SOLIDWORKS
SPOON EBOOK

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 Spoon in SolidWorks?

In today's tutorial, you will discover how to model a spoon in SOLIDWORKS. We will use some interesting features like the Projected Curve, the Surface Loft, the Move Body tool and many more.

Open a New Part with model units set to millimeters

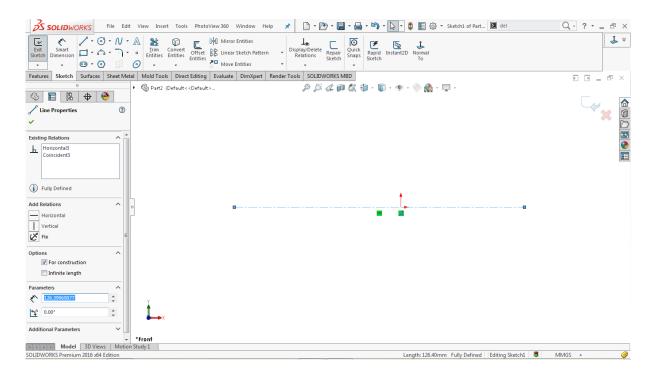Save the File with the following name: **Spoon.SLDPRT** 💾

Create a 2D Sketch on the Front Plane

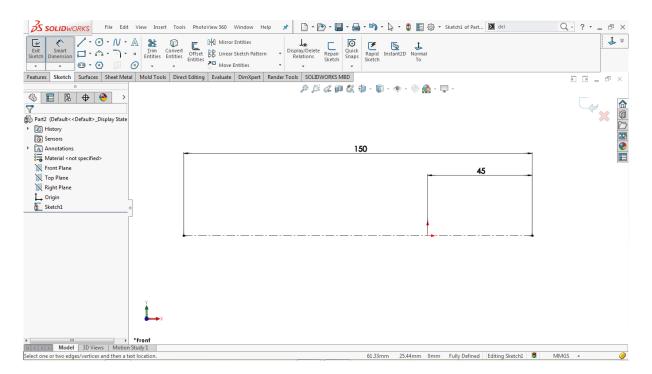Select the **Front Plane** from the Feature tree and click on the sketch icon

Go to: Tools > Sketch Entities > Centerline

Draw a horizontal centerline coincident to the origin, as shown in the image below

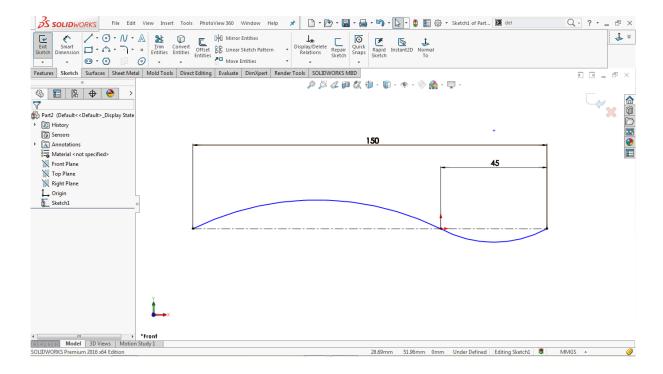**Go to: Tools > Dimensions > Smart**

Assign dimensions to the centerline, as shown in the image below

Go to: Tools > Sketch Entities > 3 Point Arc

Draw the first arc starting from the right end of the centerline up to the origin

Draw another arc in continues to the first arc up to the left end of the centerline, as shown in the image below
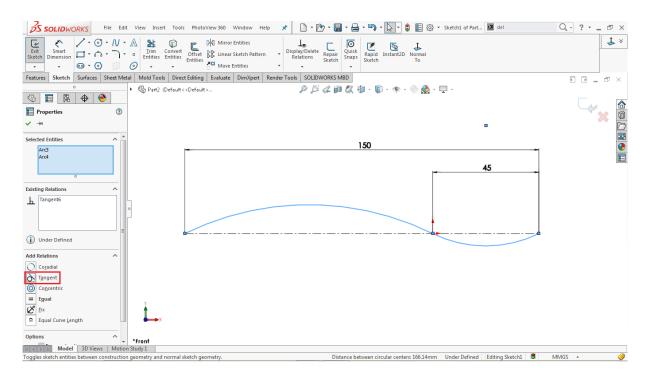
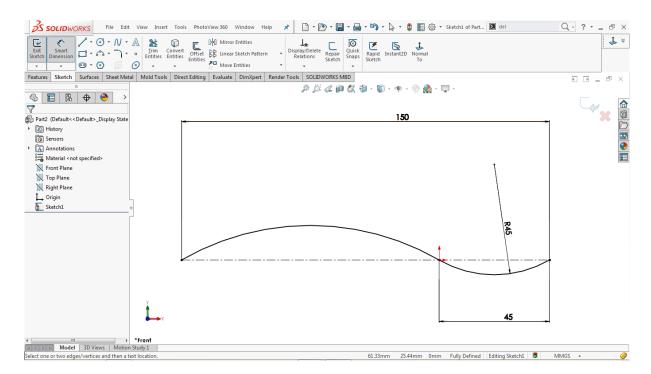Hold the **CTRL** key, select both arcs by clicking on each of them

