Université de Biskra

Faculté des Sciences et de la Technologie

Département de Génie Mécanique

TP De CFAO

Le 05/04/2020

Ce document contient les différentes étapes pour réaliser une cuillère (spoon) par le logiciel que nous avons l'habitude d'utiliser en salle : Solidworks. J'ai préféré garder la langue du texte original du document (anglais)

Travail demandé : suivre cette méthode pour réaliser cette pièces a domicile et m'envoyer votre travail, spoon.sldprt par courriel.

N'hésitez pas à poser toute question.

L'enseignant :

M.Benmachiche

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

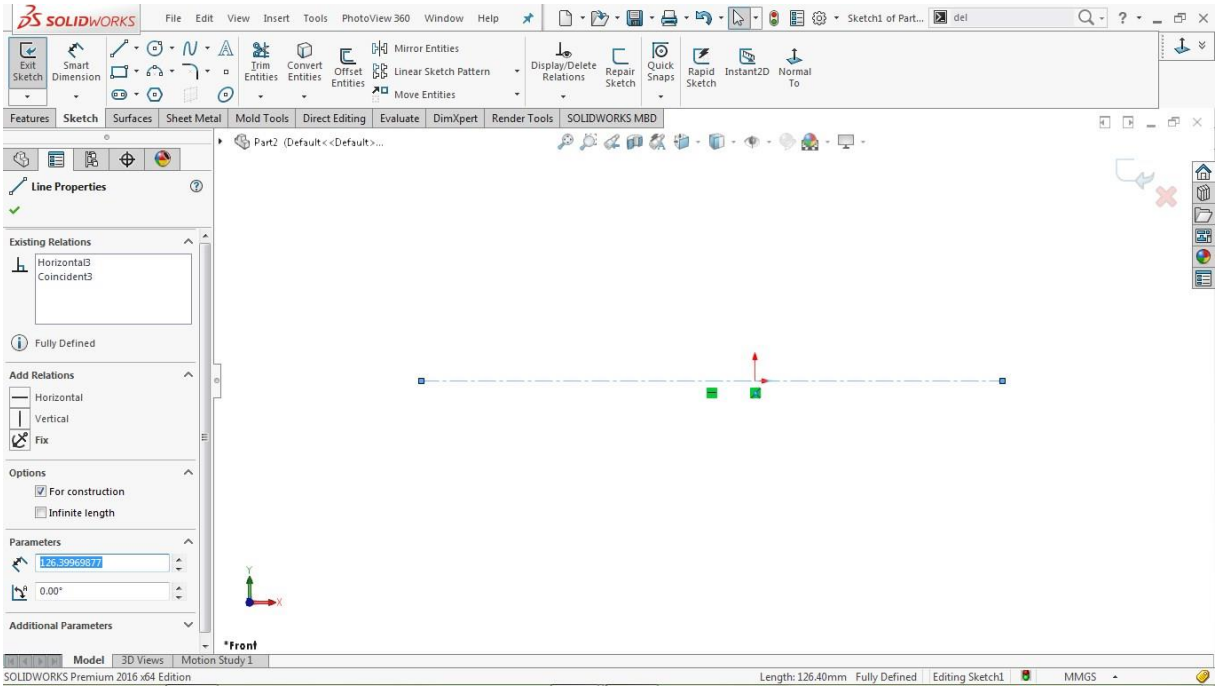
Save the File with the following name: **Spoon.SLDPRT**

Create a 2D Sketch on the Front Plane

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

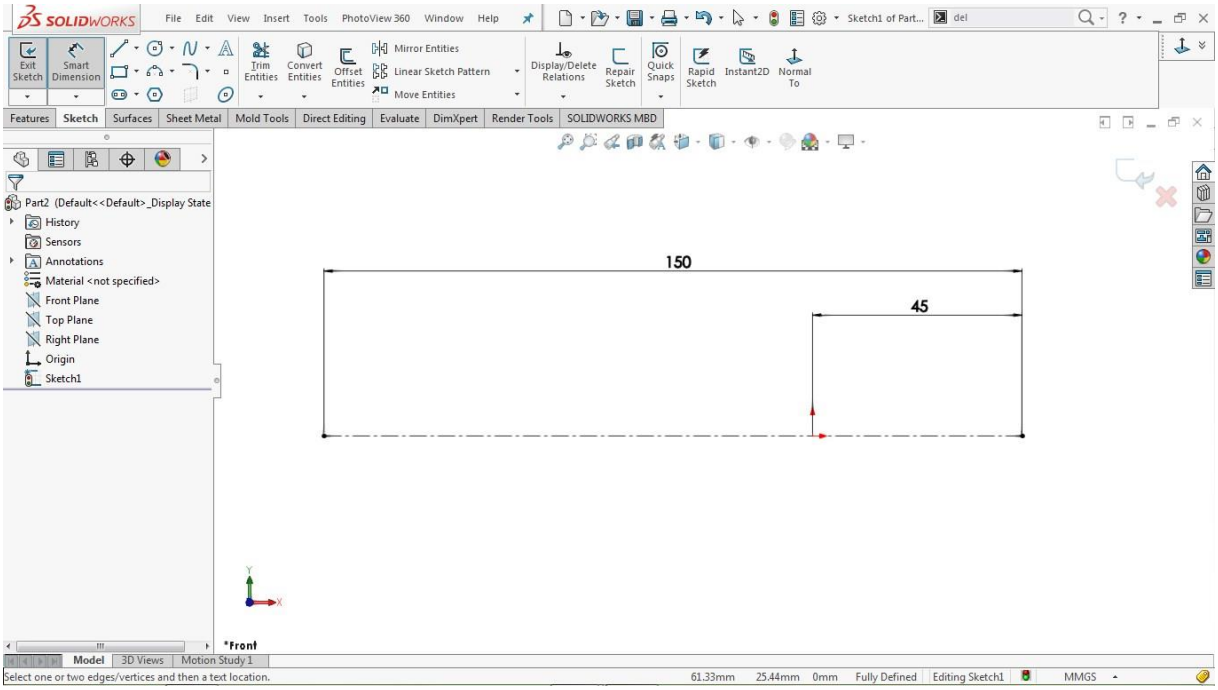
Go to: Tools > Sketch Entities > Centerline

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



Go to: Tools > Dimensions > Smart

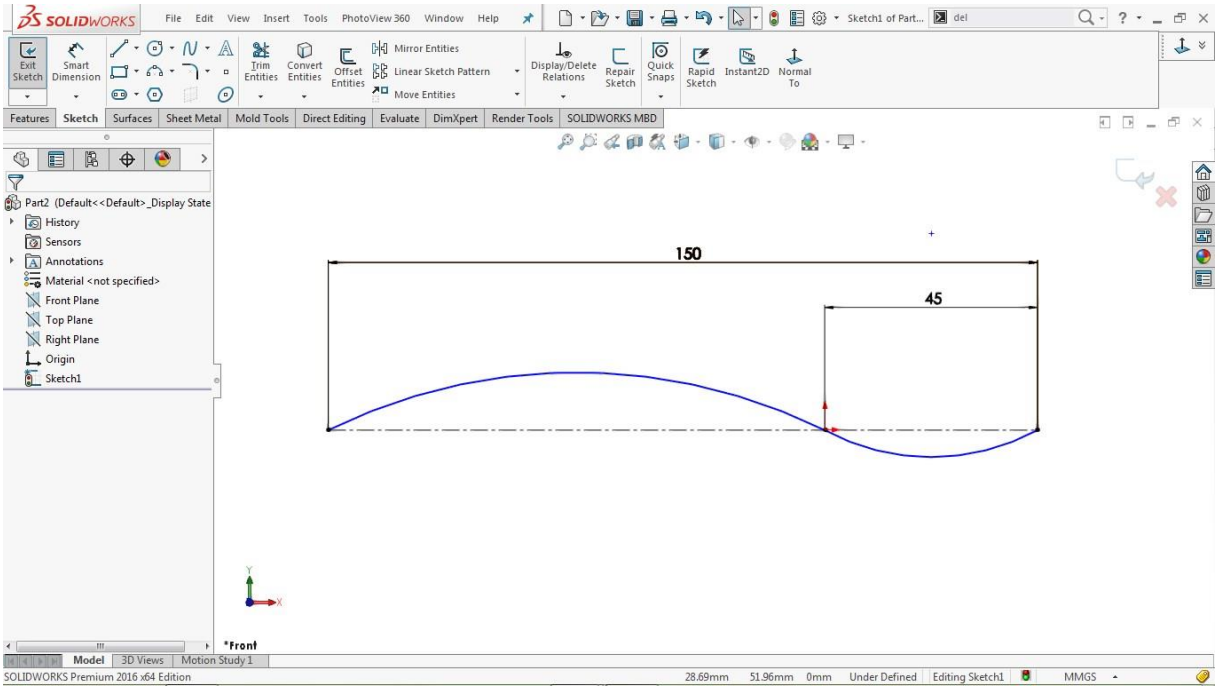
Assign dimensions to the centerline, as shown in the image below



Go to: Tools > Sketch Entities > 3 Point Arc

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

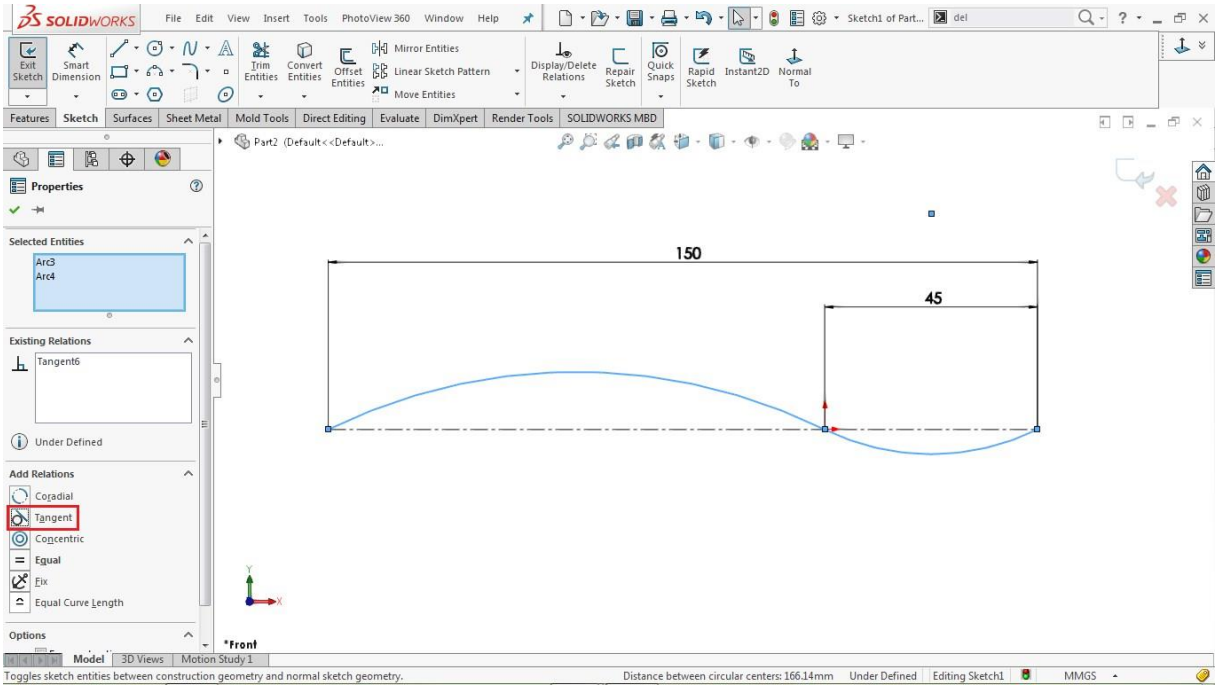
Draw another arc in continues to the first arc up to the left end of the centerline, as shown in the image below



