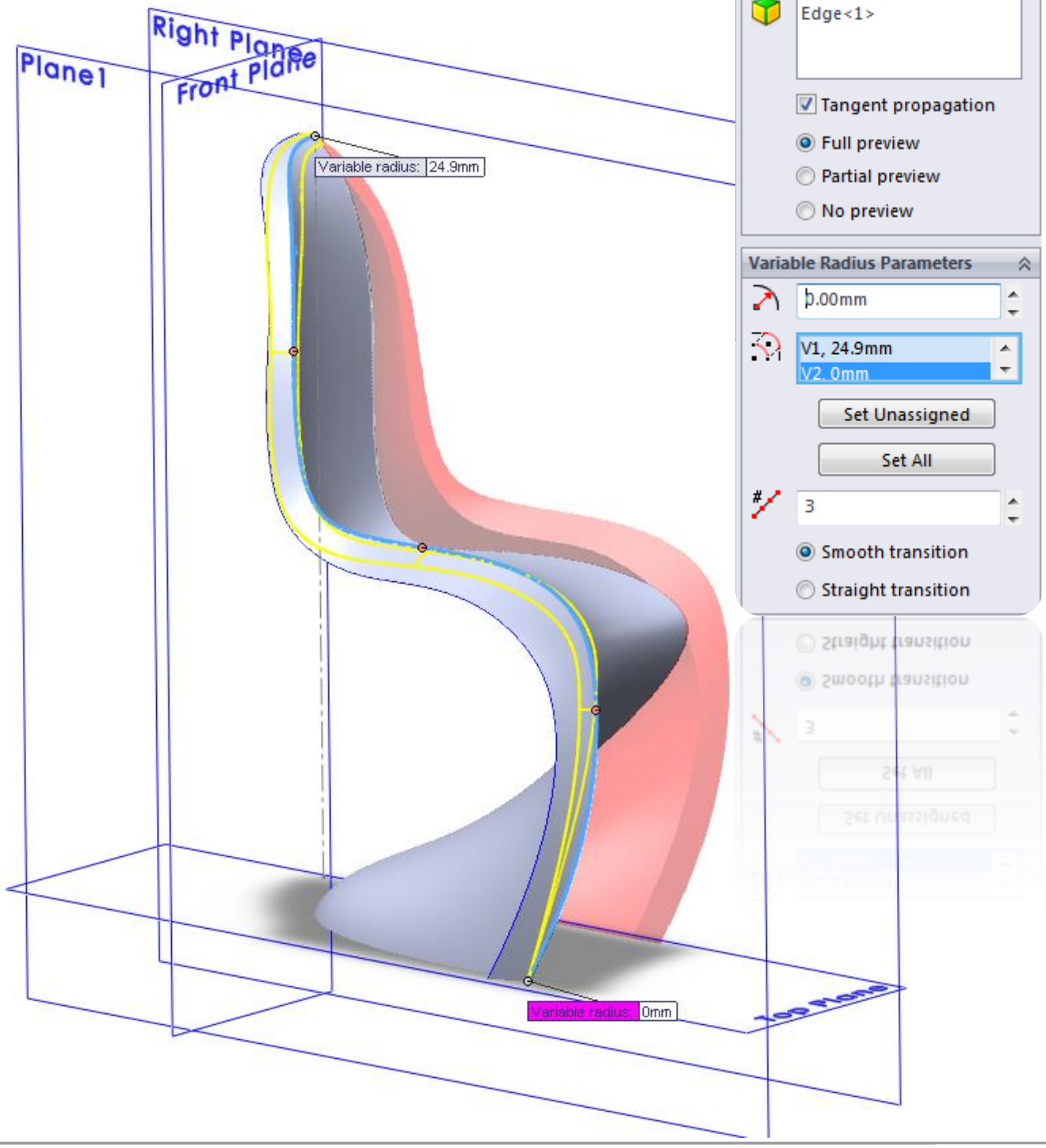
Change the Variable Radius Parameters

Select V1

Change the **Radius** into 24.9 mm

Select V2

Change the **Radius** into 0 mm