Go to **Properties > Add Relations**

Click on the **tangent relation** ⟋ to add a tangency relation, as shown in the image below

Go to: **Tools > Dimensions > Smart** ⟍

Click on the smaller arc and assign the radius of **45mm**, as shown in the image below

Click on the Close sketch icon to close the sketch area.
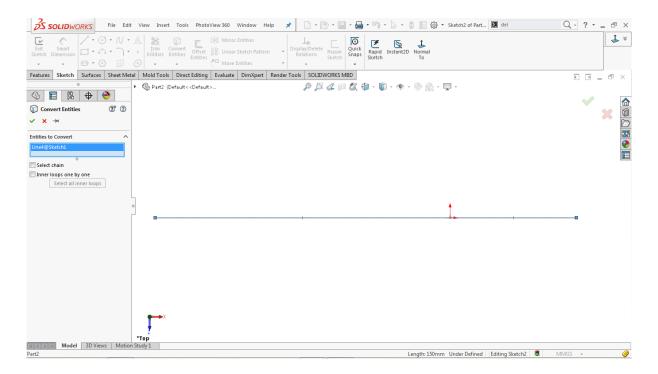
Create a 2D Sketch on the Top Plane

Select the **Top Plane** from the Feature tree and click on the sketch icon

Go to: Tools > Sketch Tools > Convert Entities

Go to the sketch area and click on **Sketch1,** as shown in the image below

Click **OK** ✔ to complete the operation.

Click on the converted horizontal line

Go to **Properties > Options**

Tick the **For Construction** checkbox to convert it into a construction line

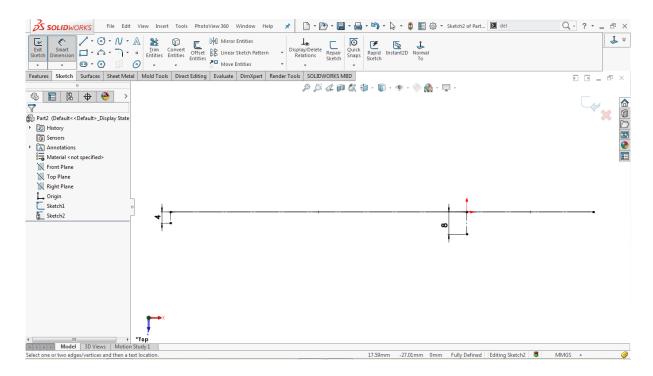**Go to: Tools > Sketch Entities > Centerline**

Draw a vertical centerline coincident to the right endpoint of the horizontal centerline line

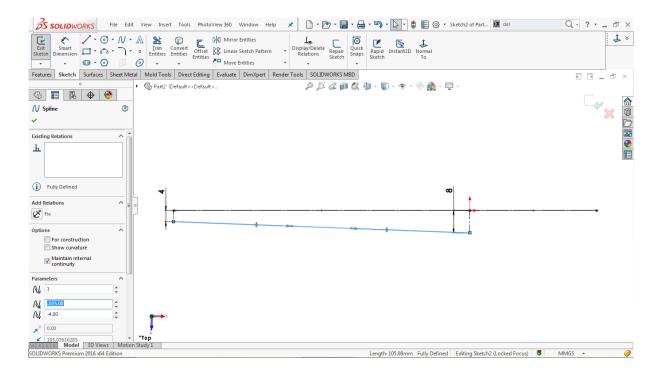Draw another vertical centerline coincident to the origin

Go to: Tools > Dimensions > Smart

Click on each centerline and assign the dimensions, as shown in the image below

Go to: **Tools > Sketch Entities > Spline** $\mathcal{N}$

Draw a two-point spline by using the lower endpoints of the vertical centerlines, as shown in the image below

Click on the spline and drag both handles one by one to activate them
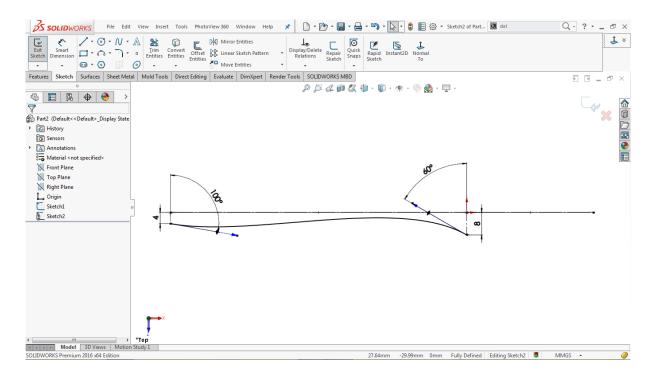
Go to: Tools > Dimensions > Smart

Hold the **CTRL** key, select the left spline handle and the corresponding vertical centerline of **4mm**

Assign an angular dimension of **100 degrees** between them

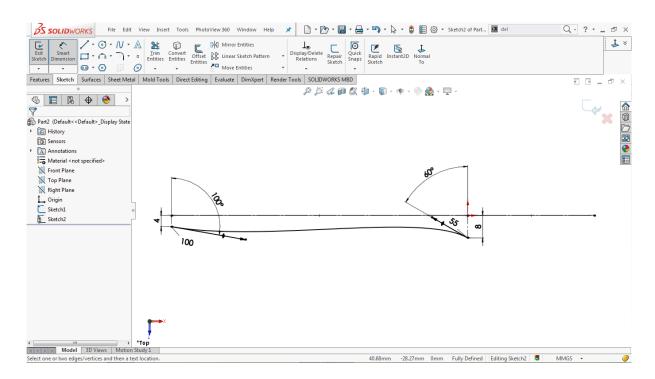Select the right spline handle and the corresponding vertical centerline of **8mm**