Hold the **CTRL** key, select both arcs by clicking on each of them

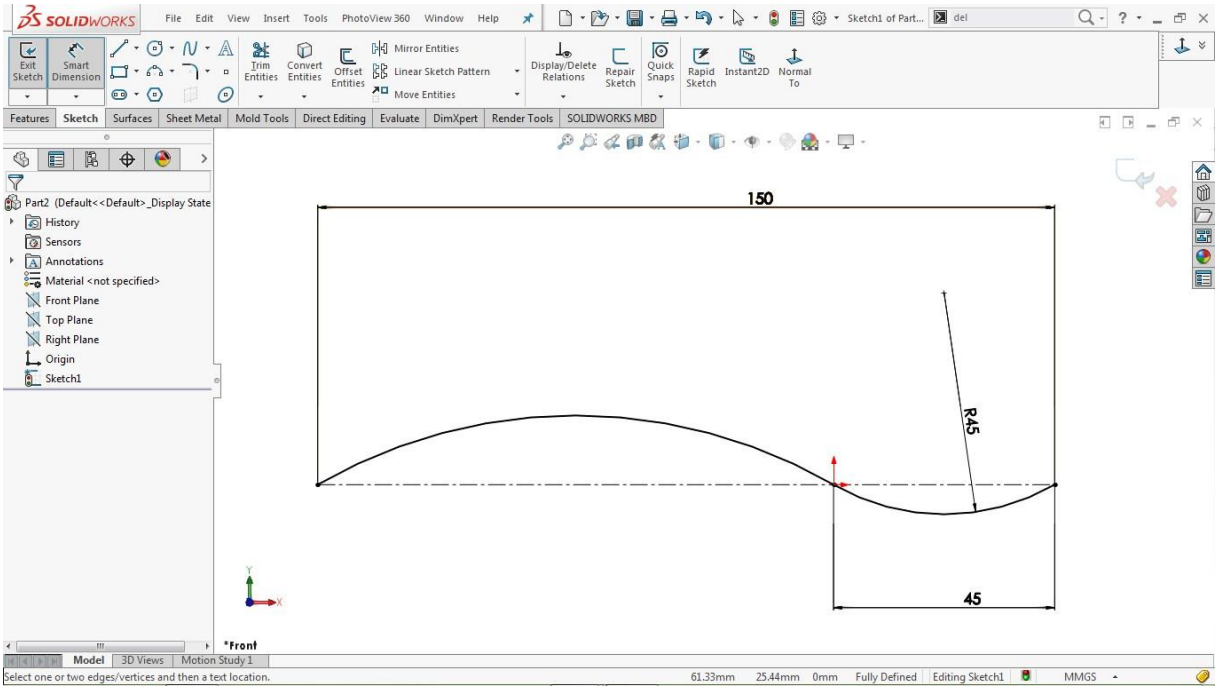
Go to **Properties > Add Relations**

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



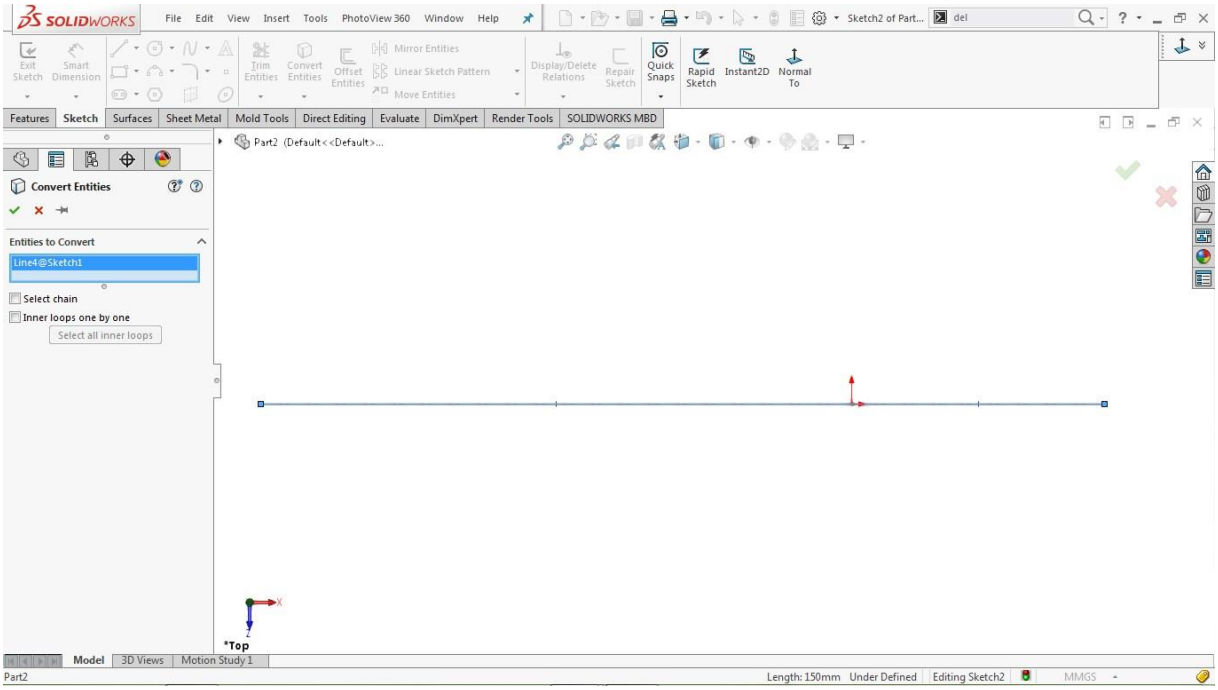
Click on the Close sketch icon  to close the sketch area.

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

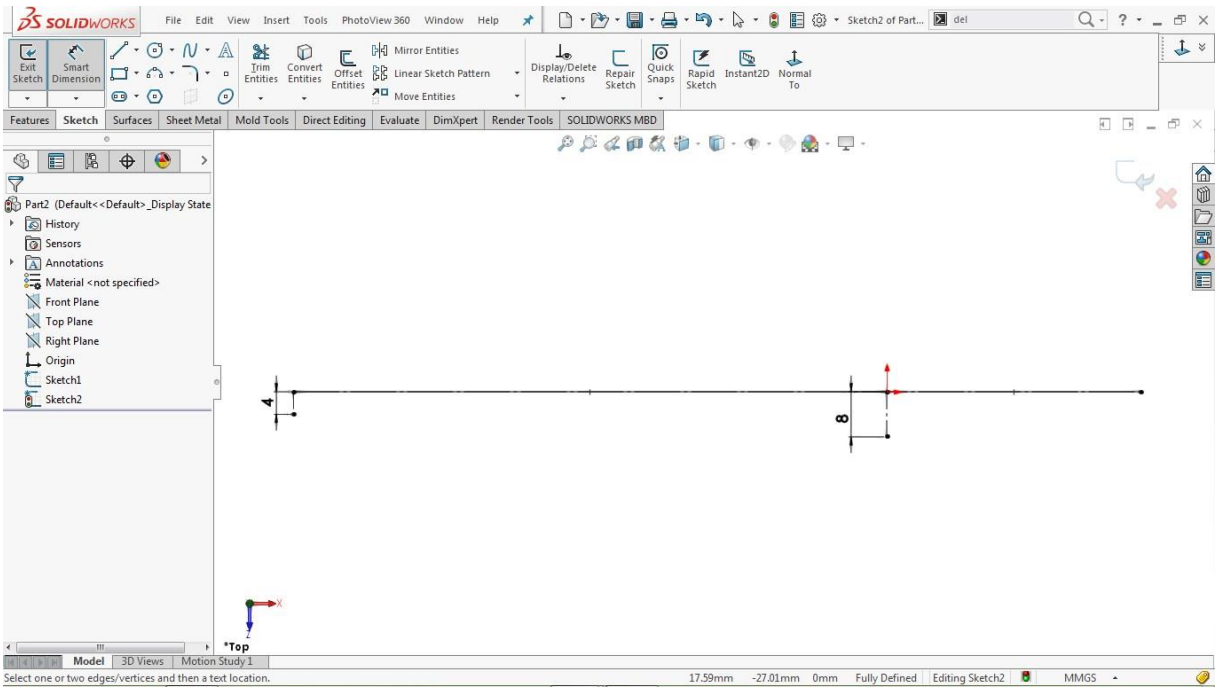
Go to: Tools > Sketch Entities > Centerline

