

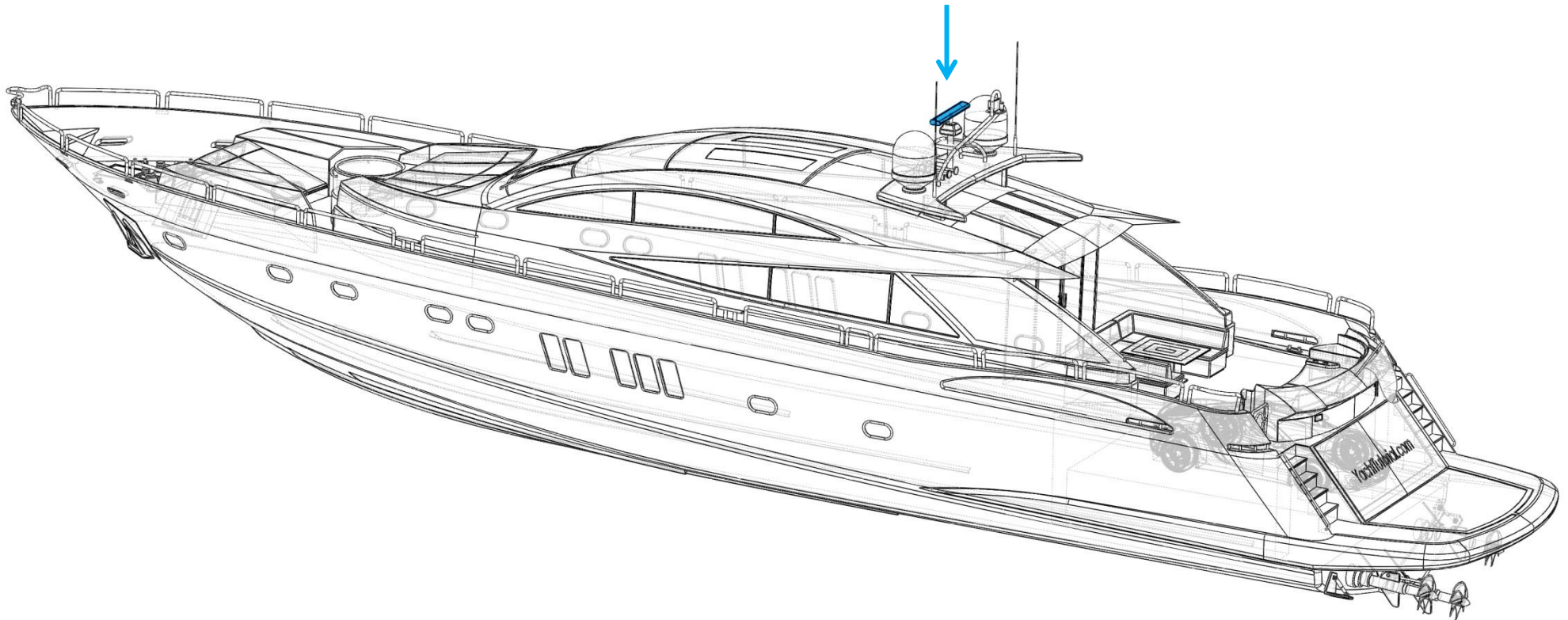


DIFFICULTY LEVEL



J.W. Zuyderduyn

HOW TO MODEL THE
2. RADAR
IN SOLIDWORKS®



HOW TO MODEL THE **2. RADAR** IN SOLIDWORKS®

Open SolidWorks

Go to: [Start > Programs > SolidWorks](#) 

Create a new part

Open a new part with model units set to millimeters.