Assign an angular dimension of **60 degrees,** as shown in the image below

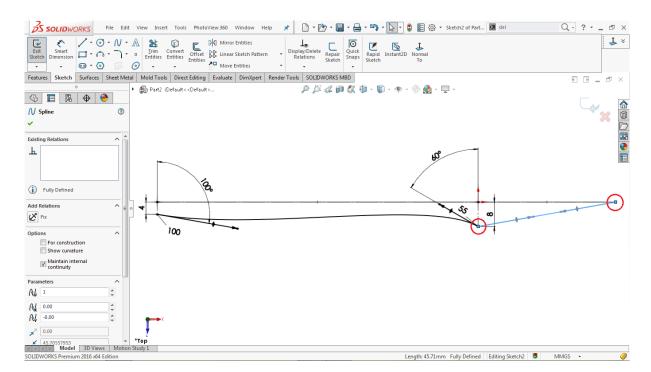Click on the **left handle** and assign the handle length of **100mm**

Click on the **right handle** and assign the handle length of **115mm,** as shown in the image below

Go to: Tools > Sketch Entities > Spline

Draw a two-point spline between the red encircled points, as shown in the image below

Click on the new spline and drag both handles one by one to activate them
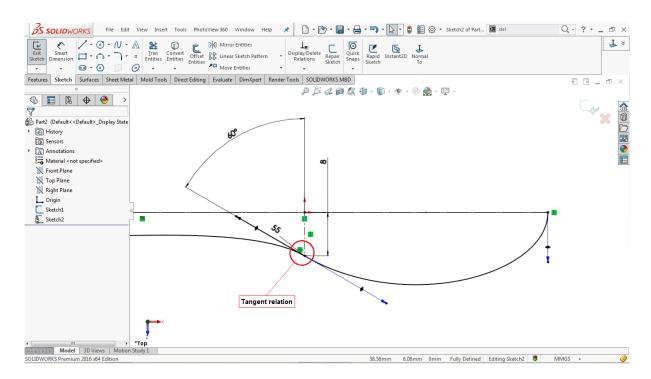
Select the **right handle** of the spline

Go to **Property Manager > Spline > Add Relations**

Then, assign a **vertical relation** | between them

 Hold the **CTRL** key, select the left spline and the right spline

Go to **Property Manager > Spline > Add Relations**

Click on the **tangent relation** to add a tangent relation between the lines, as shown in the image below

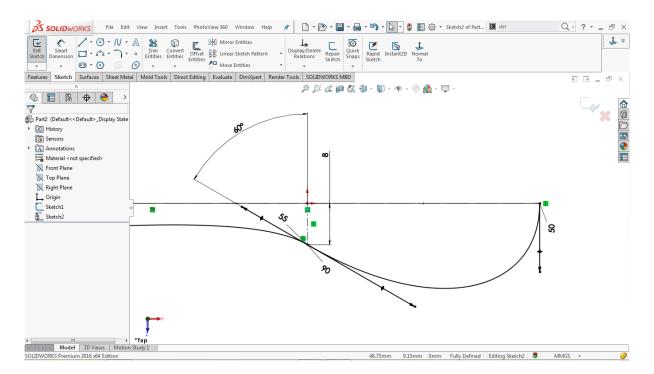**Go to: Tools > Dimensions > Smart**

Click on the **left handle** of the new spline

Assign the handle length of **90 mm**

Click on the **right handle** of the new spline

Assign the handle length of **50mm,** as shown in the image below

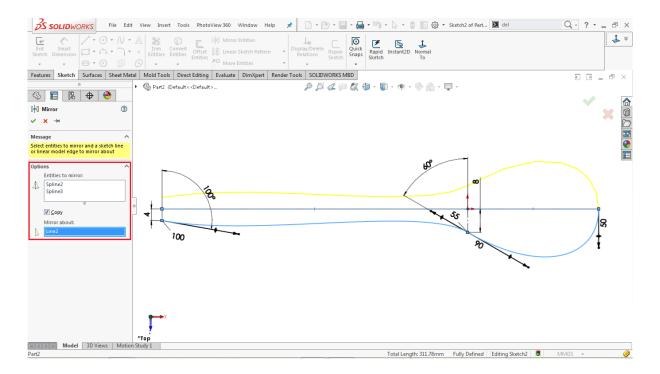**Go to: Tools > Sketch Tools > Mirror** 🔲 or click on the **Mirror Entities** directly