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

Draw another vertical centerline coincident to the origin

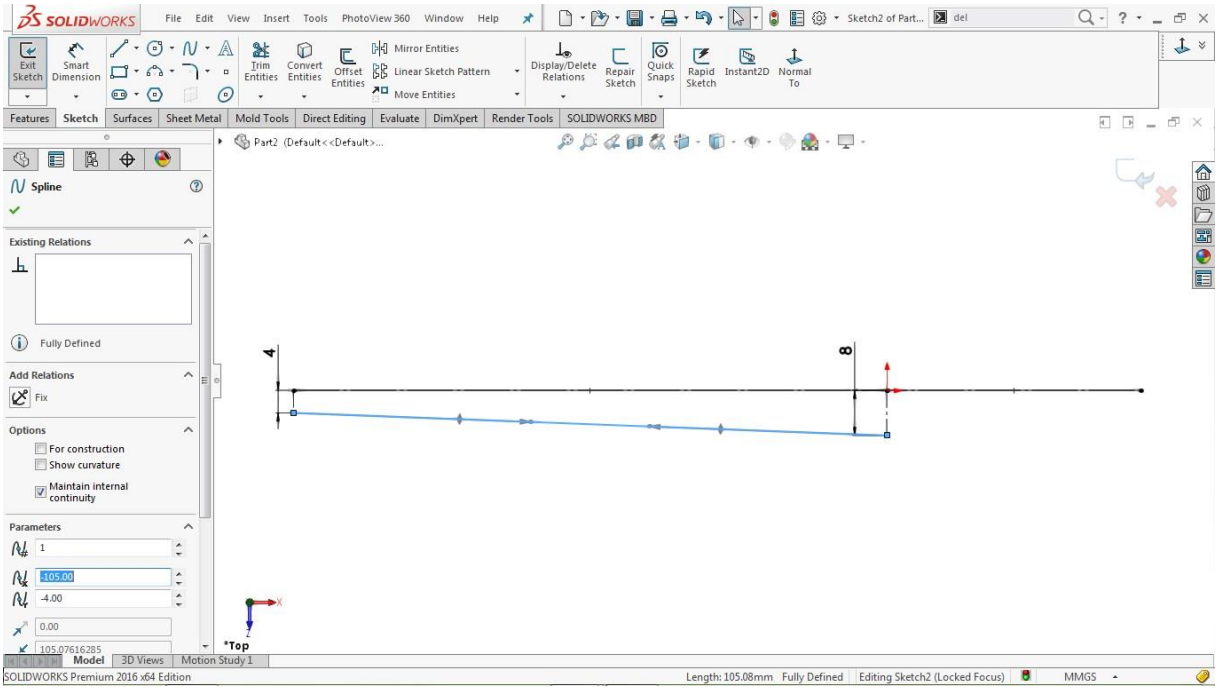
Go to: Tools > Dimensions > Smart

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



Go to: Tools > Sketch Entities > Spline

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

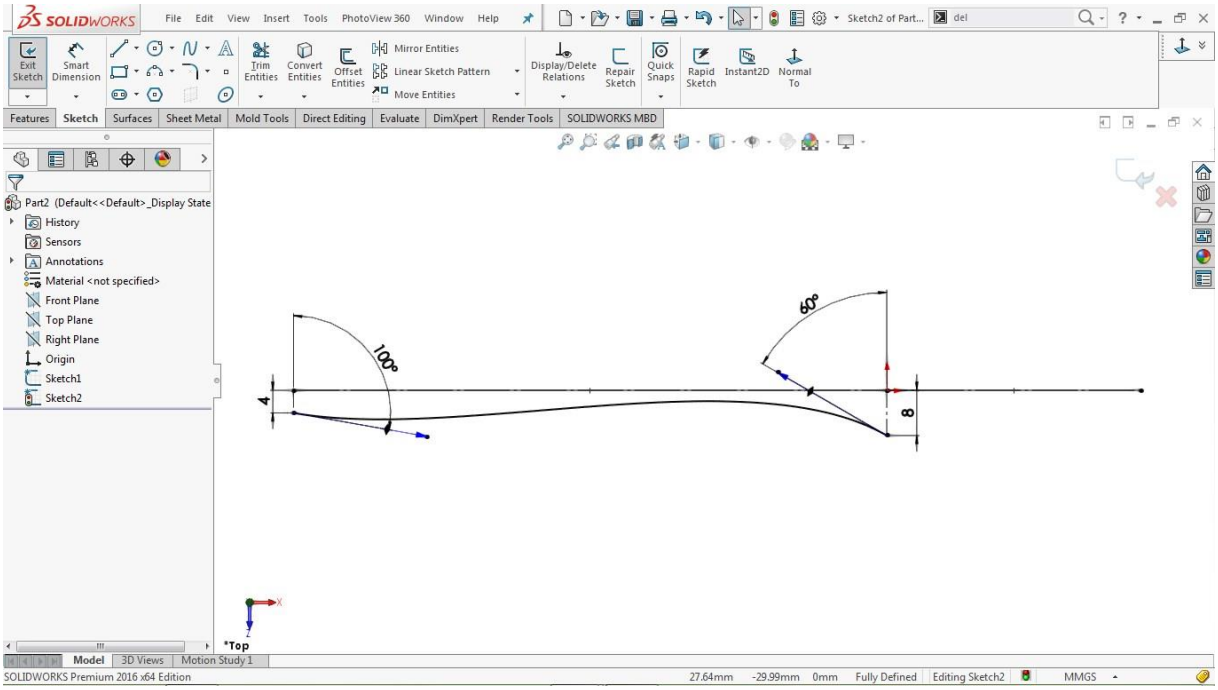
Go to: Tools > Dimensions > Smart 

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

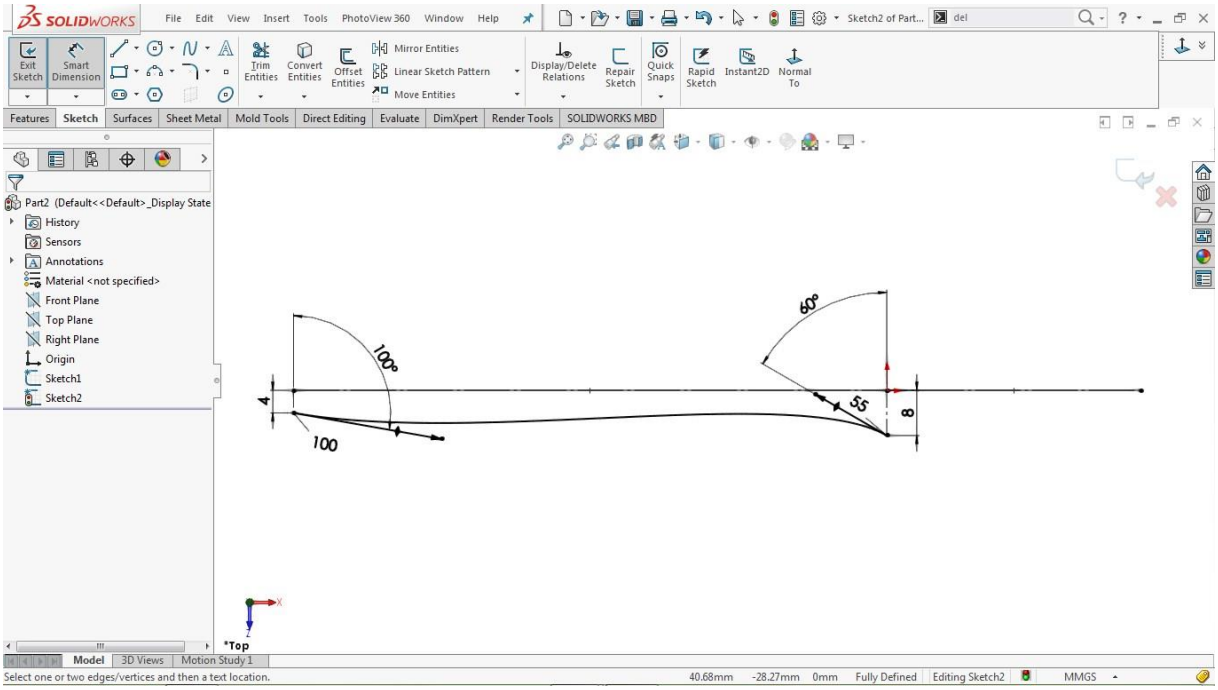
Select the right spline handle and the corresponding vertical centerline of **8mm**

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



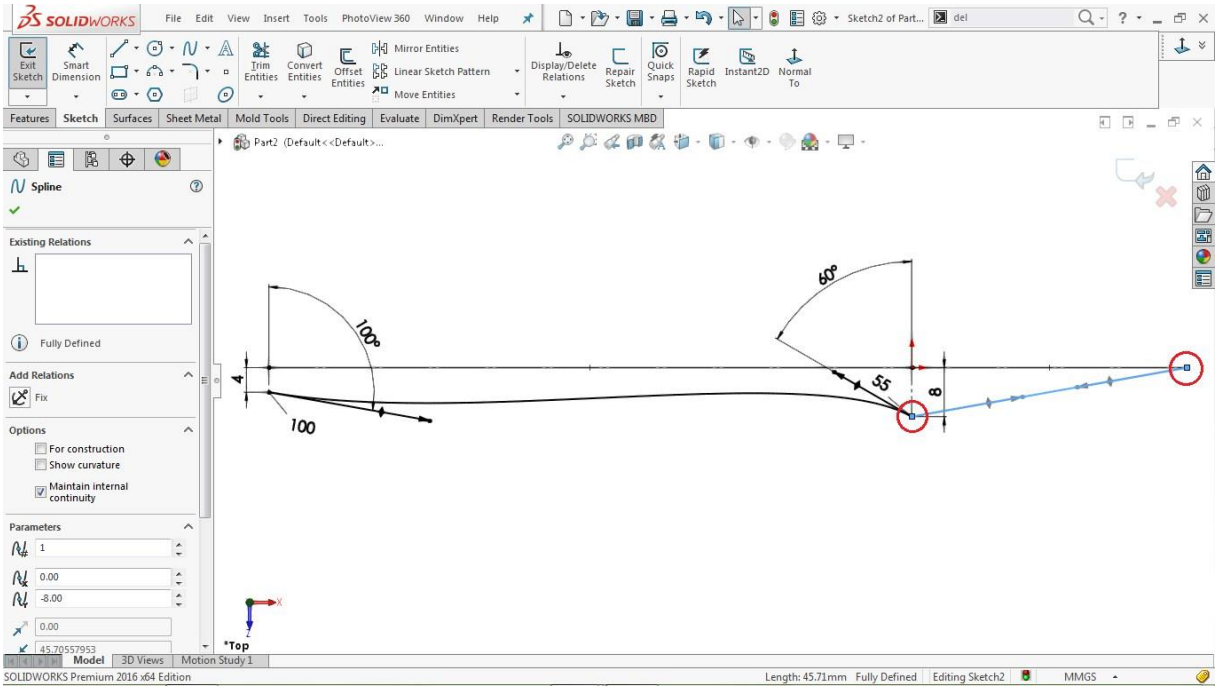
Click on the **left handle** and assign the handle length of **100mm**

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

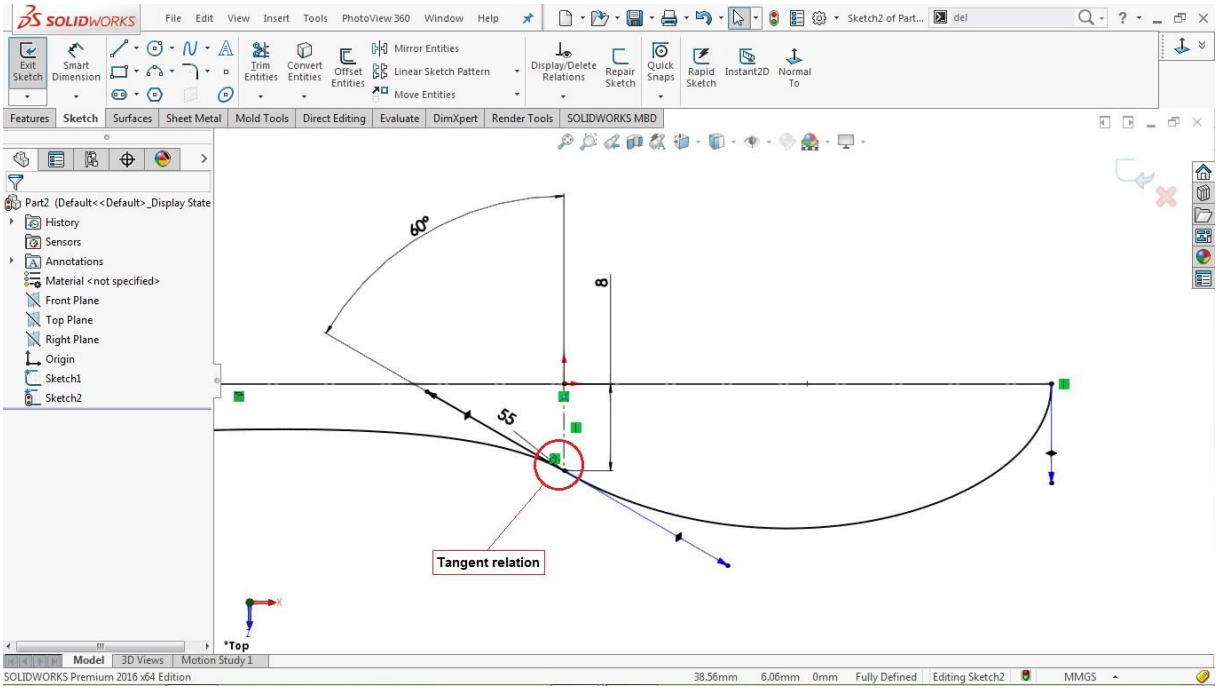
Go to **Property Manager > Spline > Add Relations**