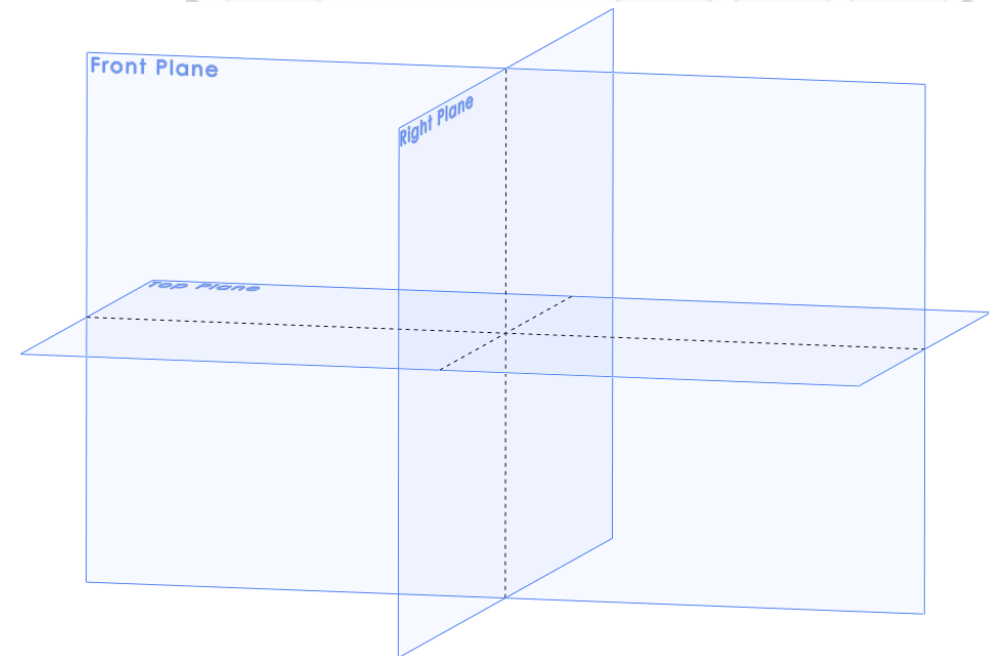
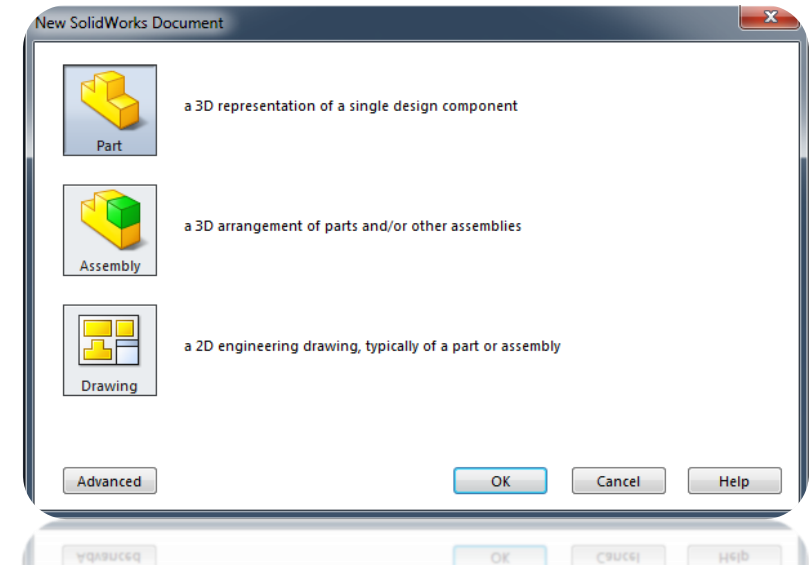
Go to: [File > New > Part](#)



Select the three planes in the feature tree and make them visible by clicking on the glasses. 

Go to: [File > Save](#) 