Go to the property manager, click on the **Entities to Mirror** 🔺 input box

Select both splines from the 2D sketch

Go to the property manager, click on the **Mirror About** 🔺 input box

Select the horizontal construction line, as shown in the image below

Click **OK** ✔ to complete the operation.
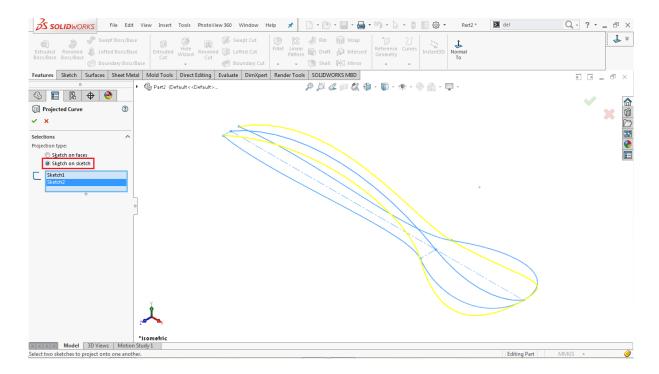
Use the Project Curve Tool:

Go to: Insert > Curve > Projected 🔲

Go to the property manager and select **Sketch on Sketch** as a projection type

Go to the modelling area and click on the **Sketch1** and **Sketch2**

Then a projected curve is generated in yellow which is shown in the image below
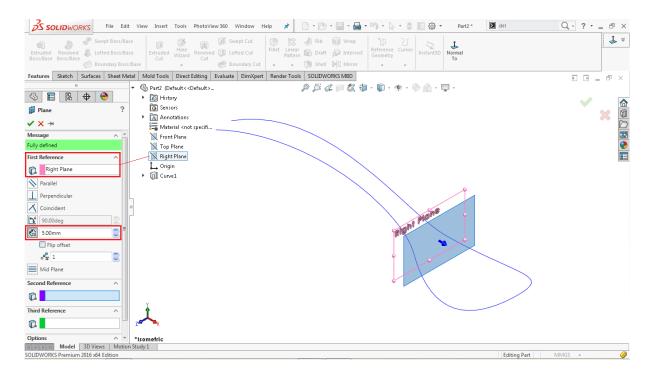

Click **OK** ✔ to complete the operation.

Make a Reference Plane (Plane1)

Go to: Insert > Reference Geometry > Plane

Select the **Right Plane** from the feature tree as a **First Reference**

Scroll down in the property manager and assign an offset value of **5mm**, as shown in the image below
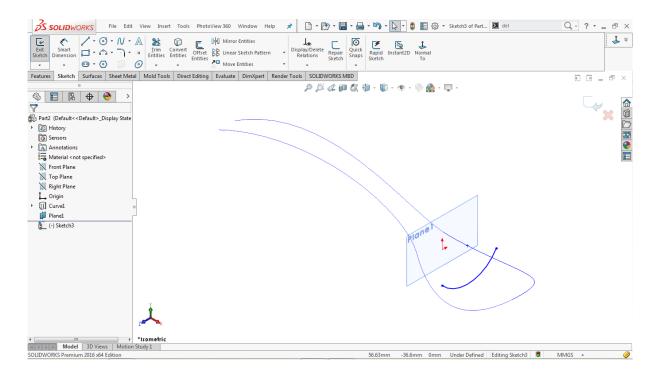

Click **OK** to complete the operation.

Create a 2D Sketch on Plane1

Select **Plane1** from the Feature tree and click on the sketch icon

Go to: Tools > Sketch Entities > 3 Point Arc

Draw a concave **3 point arc** anywhere on the **Plane1,** as shown in the image below

Hold the **CTRL** key, select the left endpoint of the arc and the corresponding edge of the projected curve

Go to **Properties > Add Relations**

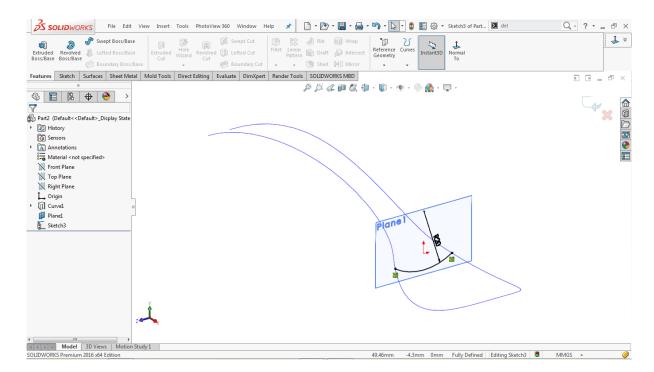Click on the **pierce relation** to pierce the endpoint to the projected curve

Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

Go to **Properties >> Add Relations**

Click on the **pierce relation** to pierce the second endpoint to the projected curve

Go to: Tools > Dimensions > Smart

Click on the arc and assign a radius of **20mm,** as shown in the image below

Click on Close sketch ⤵ to close the sketch area.