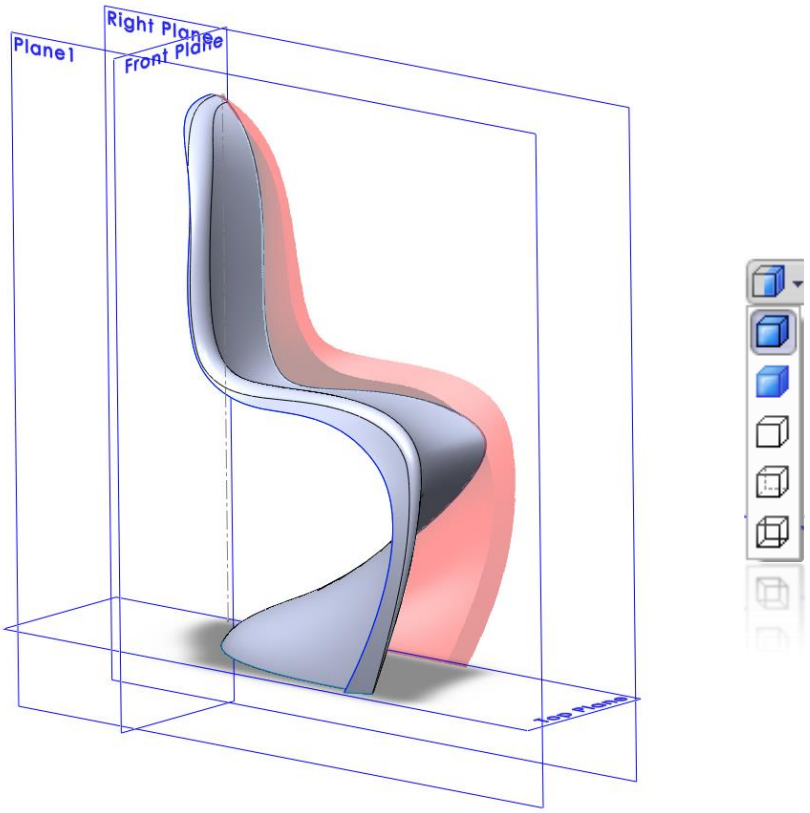


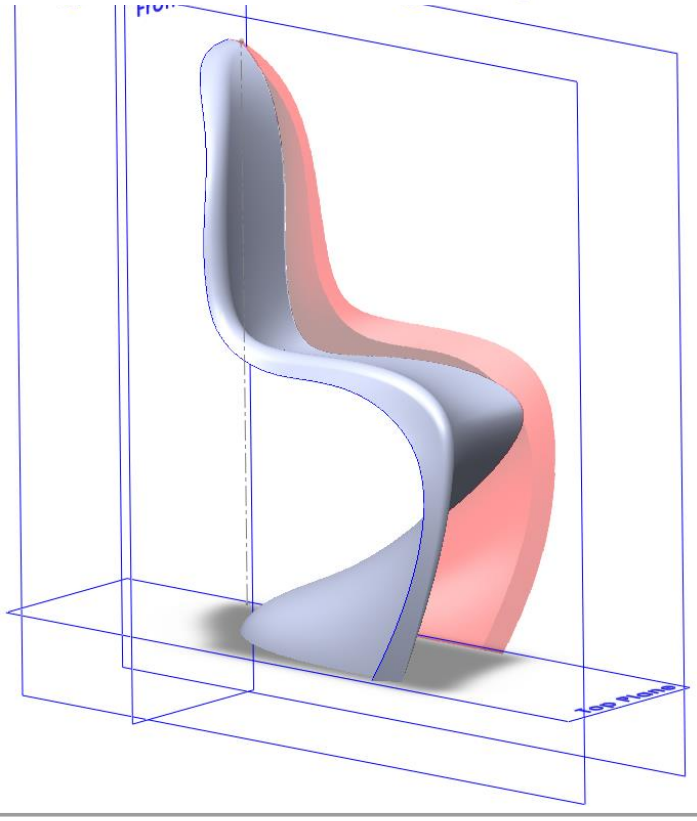
Change the display style

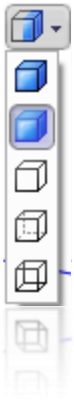
This helps us to assess the surface transitions better