Give this part the following name: **Radar**



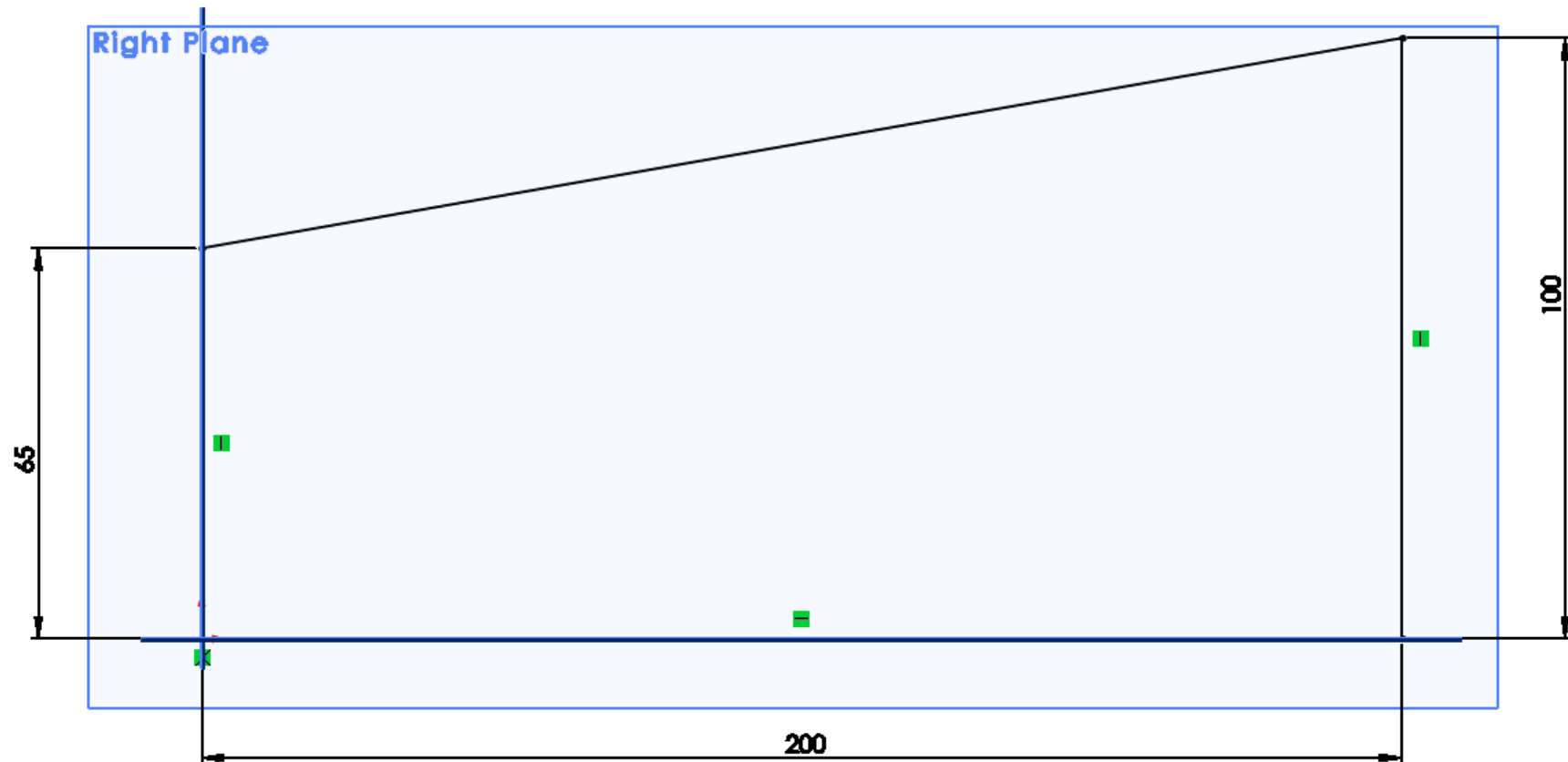
Create a new sketch on the Right Plane

Select the **Right Plane** in the Feature Manager