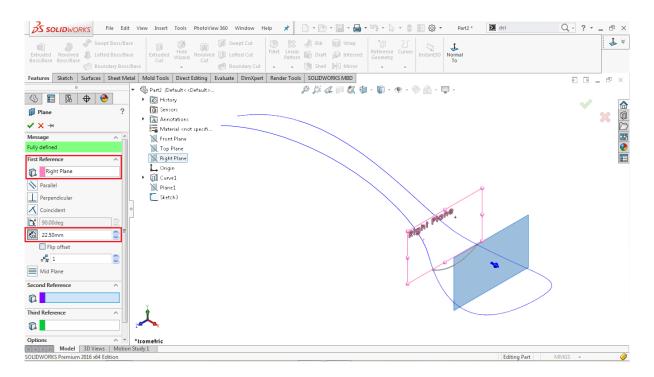Go to the feature tree, click on the **Plane1** and hit the **Eye** 👁 icon to hide that plane

Make a Reference Plane (Plane2)

Go to: **Insert > Reference Geometry > Plane** 📐

Select the **Right Plane** from the feature tree as a **First Reference**

Scroll down in the property manager and assign an offset value of **22.5mm**, as shown in the image below

Click **OK** to complete the operation.
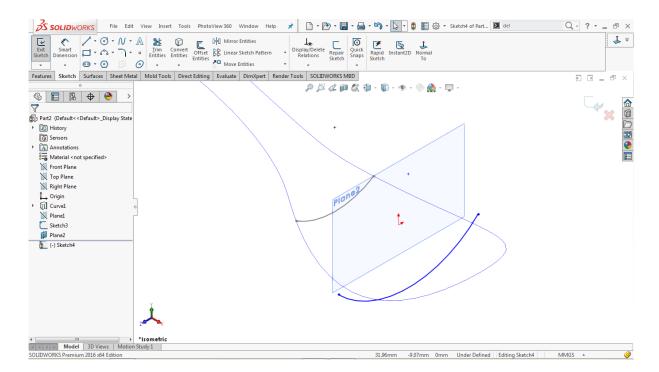
Create a 2D Sketch on Plane2

Select **Plane2** from the Feature tree and click on the sketch icon

Go to: **Tools > Sketch Entities > 3 Point Arc**

Draw a concave **3 point arc** anywhere on **Plane2,** as shown in the image below

Hold the **CTRL** key, select the left endpoint of the arc and the corresponding edge of the projected curve

Go to **Properties > Add Relations**
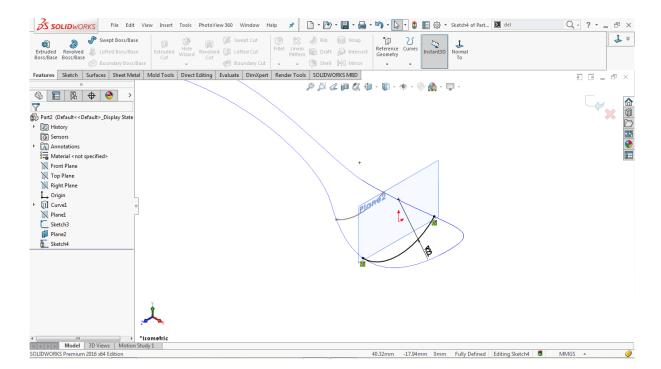
Click on the **pierce relation**

Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

Go to **Properties >> Add Relations**

Click on the **pierce relation**

Go to: Tools > Dimensions > Smart

Click on the arc and assign a radius of **22mm,** as shown in the image below

Click on the Close sketch icon to close the sketch area.

Go to the feature tree, click on **Plane2** and hit the **Eye** icon to hide that plane

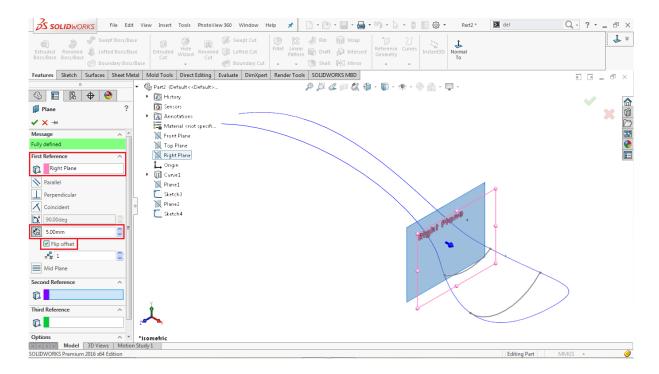**Make a Reference Plane (Plane3)**

Go to: Insert > Reference Geometry > Plane

Select the **Right Plane** from the feature tree as a **First Reference**

Scroll down in the property manager and assign the offset value of **5mm,**

And tick the **Flip offset** checkbox, as shown in the image below
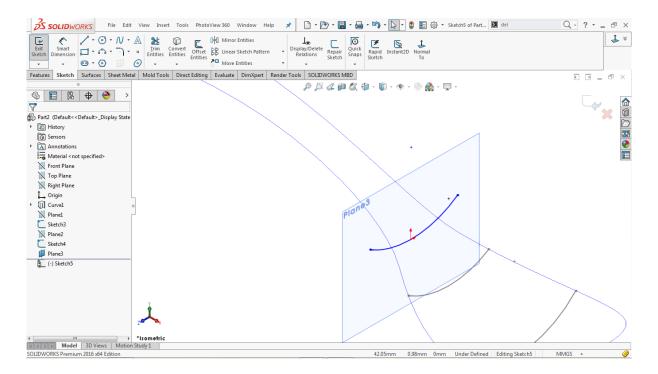

Click **OK** ✔ to complete the operation.