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



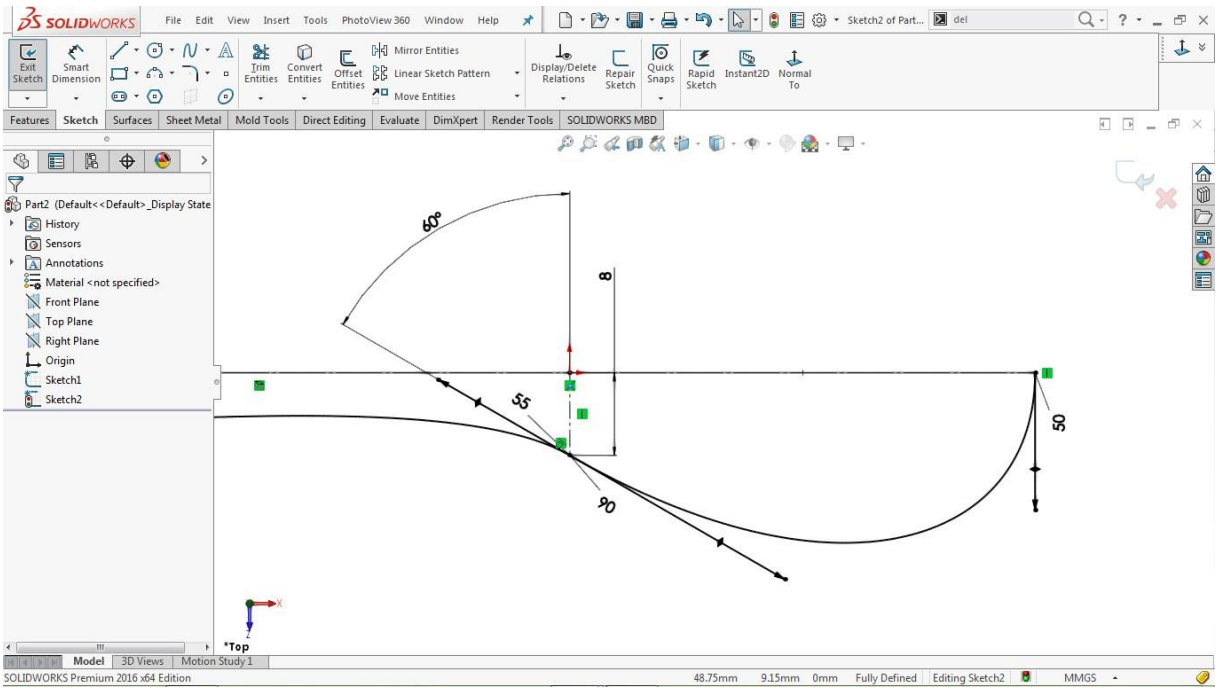
Go to: Tools > Dimensions > Smart

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



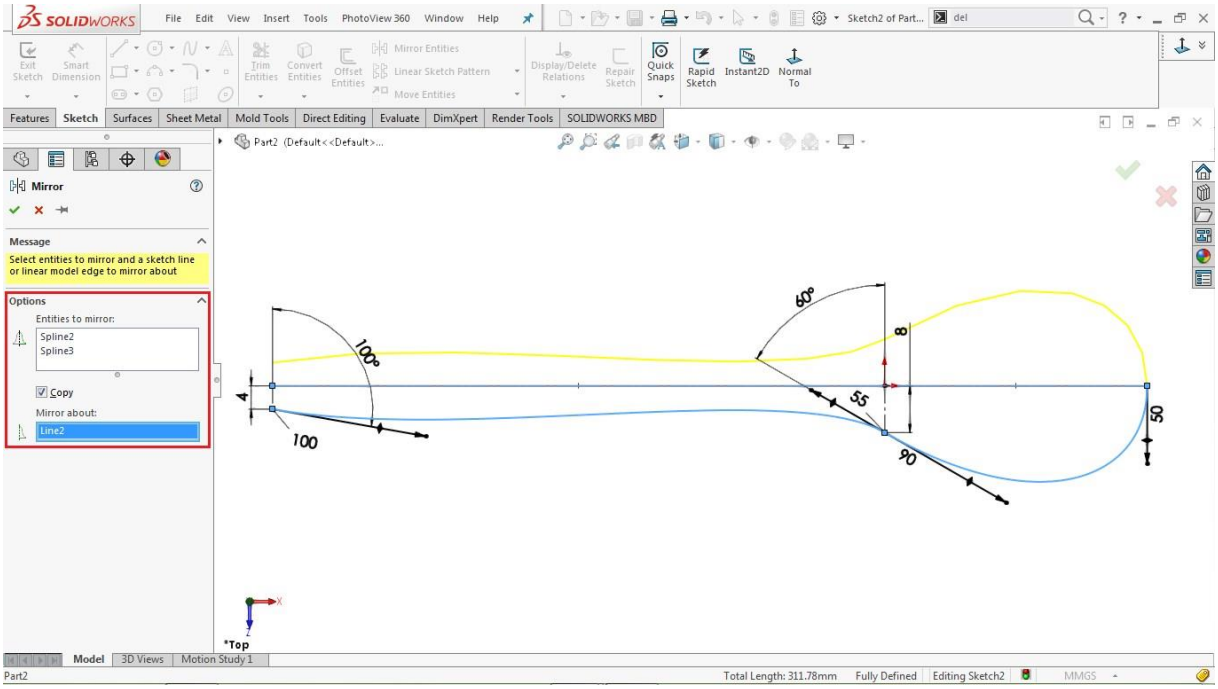
Go to: Tools > Sketch Tools > Mirror or click on the **Mirror Entities** directly

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.

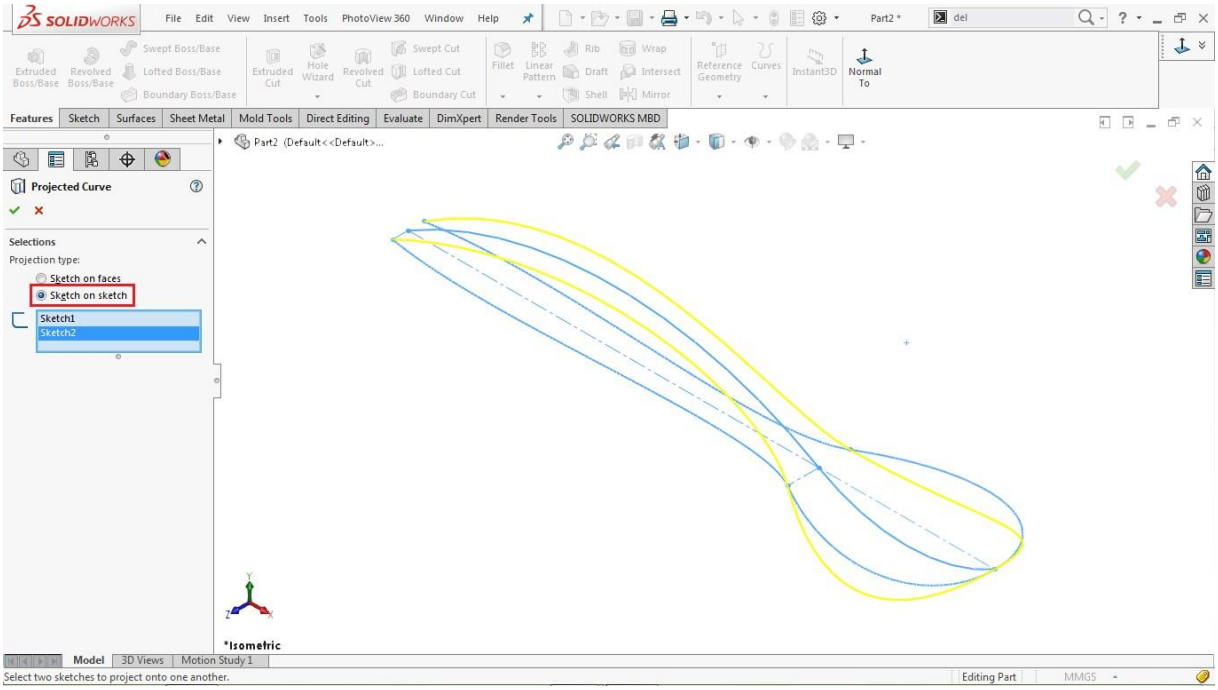
Use the Project Curve Tool:

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



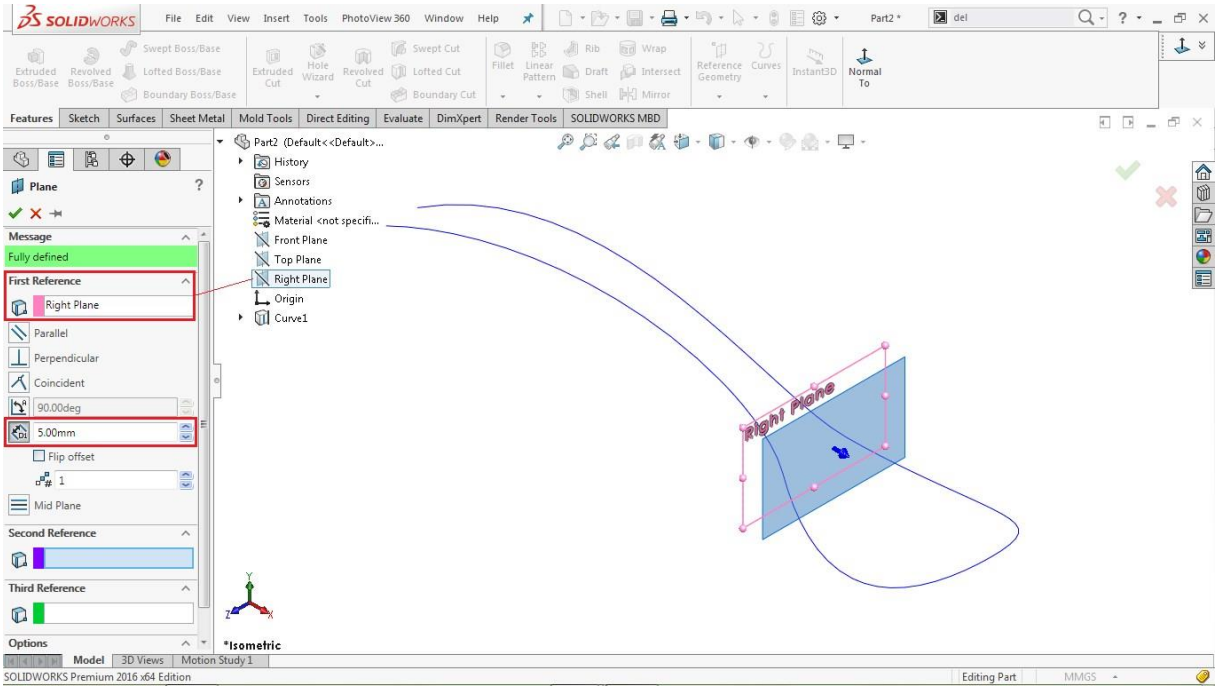
Click **OK** to complete the operation.

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



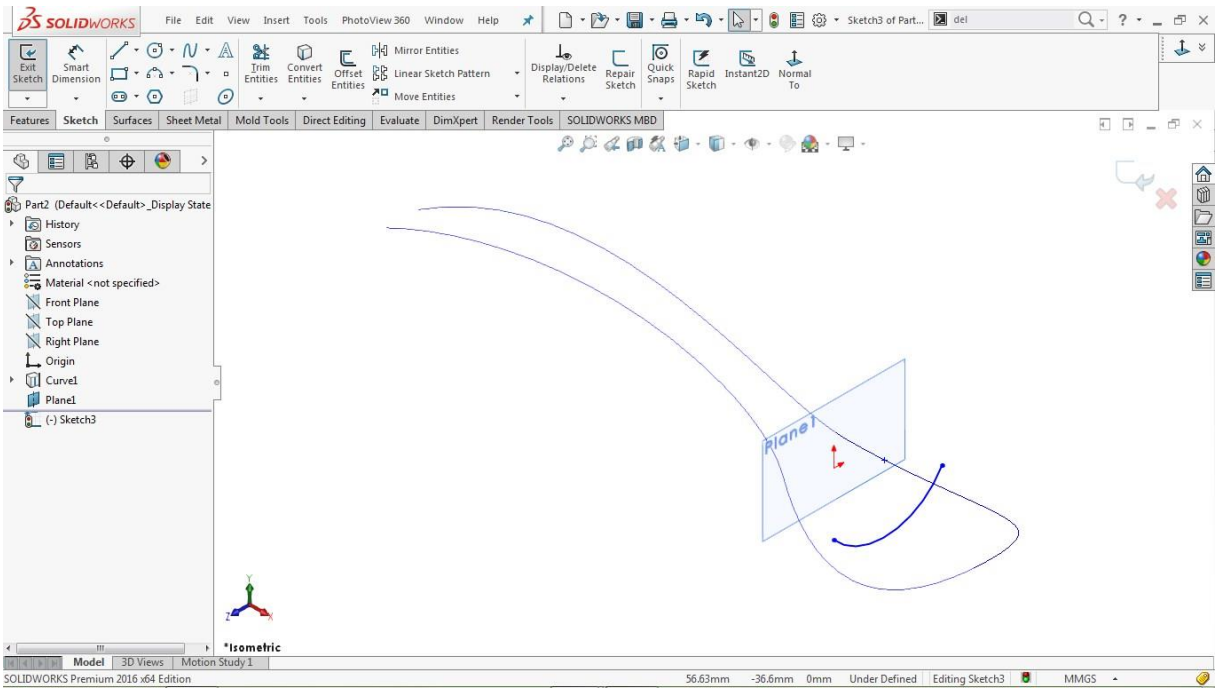
Click **OK** to complete the operation.

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

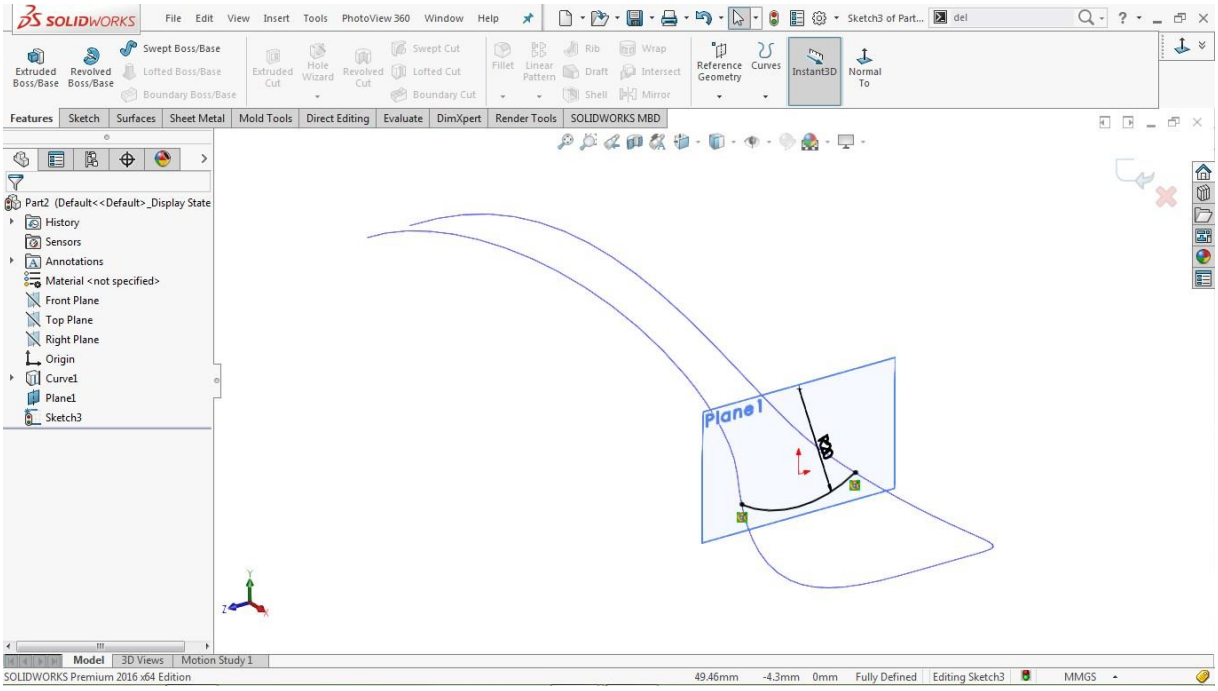
Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

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.