Click at the **2D Sketch** icon to make a new sketch 

Draw the four lines as shown in the picture 

Add the dimensions  as shown in the picture

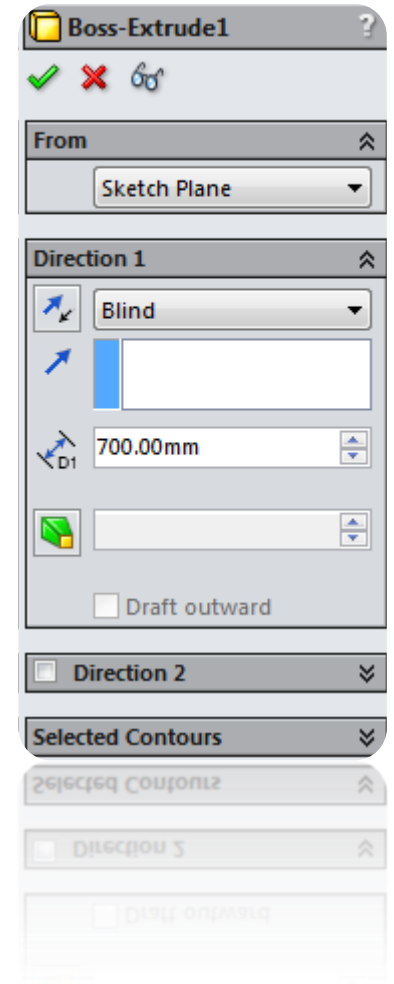
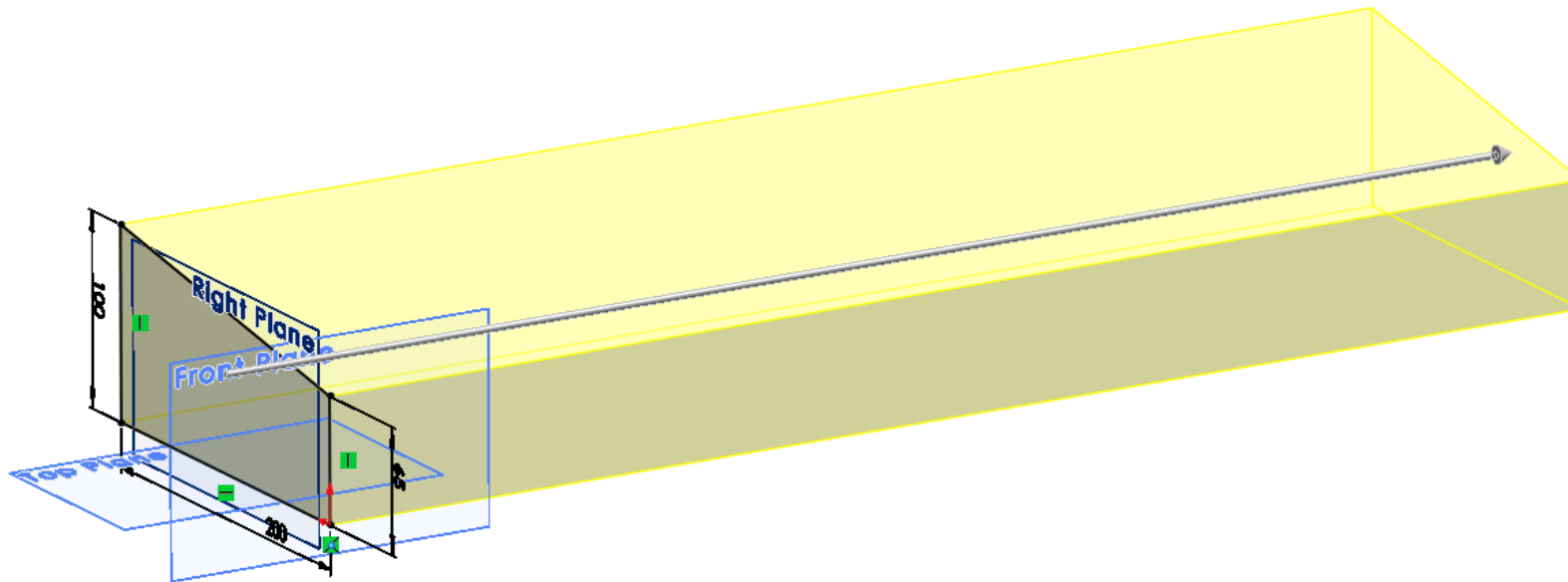