Create a 2D Sketch on Plane3

Select **Plane3** from the Feature tree and click on the sketch icon ⌨

Go to: Tools > Sketch Entities > 3 Point Arc ⌓

Draw a concave **3point arc** anywhere on the **Plane3,** as shown in the image below

Hold **CTRL** key, select the left endpoint of the arc and the corresponding edge of the projected curve

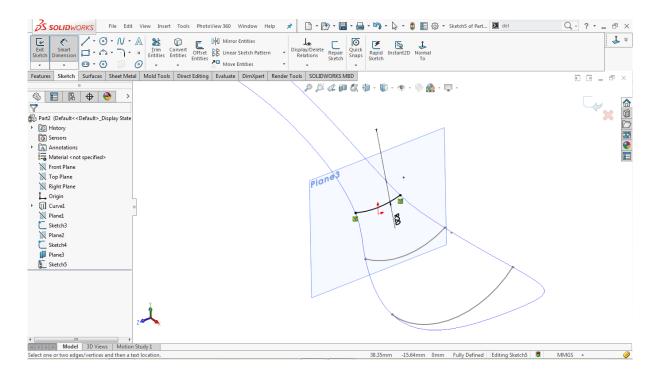Go to **Properties > Add Relations**

Click on the **pierce relation**

Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

Go to **Properties > Add Relations**

Click on the **pierce relation**

Go to: Tools > Dimensions > Smart

Click on the arc and assign a radius of **20mm,** as shown in the image below

Click on the Close sketch icon to close the sketch area.

Go to the feature tree, click on the **Plane3** and hit the **Eye** icon to hide that plane
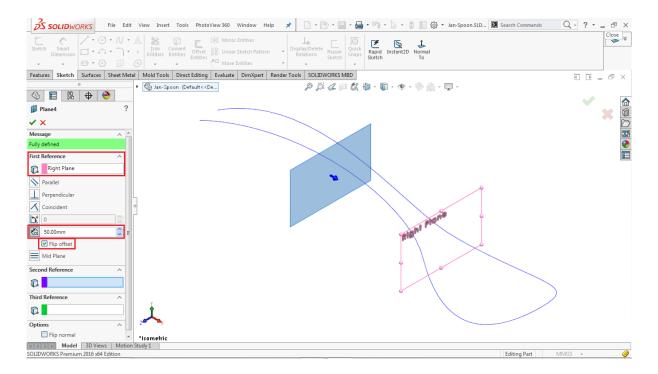
Make a Reference Plane (Plane4)

Go to: Insert > Reference Geometry > Plane

Select the **Right Plane** from the feature tree as a **First Reference**

Go down in the property manager and assign an offset value of **50mm,**

Tick the **Flip offset** checkbox, as shown in the image below
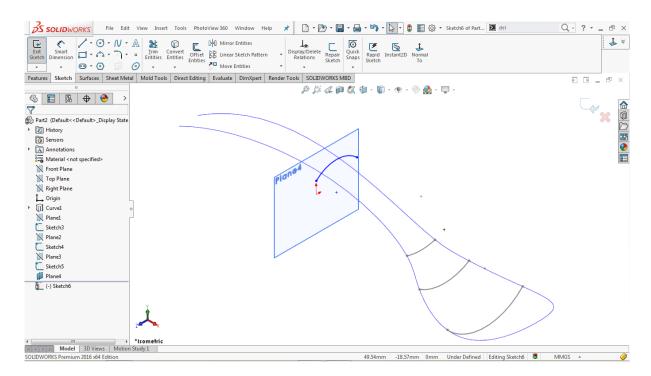

Click **OK** ✔ to complete the operation.

Create a 2D Sketch on Plane4

Select **Plane4** from the Feature tree and click on the sketch icon

Go to: Tools > Sketch Entities > 3 Point Arc

Draw a convex **3 point arc** anywhere on the **Plane4,** as shown in the image below

Hold **CTRL** key, select the left endpoint of the arc and the corresponding edge of the projected curve

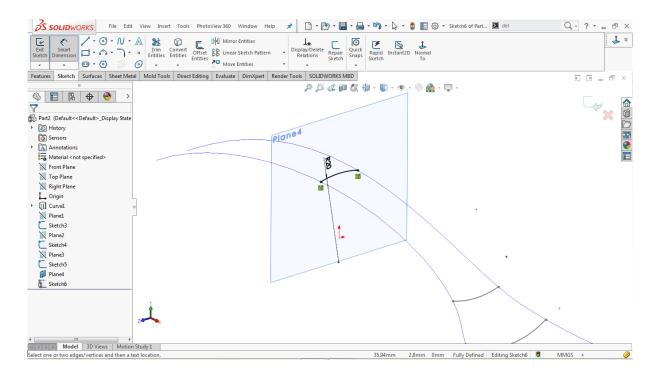Go to **Properties > Add Relations**

Click on the **pierce relation**

Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

Go to **Properties > Add Relations**

Click on the **pierce relation**

Go to: Tools > Dimensions > Smart

Click on the arc and assign a radius of **20mm,** as shown in the image below

Click on the Close sketch icon ⬐ to close the sketch area.