Click at the **"Display Style Box"**

Change the display style from **Shaded with Edges** into **Shaded**



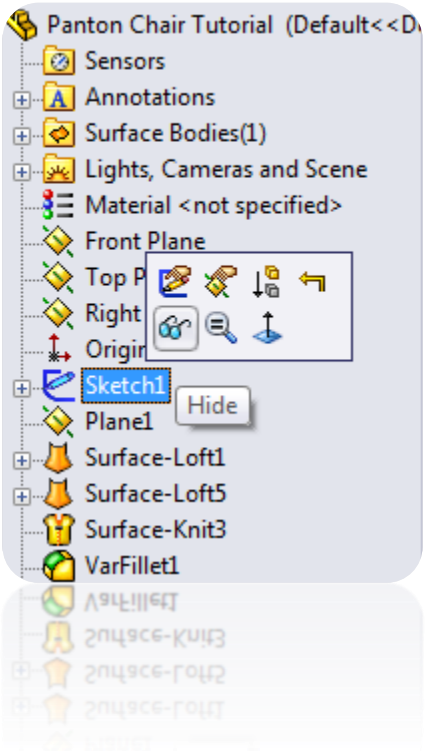


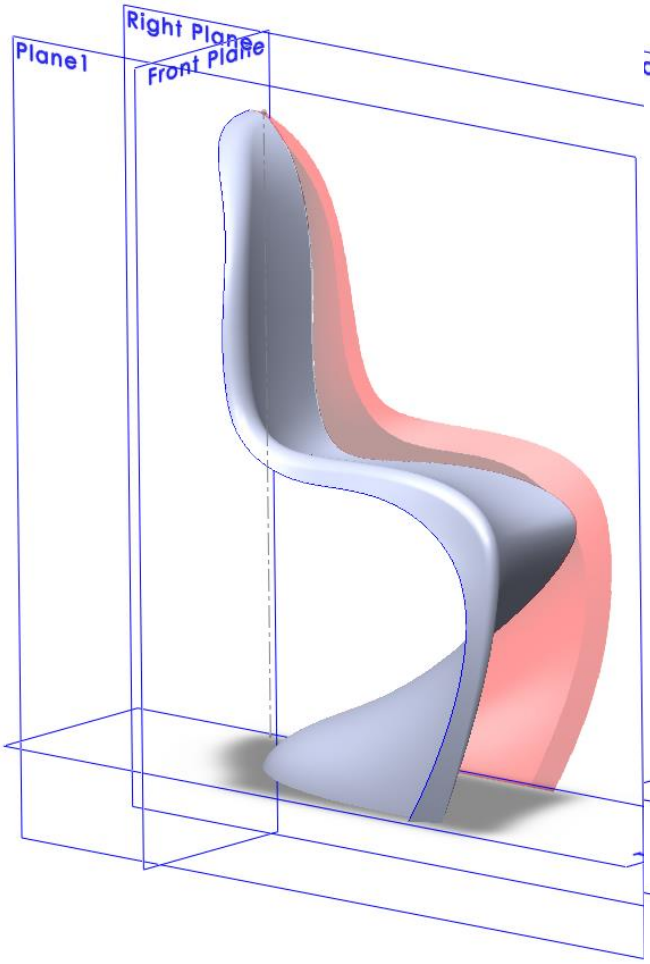


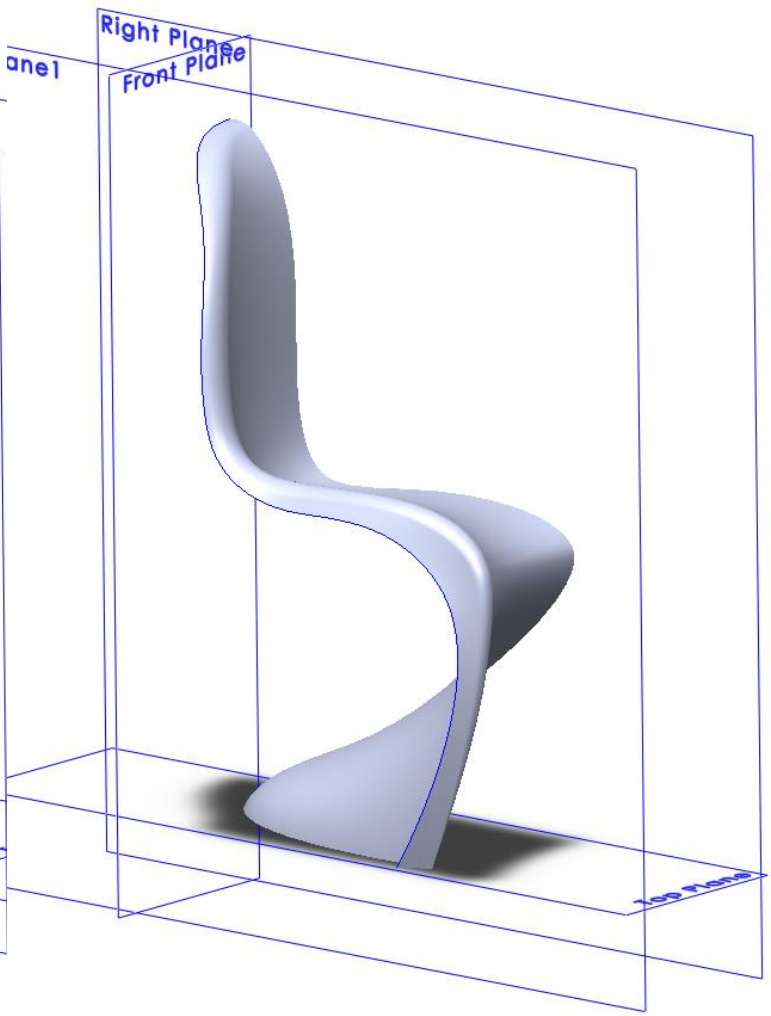
Hide the reference picture

Click at Sketch1 in the feature tree