Create an Extrude

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

Change the Depth into 700 mm 

Click OK 



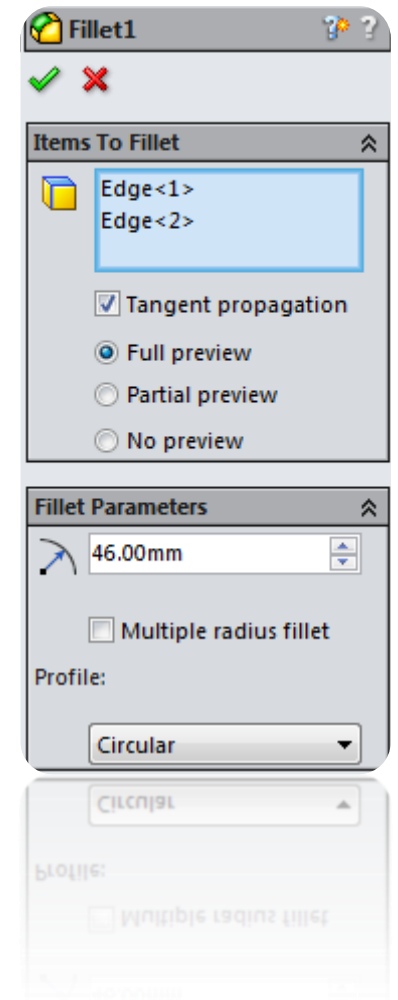
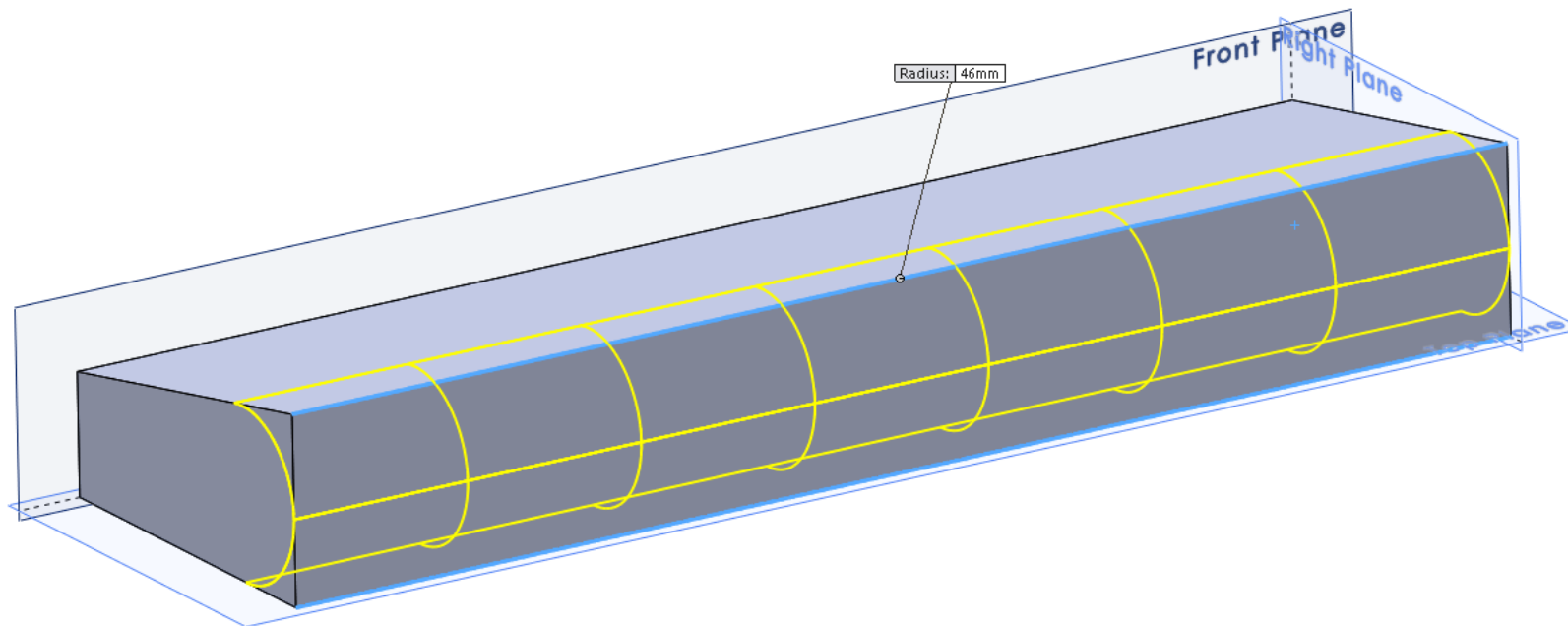
Create a Fillet