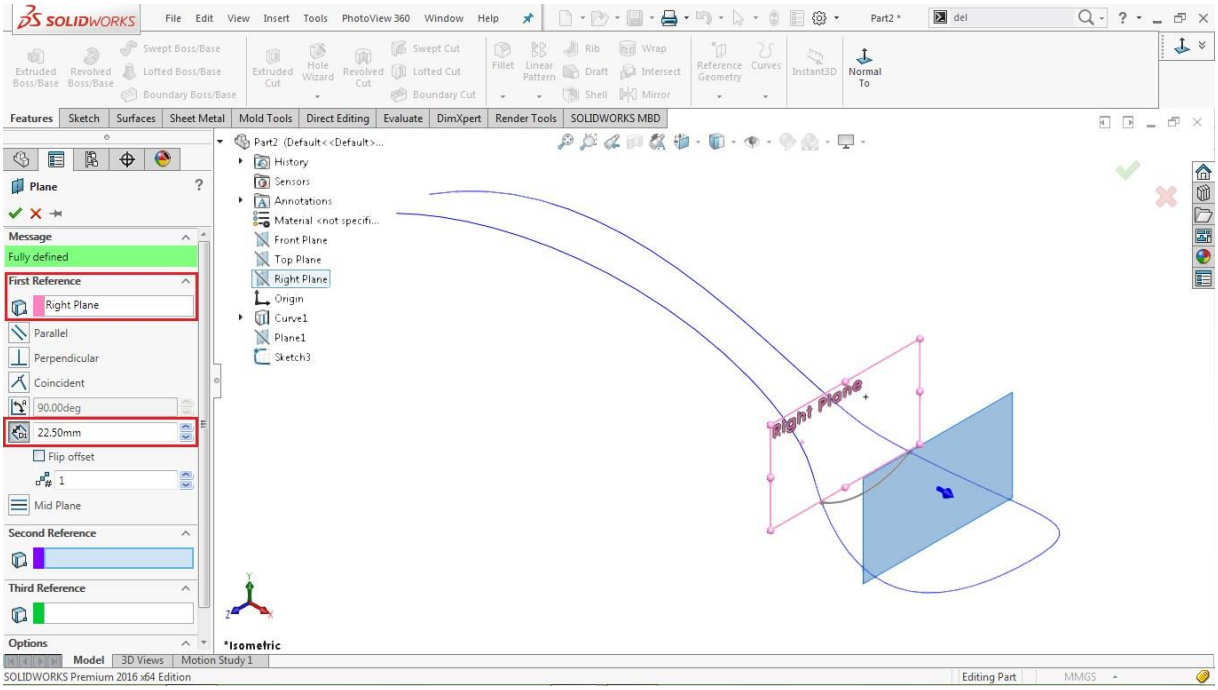
Go to the feature tree, click on the **Plane1** and hit the **Eye** icon to hide that plane

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



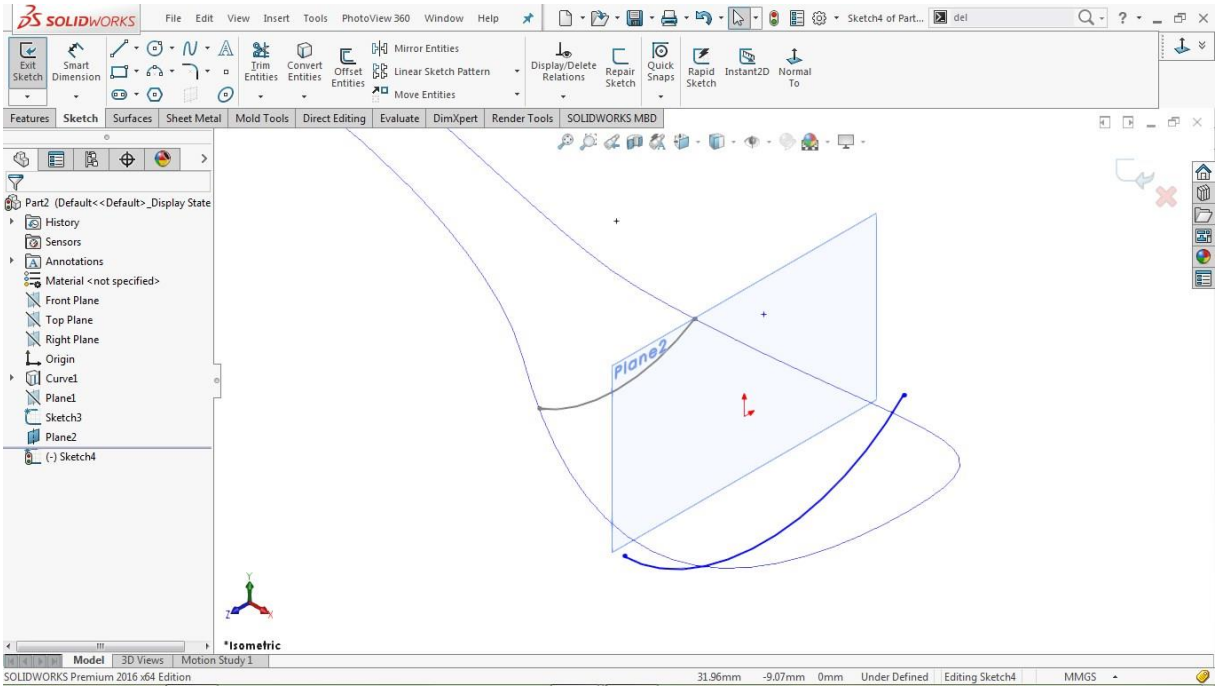
Click **OK**  to complete the operation.

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**

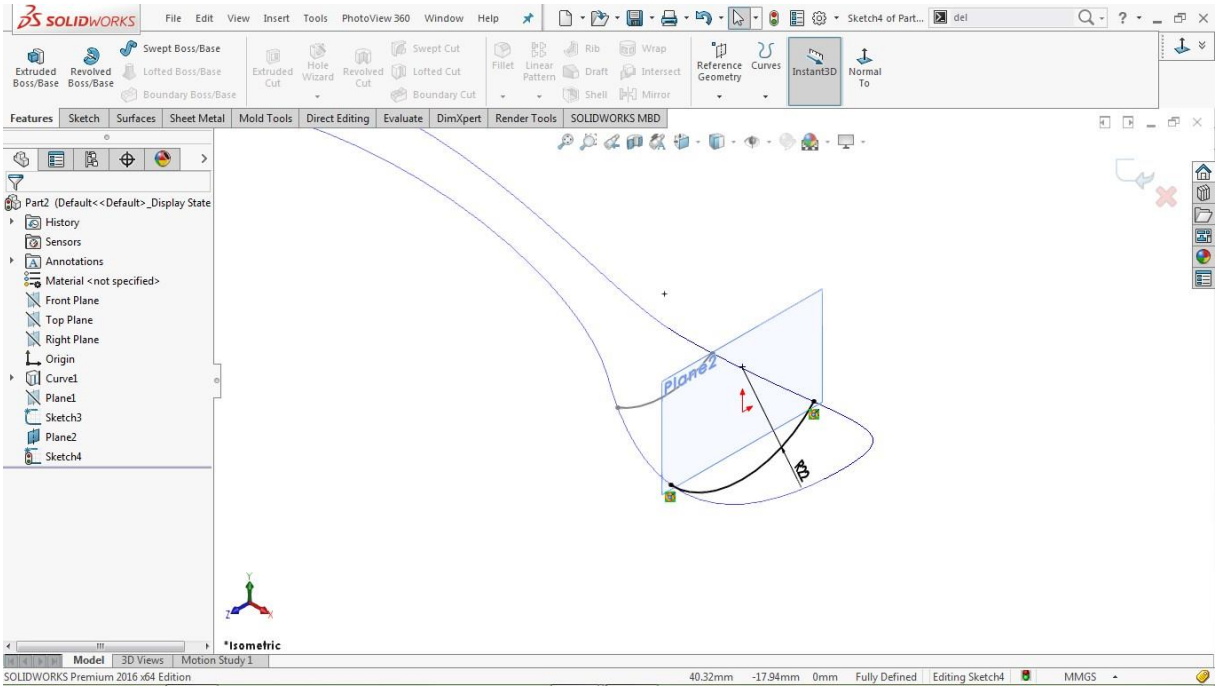
Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

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

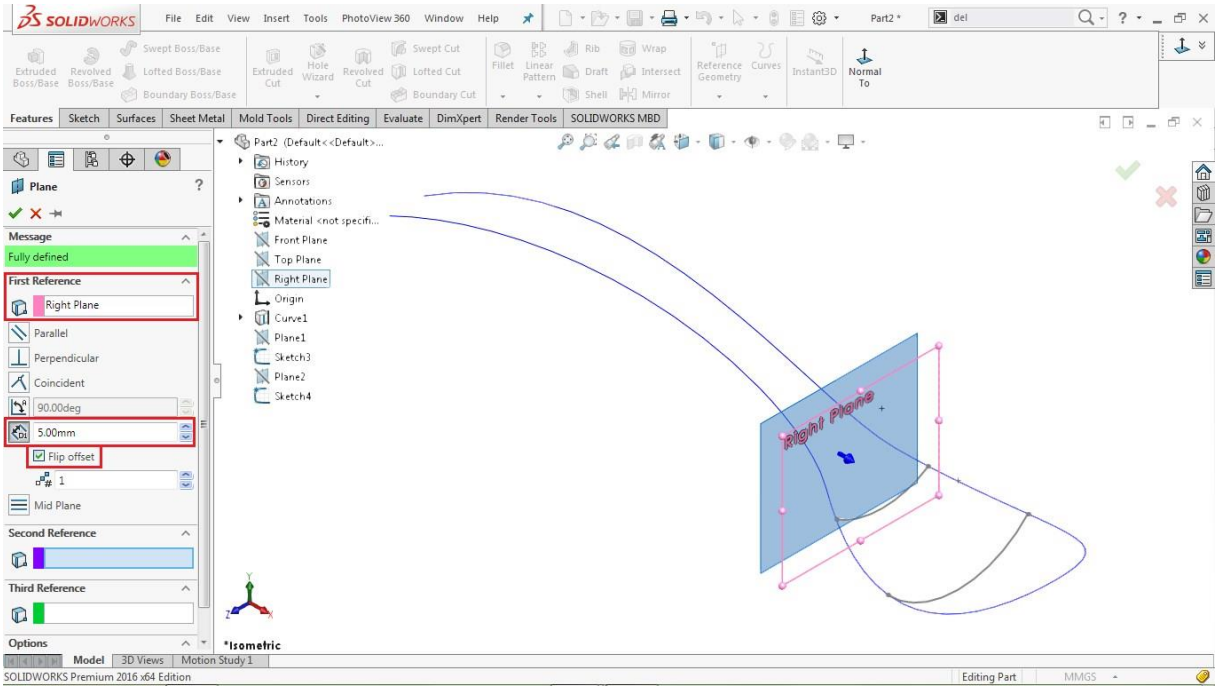
Make a Reference Plane (Plane3)

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



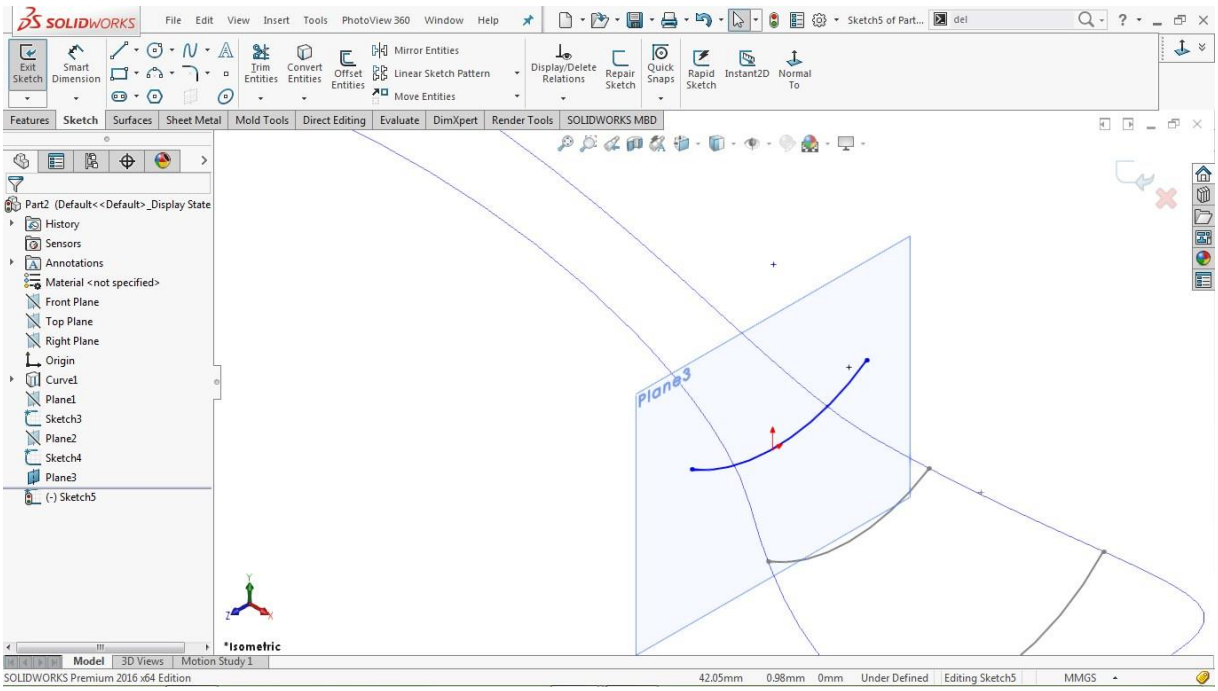
Click **OK** to complete the operation.