Click on the **Glasses** to Hide the body







Mirror and merge the chair

Go to: **Insert > Pattern/Mirror > Mirror**

Mirror Face/Plane : Right Plane

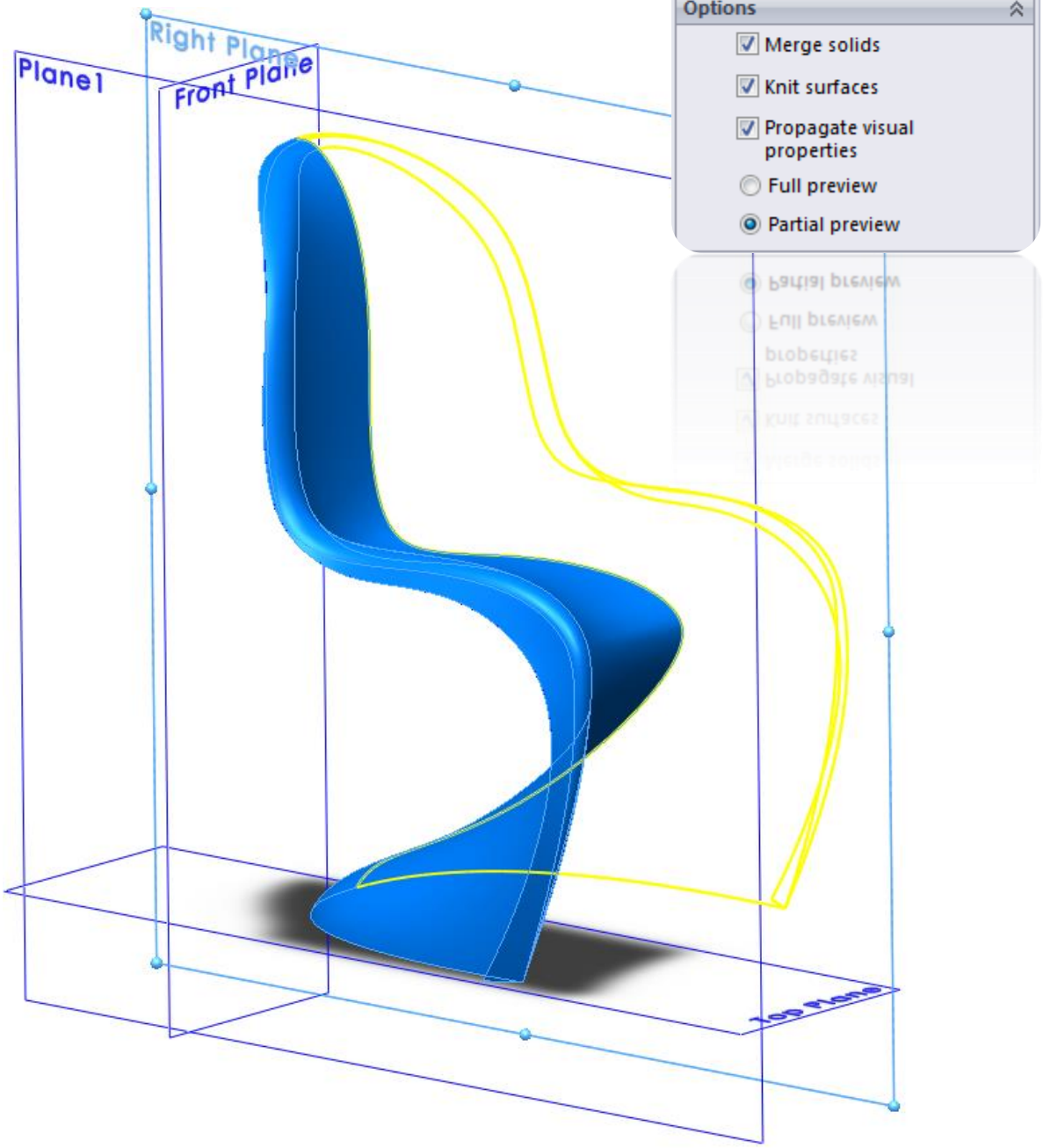
Select the “Bodies to Mirror” option

Bodies to Mirror : Select the surface body.