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

Click at the two blue edges as shown in the picture

Change the Radius into 46 mm 

Click OK 



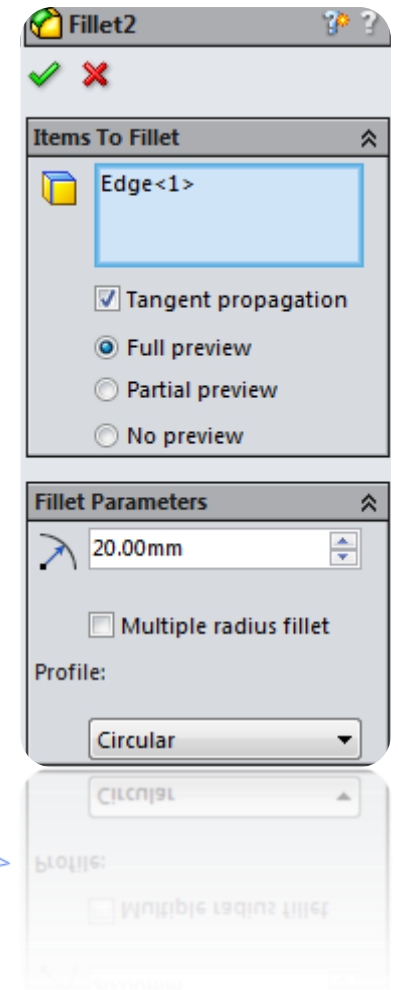
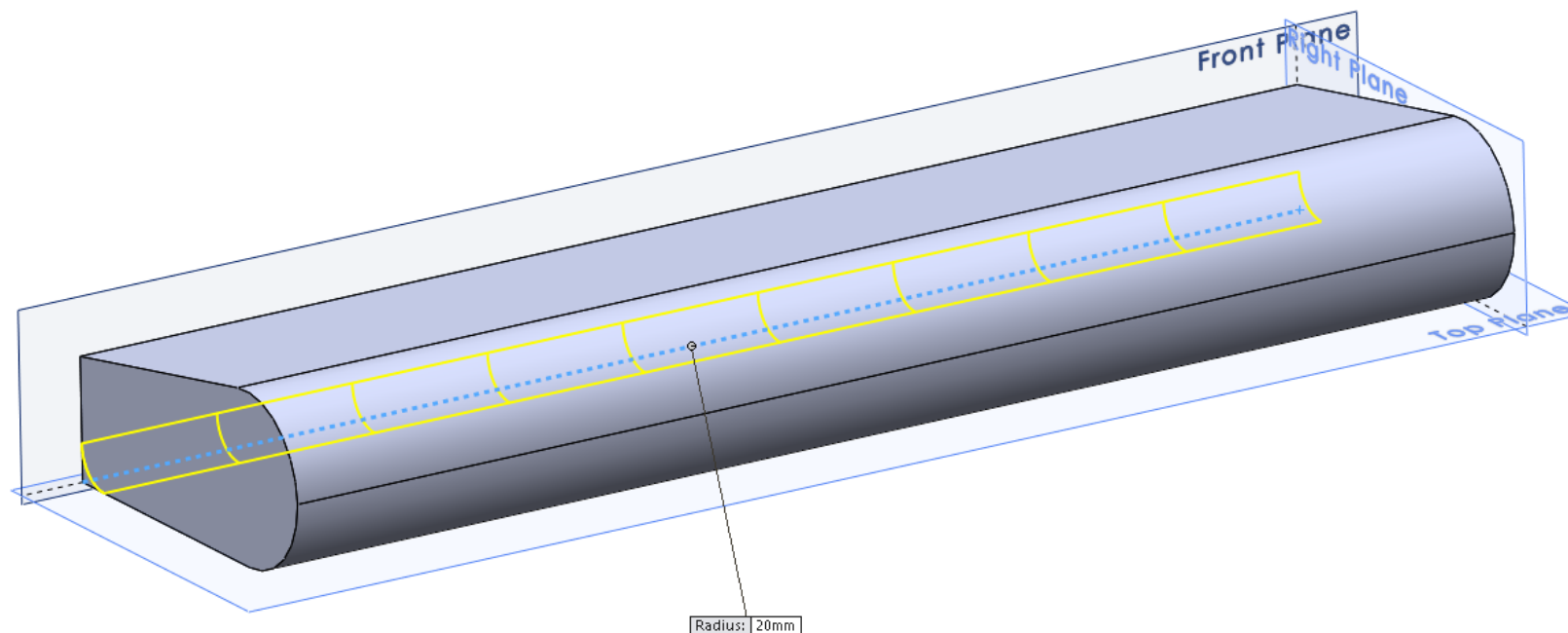
Create another Fillet