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**

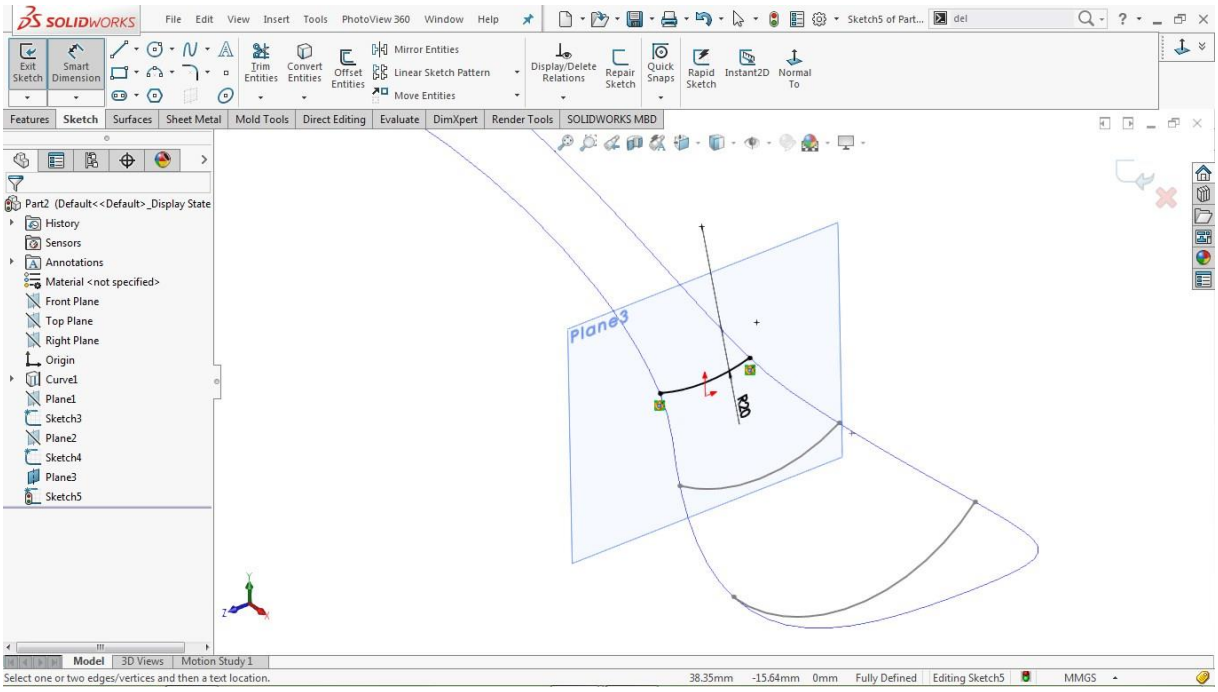
Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

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

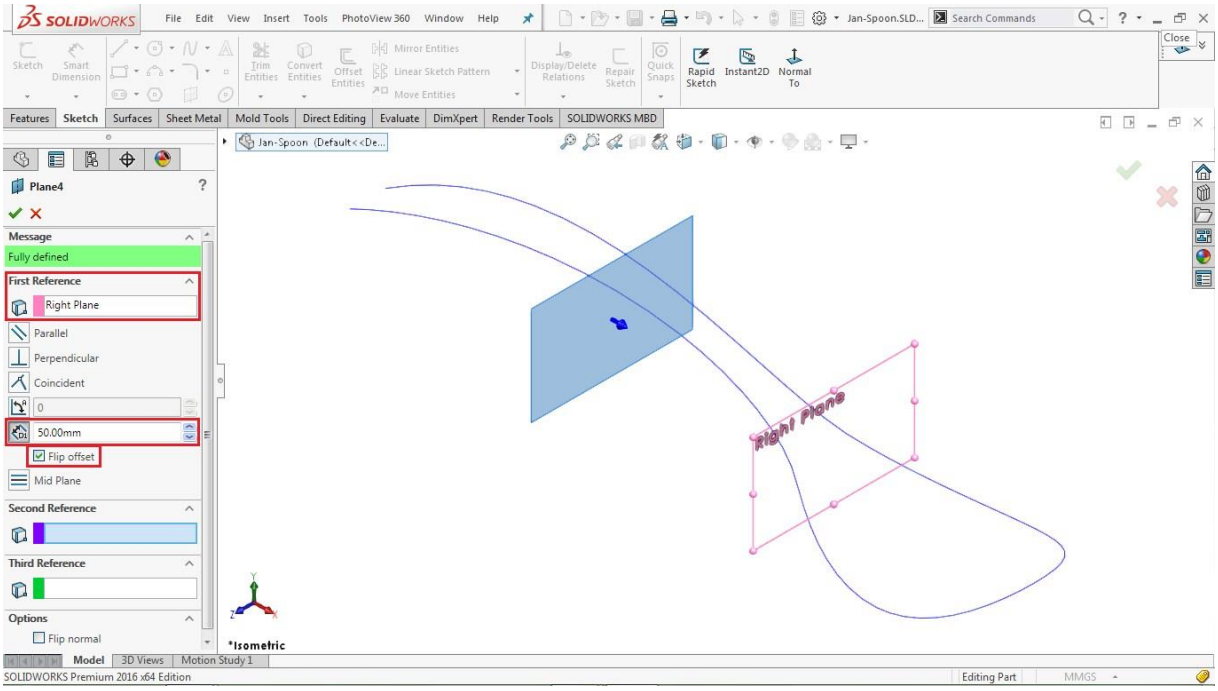
Make a Reference Plane (Plane4)

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



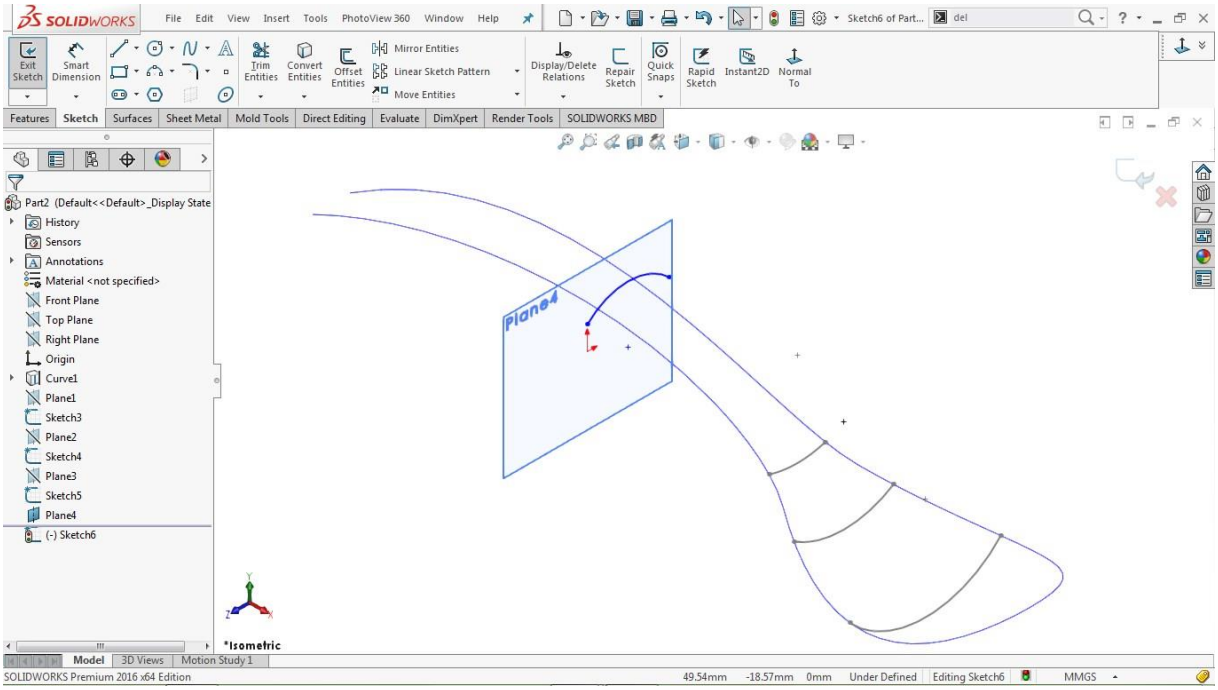
Click **OK**  to complete the operation.

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**

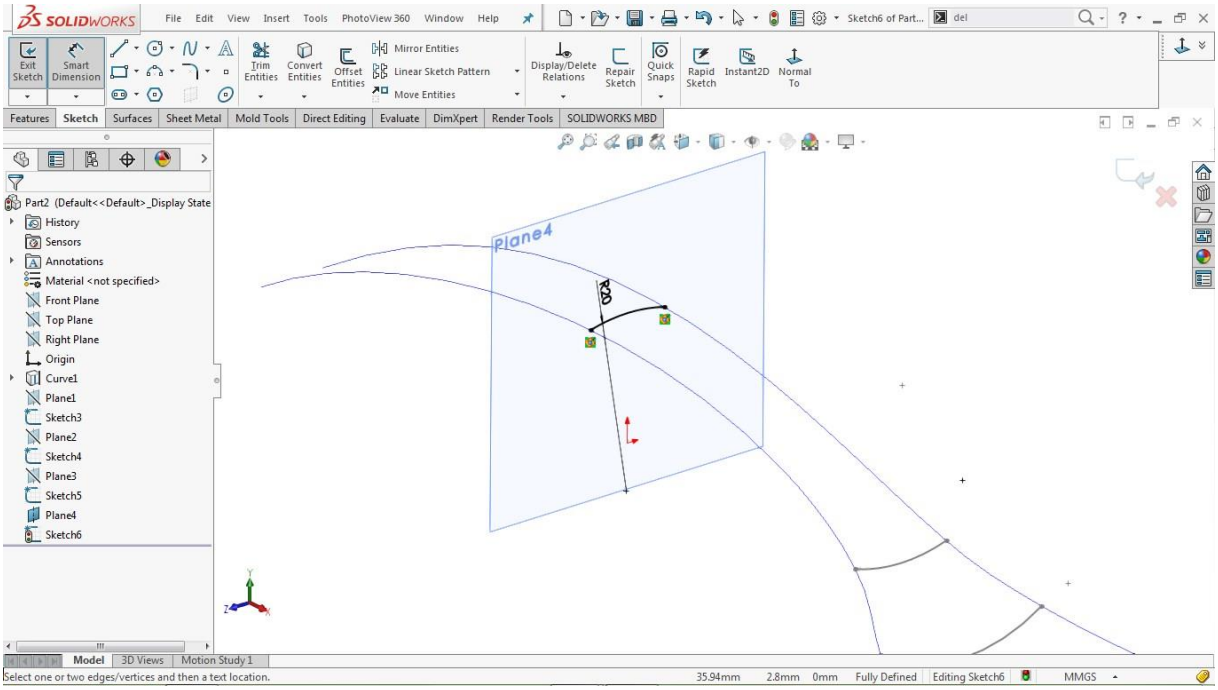
Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

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.