Select the ‘’Merge solids’’ option

Select the ‘’Knit surfaces’’ option

Click OK



Change the color of the chair

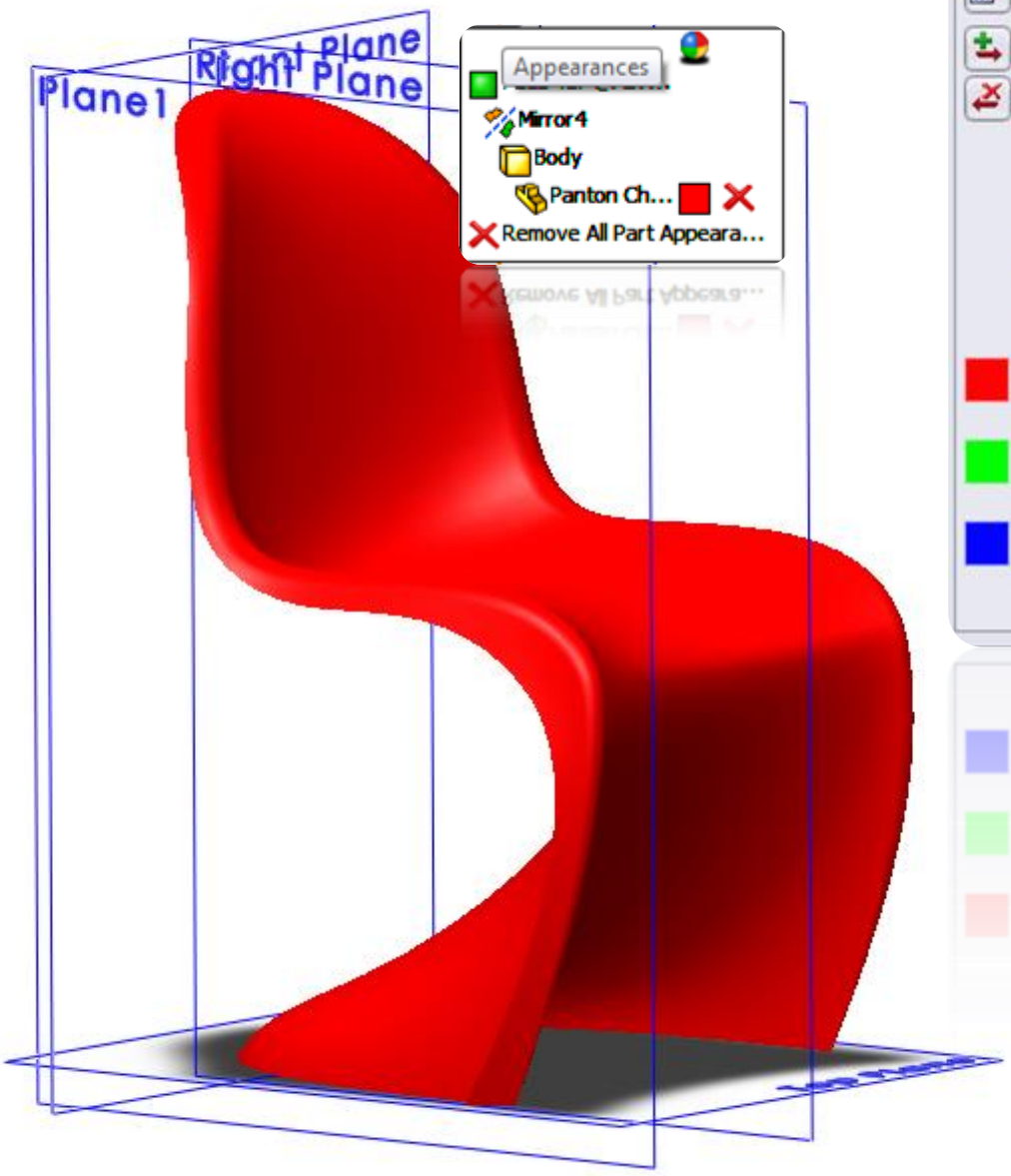
Select an arbitrary face of the chair

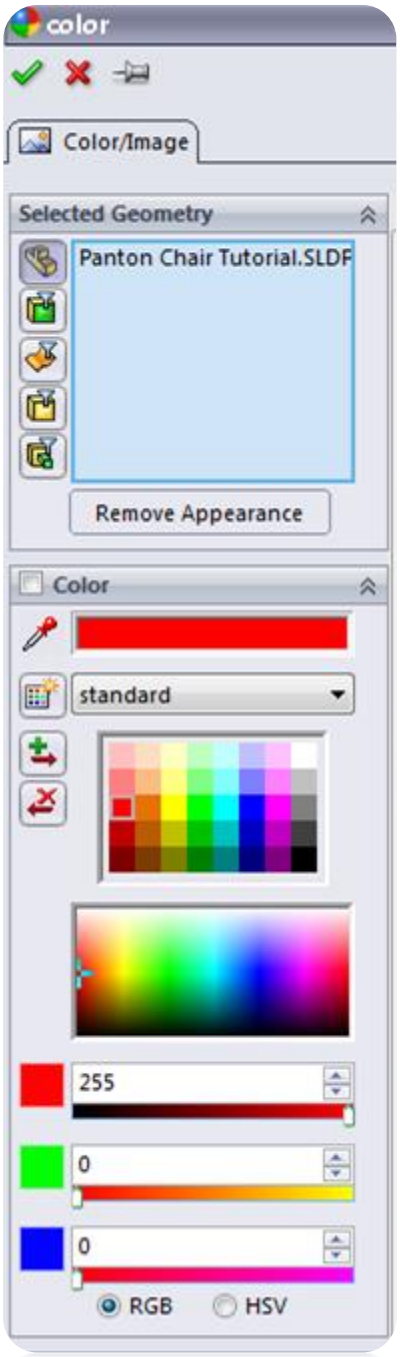
Click at the appearances button

Click at the Part name

Change the color into red

Click OK





Create a wall thickness

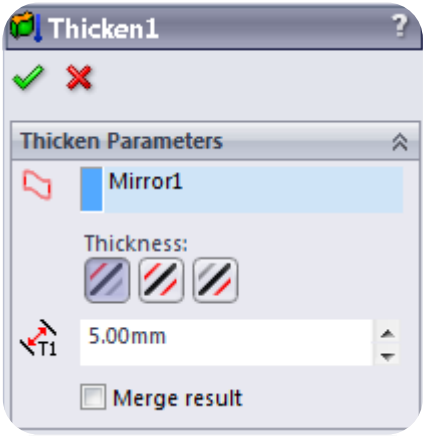
Go to: **Insert > Boss/Base > Thicken** or click at the Thicken icon