Go to the feature tree, click on **Plane4** and hit the **Eye** 👁 icon to hide that plane
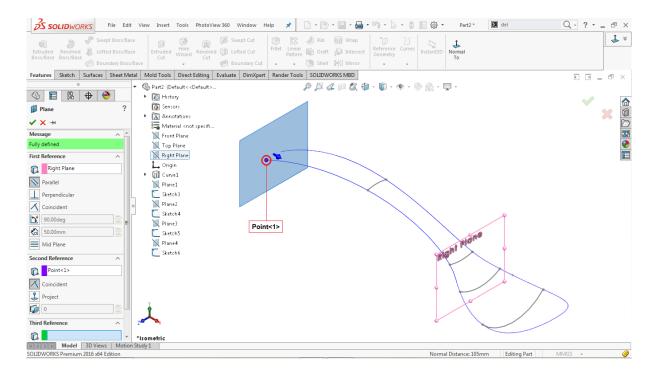
Make a Reference Plane (Plane5)

Go to: Insert > Reference Geometry > Plane 🗔

Select the **Right Plane** from the feature tree as a **First Reference**

Select the **Point<1>** from the modeling area as a **Second Reference**, as shown image below
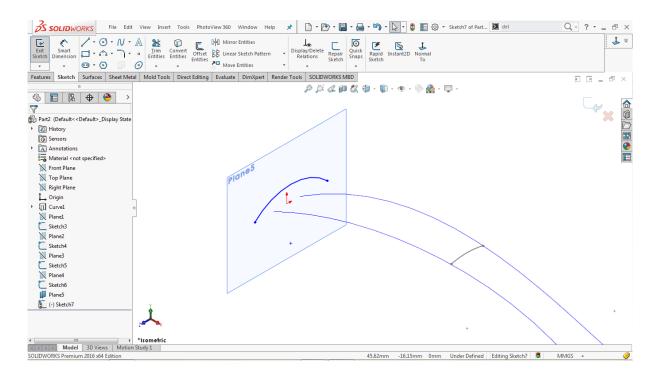

Click **OK** ✓ to complete the operation.

Create a 2D Sketch on Plane5

Select **Plane5** from the Feature tree and click on the sketch icon

Go to: Tools > Sketch Entities > 3 Point Arc

Draw a convex **3 point arc** anywhere on **Plane5,** as shown in the image below

Hold **CTRL** key, select the left endpoint of the arc and the corresponding edge of the projected curve

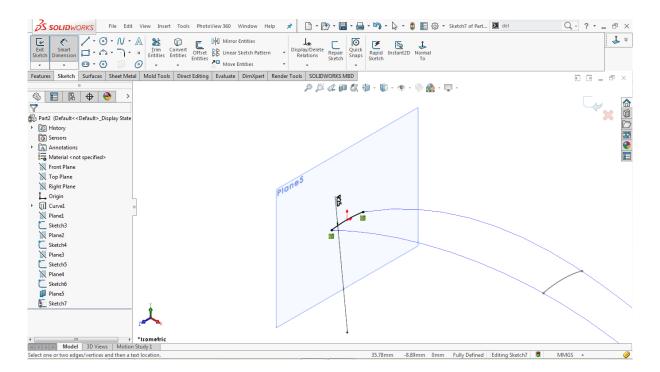Go to **Properties > Add Relations**

Click on the **pierce relation**

Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

Go to **Properties > Add Relations**

Click on the **pierce relation**

Go to: Tools > Dimensions > Smart

Click on the arc and assign a radius of **25mm,** as shown in the image below

Click on the Close sketch icon to close the sketch area.

Go to the feature tree, click on the **Plane5** and hit the **Eye** icon to hide that plane

Use the Fill Surface tool

Go to: Insert > Surface > Fill

Go to **Property Manager > Patch Boundary**
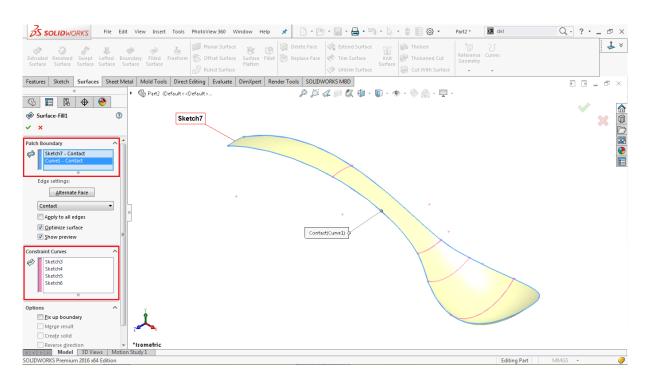
Click on the **Patch Boundary** input box

Go to the modeling area and select the blue **Projected Curve** and **Sketch7**

Go to **Property Manager > Constraint Curves**

Click on the **Constraint Curves** input box

Go to the **Feature Tree** and select **Sketch3, Sketch4, Sketch5,** and **Sketch6,** as shown in the image below

Leave the other parameters as default

Click **OK** ✔ to complete the surface fill operation.
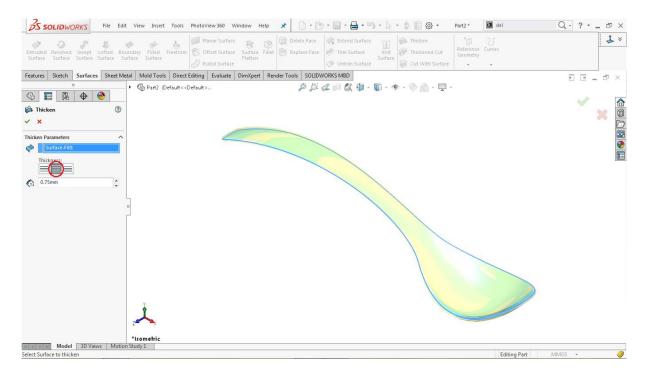
Create Thicken