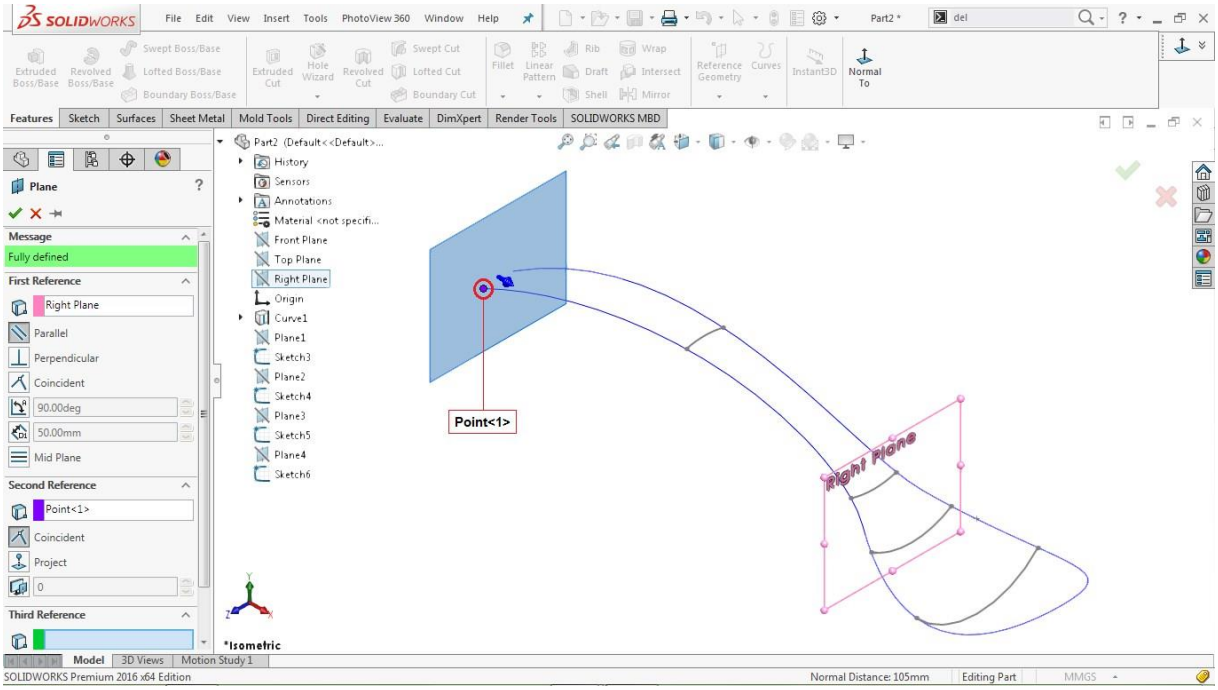
Go to the feature tree, click on **Plane4** and hit the **Eye**  icon to hide that plane

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



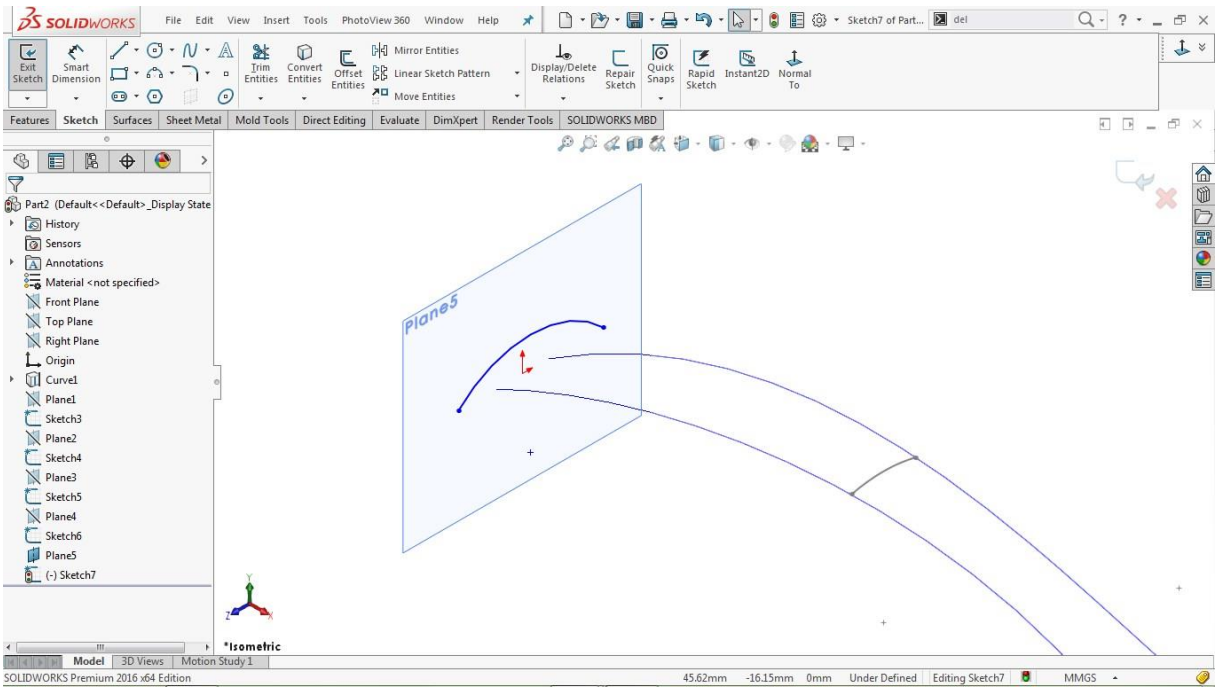
Click **OK** to complete the operation.

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**

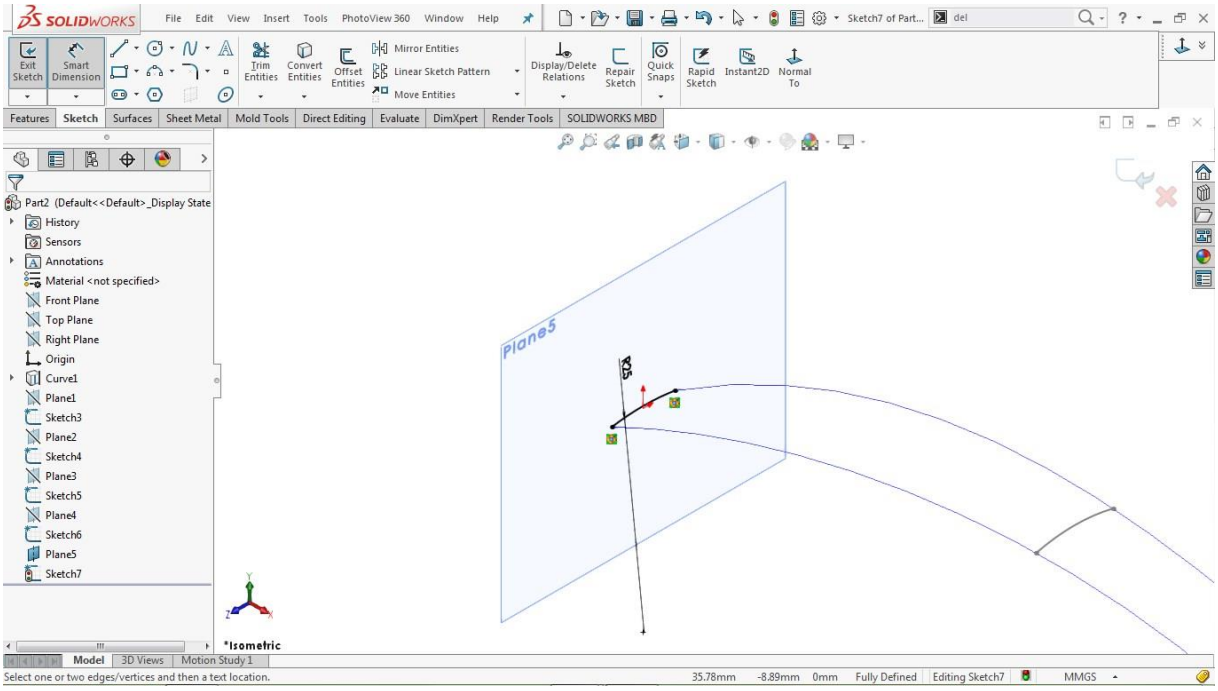
Similarly, select the right endpoint of the arc and the corresponding edge of the projected curve

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**

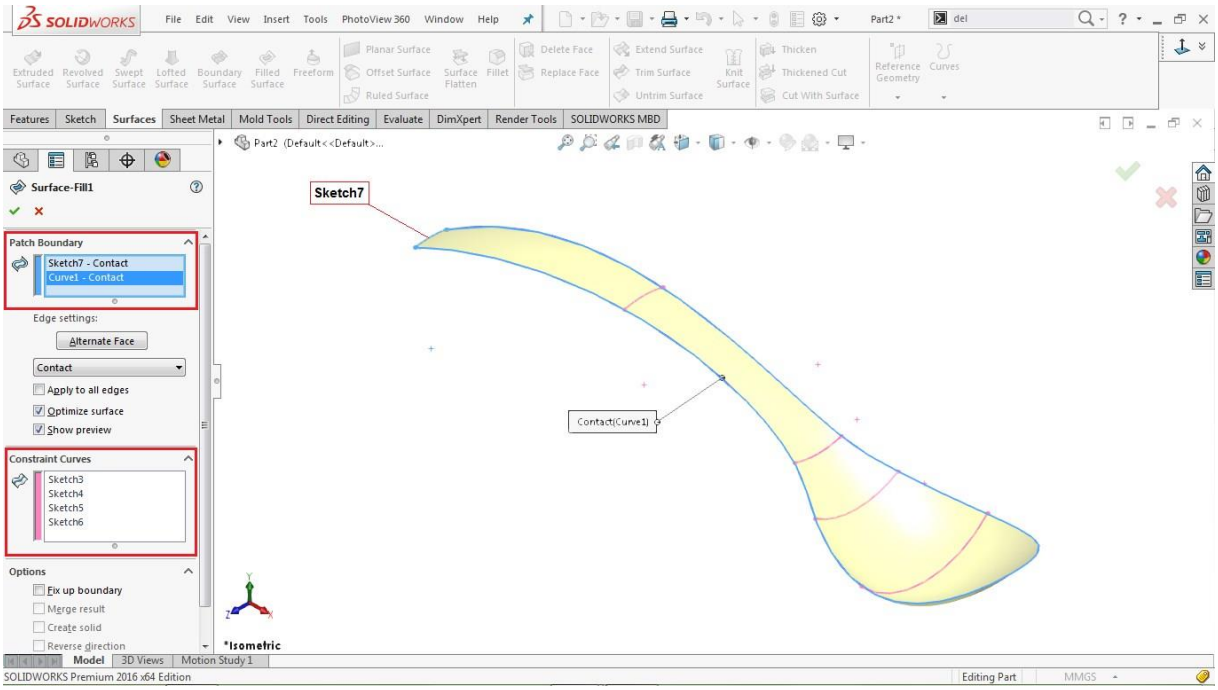
Click on the **Patch Boundary** input box

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.