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

Click at the blue edge as shown in the picture

Change the Radius into 20 mm 

Click OK 



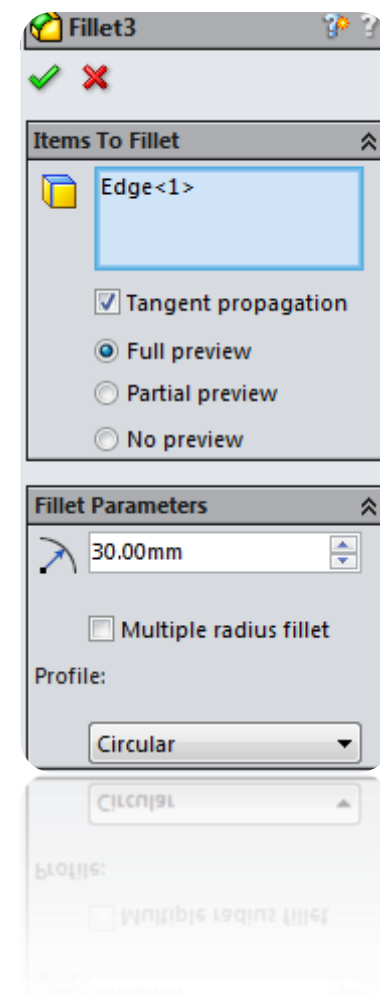
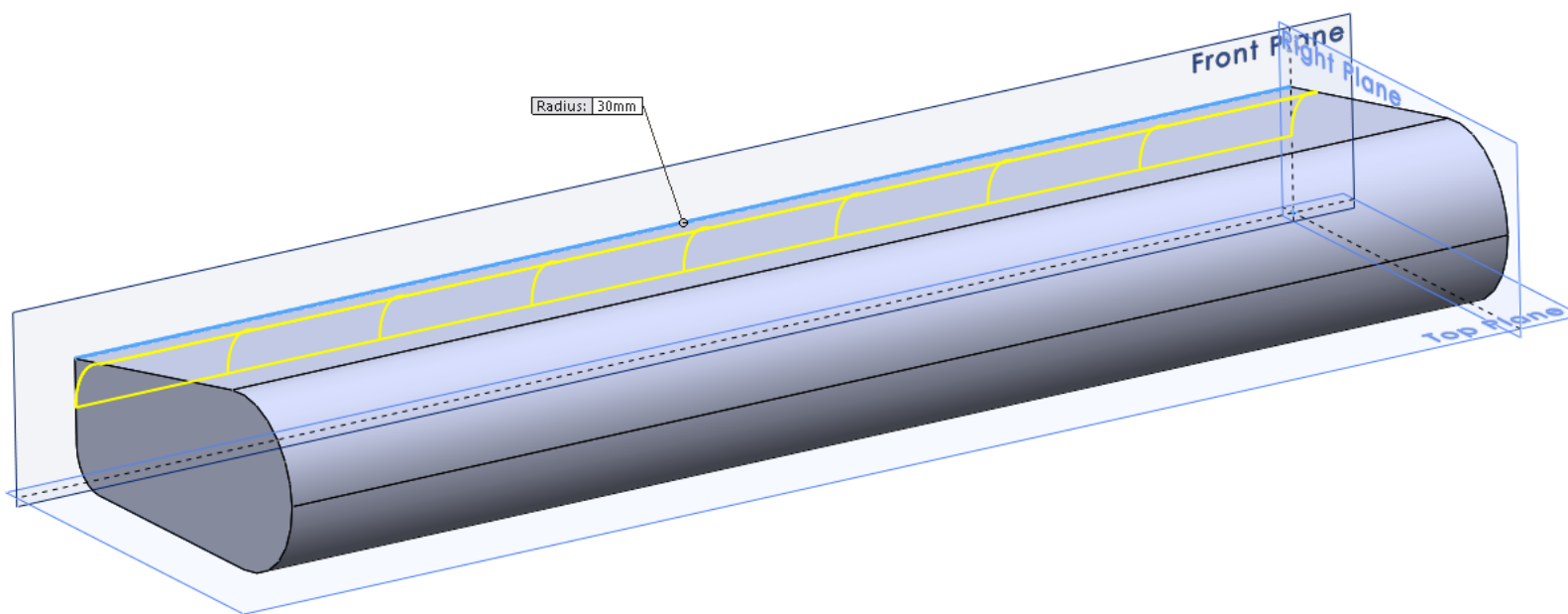
Create another Fillet

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

Click at the blue edge as shown in the picture

Change the Radius into 30 mm 

Click OK 



Select the blue surface and click at the **2D Sketch** icon to make a new sketch 

Click at the **Normal To** option to rotate the model normal to the selected surface 