Go to: Insert > Boss/ Base > Thicken

Go to the modeling area, select the filled surface as a **Thicken parameter**

Go to the property manager, select the '**Both side Thicken'** option

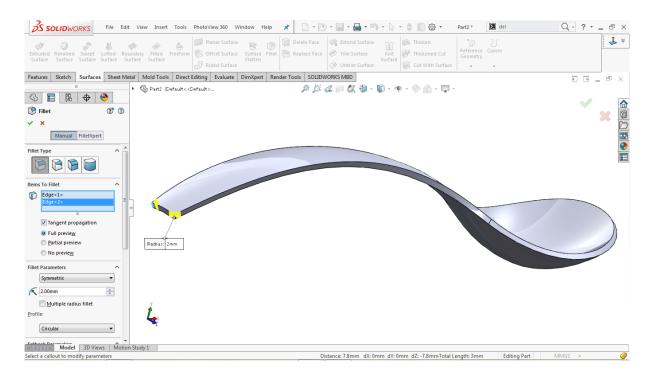Assign the thickness value of ⬚ **0.75mm,** as shown in the image below

Click **OK** ✔ to complete the thicken operation.

Use the Fillet

Go to: **Insert > Features > Fillet/Round** ⬚

Select both tiny blue edges at the end of the spoon, as shown in the image below

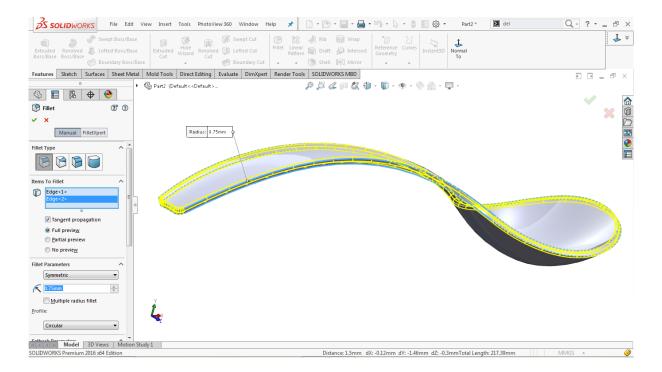Go to the property manager and assign the Fillet radius ↖ of **2mm**

Click **OK** ✔ to complete the fillet operation.

Make another Fillet

Go to: **Insert > Features > Fillet/Round** ⬚

Select both edges of the spoon, as shown in the image below

Go to the property manager and assign a Fillet radius ↖ of **0.75mm**

Click **OK** ✔ to complete the fillet operation.

Use the Move/ Copy Body feature

Go to: Insert > Features > Move/ Copy… 🗗

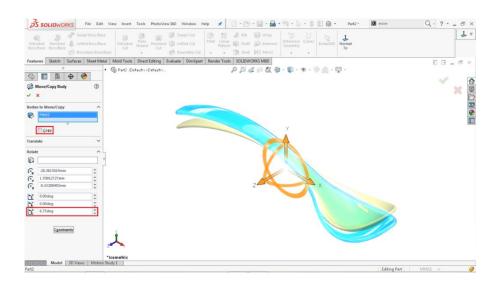Select the existing solid body from the modeling area

Go to the property manager and click on the **Rotate** option

Go to the feature tree and select **Origin** as a rotating reference

Assign the rotation values about the **X, Y** and **Z-Axis** to align the spoon with the ground

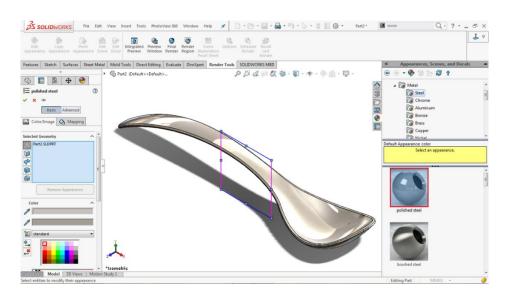↖X **X= 0 degrees,** ↖Y **Y= 6.75 degrees,** and ↖Z **Z= 0 degrees**

Leave the other parameters as default

Click **OK** ✔ to complete the operation.

Change the Appearance

Go to: PhotoView 360 > Edit Appearance…

Select **Appearances (color) > Metal > Steel > Polished Steel**

Double click on the **Polished Steel** icon

Click **OK** ✔ to apply the new appearance.

Save the File again 💾

Congratulations, you just modeled your own spoon in SolidWorks!

Did you like this free eBook and are you eager to learn more? Then you should really attend my free SolidWorks workshop.
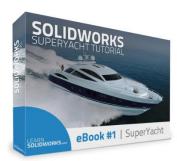
In this workshop, you are going to discover how to become a SolidWorks Pro in days instead of years without boring practice, expensive training classes, or any pointless theory:

Click here to attend my free SolidWorks workshop.

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✓ 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**.

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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