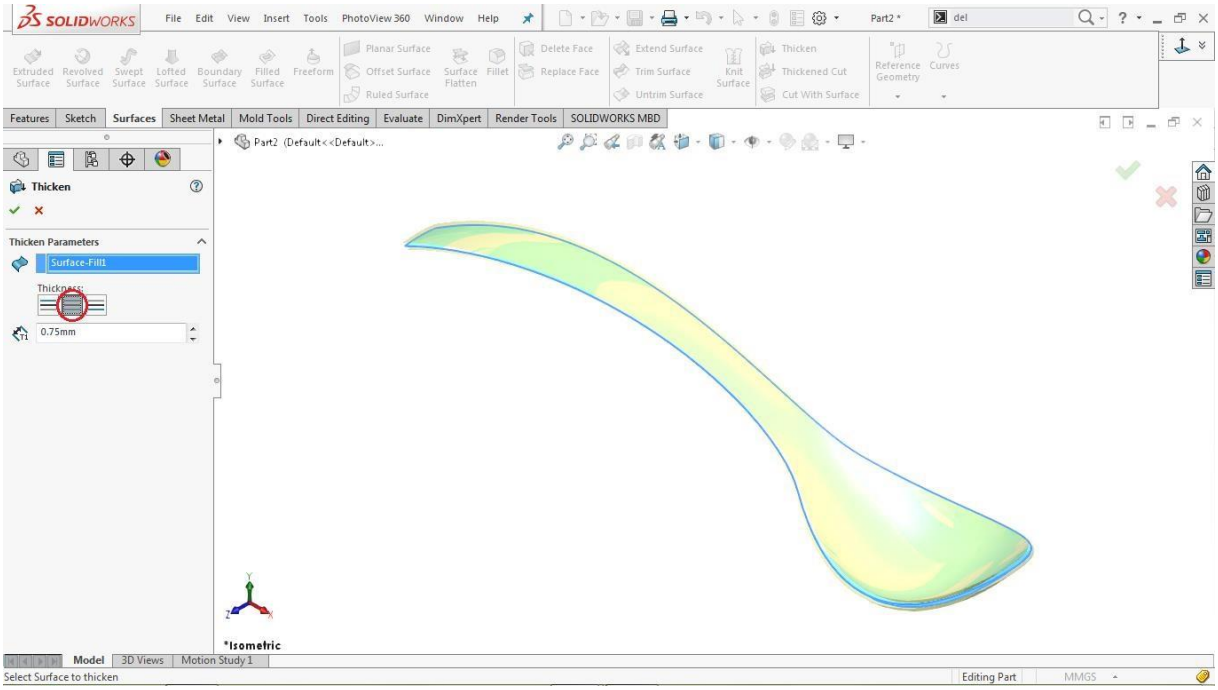
Create Thicken

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

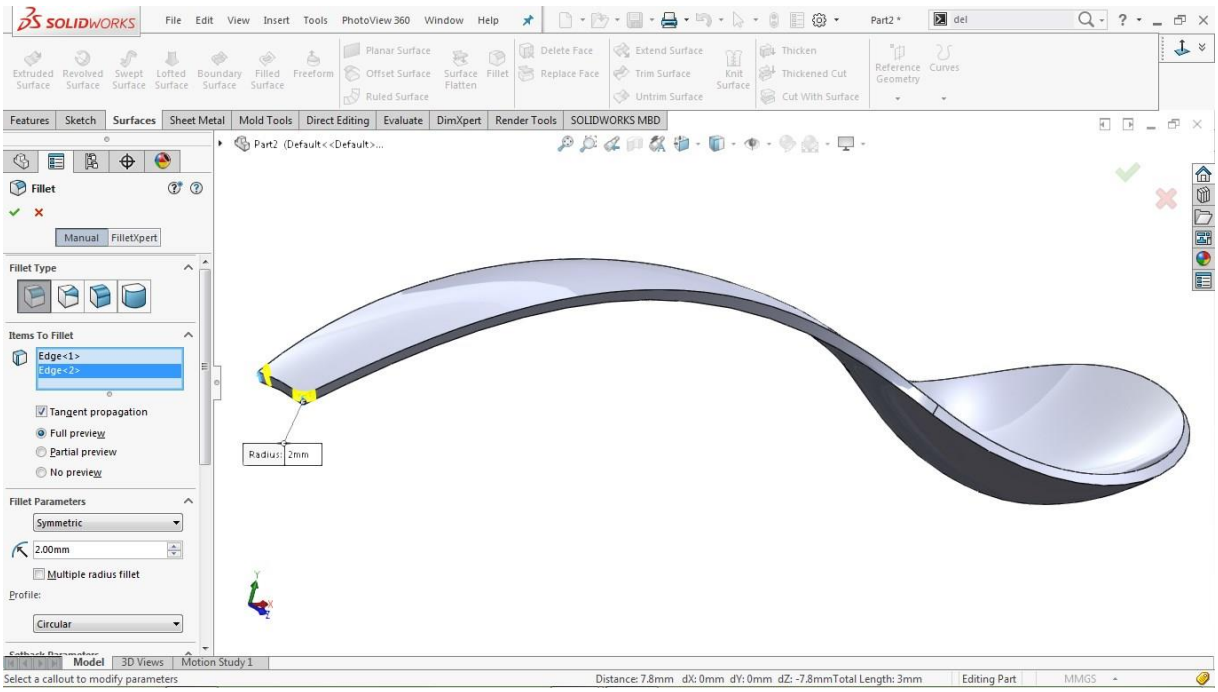


Click **OK**  to complete the thicken operation.

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



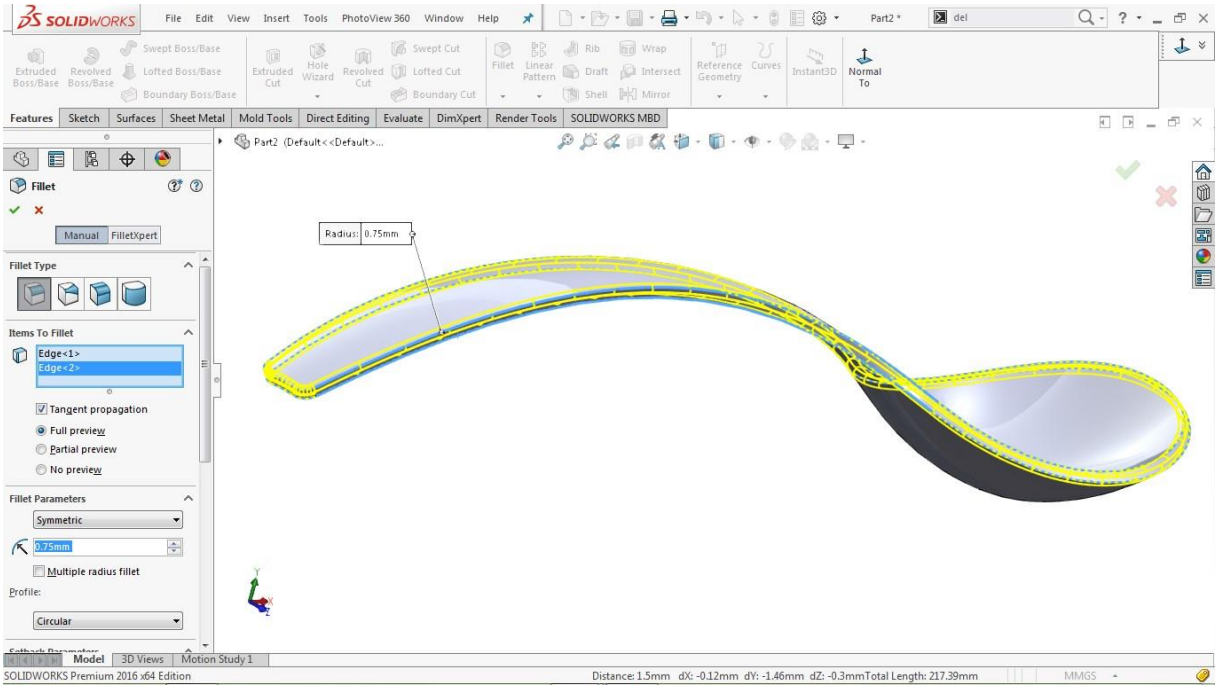
Go to the property manager and assign the Fillet radius of **2mm**

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…

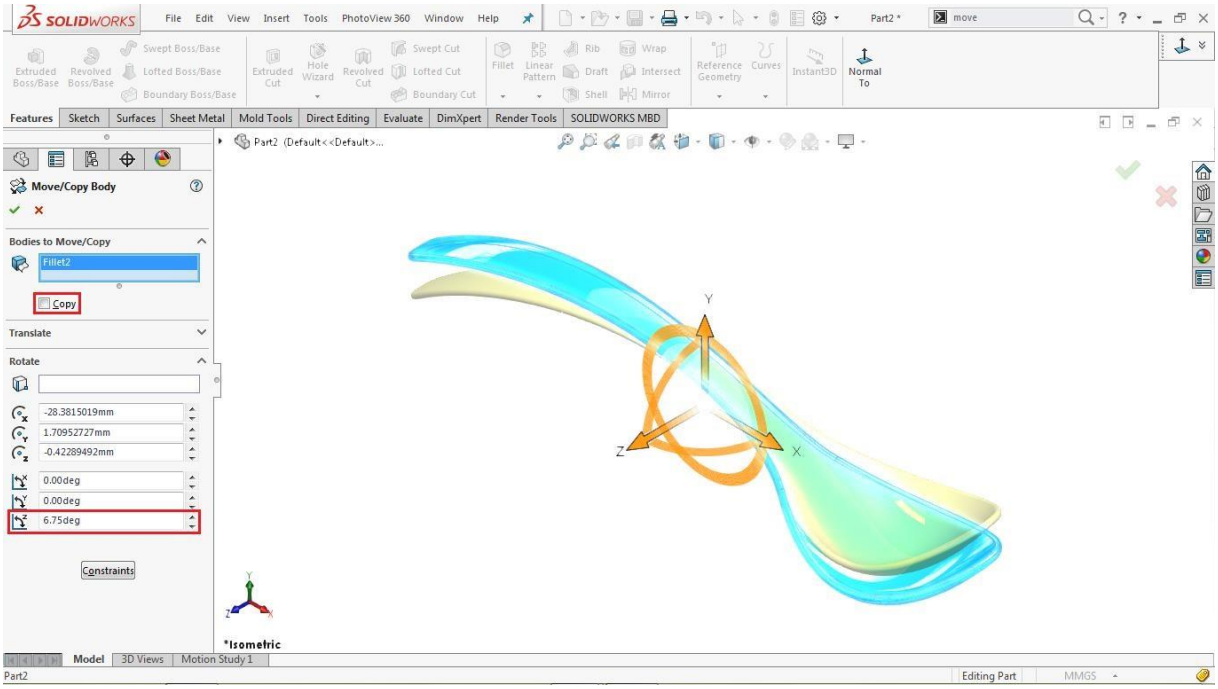
Select the existing solid body from the modeling area

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



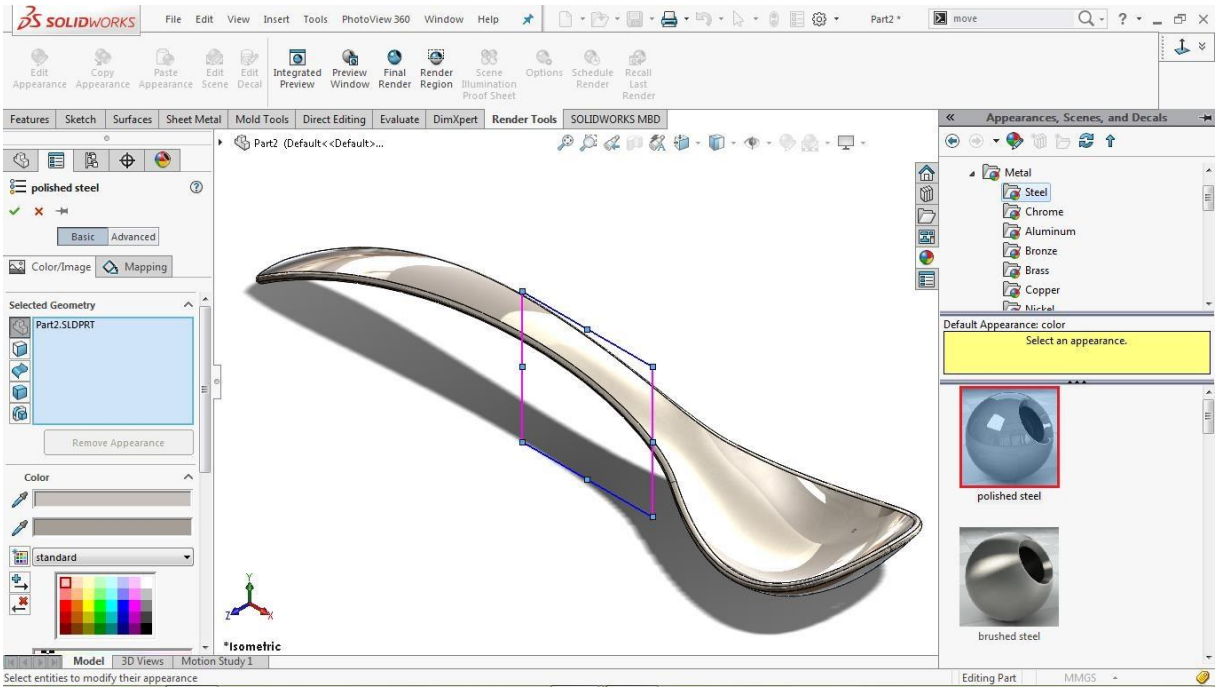
Leave the other parameters as default

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!

About the Author

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 9 SolidWorks eBooks (125.000+ downloads) and 4 premium SolidWorks training packages.

One of my SolidWorks courses is the SolidWorks Boeing 747-8 course. In this video training course, I will show you how to model this incredible Boeing 747-8 in SolidWorks.

This practical video training course is all you need if you want to increase your SolidWorks modeling skills in a fun and efficient way.

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

Happy modeling!

Jan