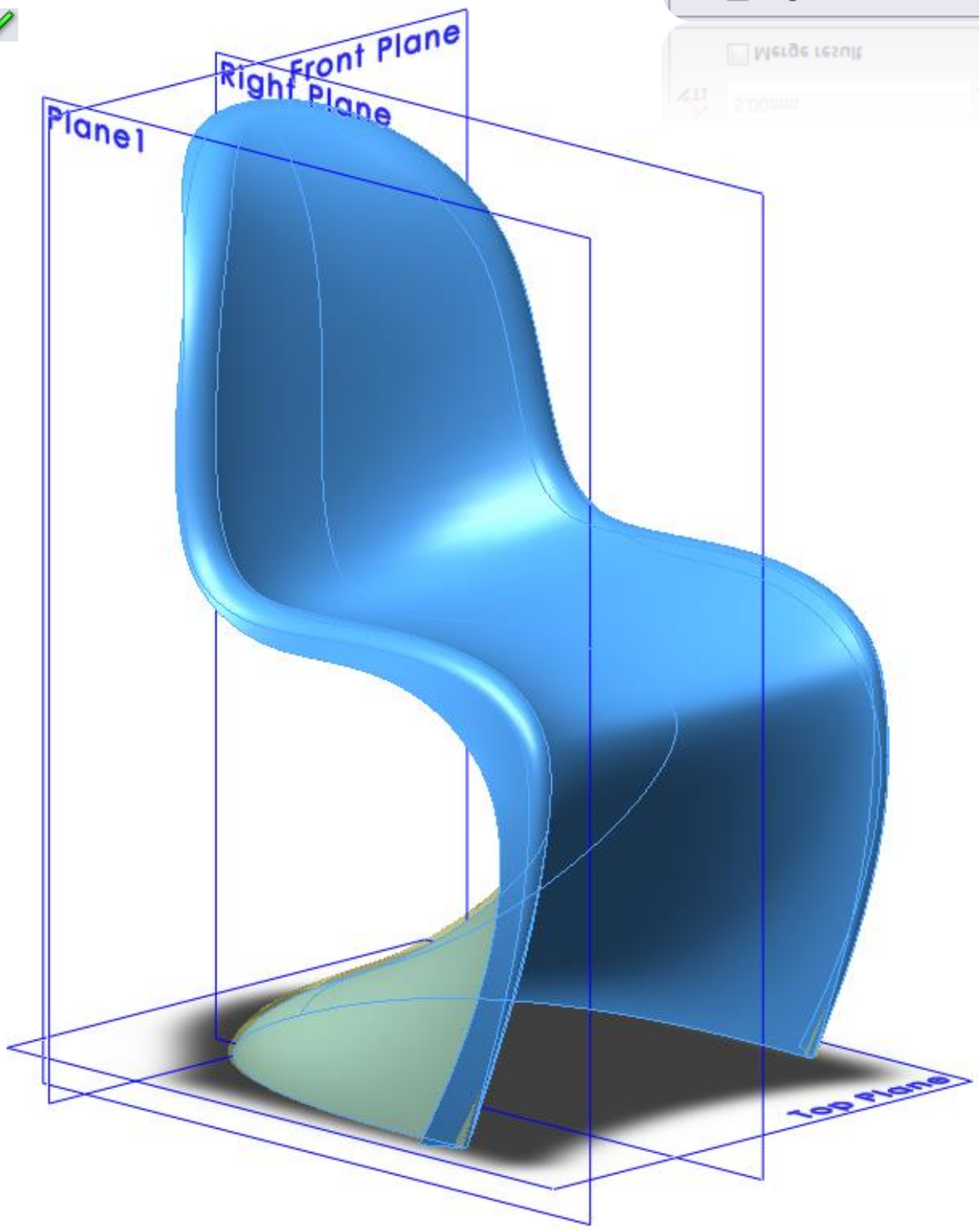
Click at the chair

Select the **"Thicken Side 1"** option

Change the wall thickness into 5 mm

Click OK





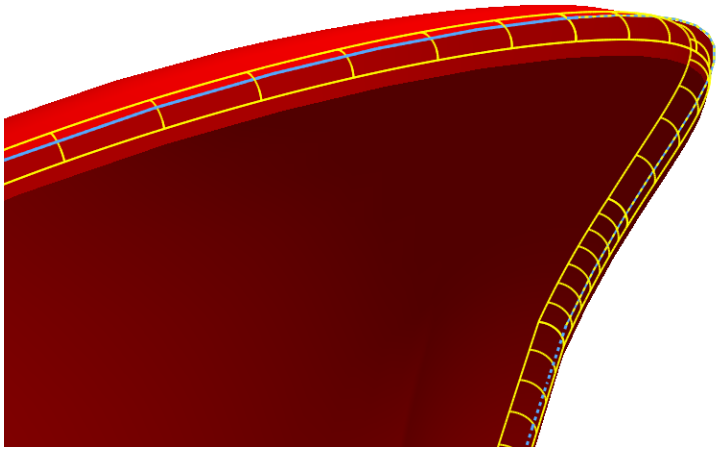
Create fillets on the edges of the chair

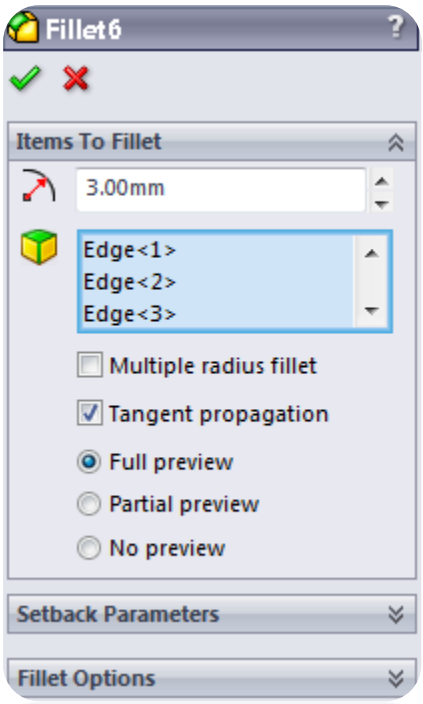
Go to: **Insert > Features > Fillet/Round** or click at the Fillet icon

Click the blue edges as shown in the picture

Change the Radius into 3 mm

Click OK





Save the part as Panton Chair

Congratulations, you just finished your own Panton Chair! ☺

Well, that's all for now. Feel free to share this document with your colleagues, family and friends.

I hope that you've learned something from this tutorial. Looking for more?

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Your SolidWorks Teacher,

Jan-Willem Zuyderduyn

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