Offset the edges of the blue surface

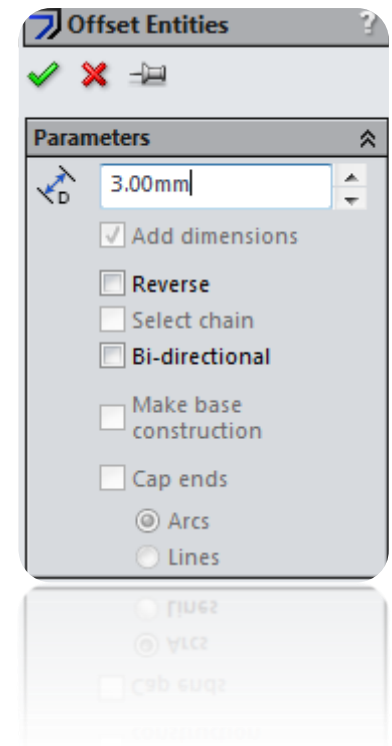
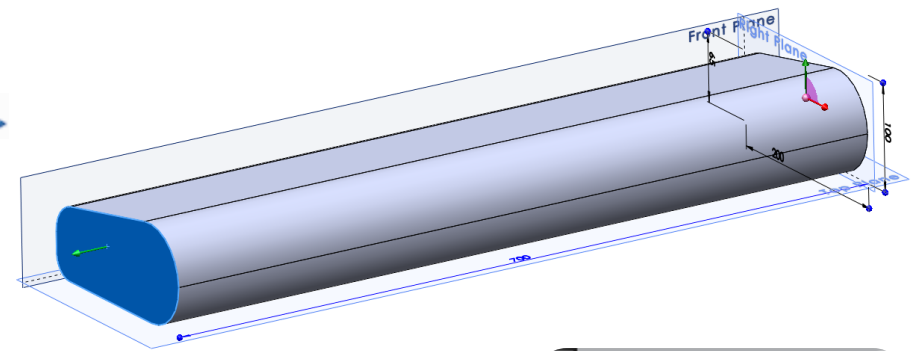
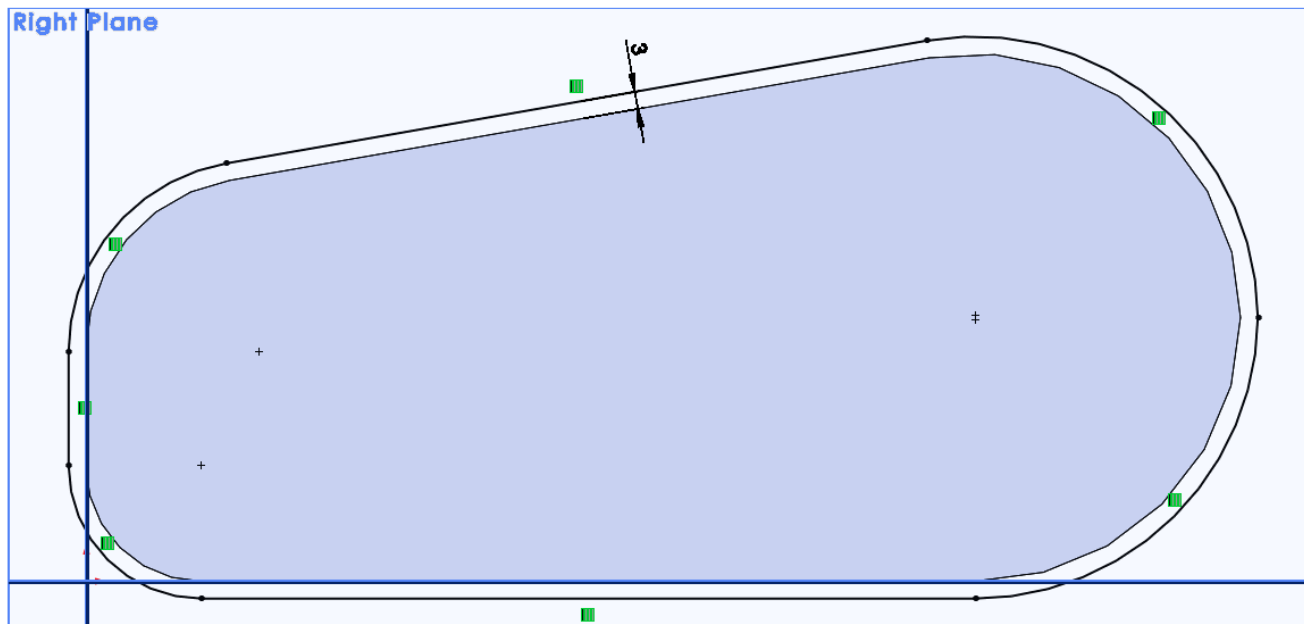
Select the blue surface again

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

Change the **Offset Distance** into 3 mm 

Make sure that the offset is directed **outwards**. If not, click at the **Reverse** button to change the direction.

Click OK 

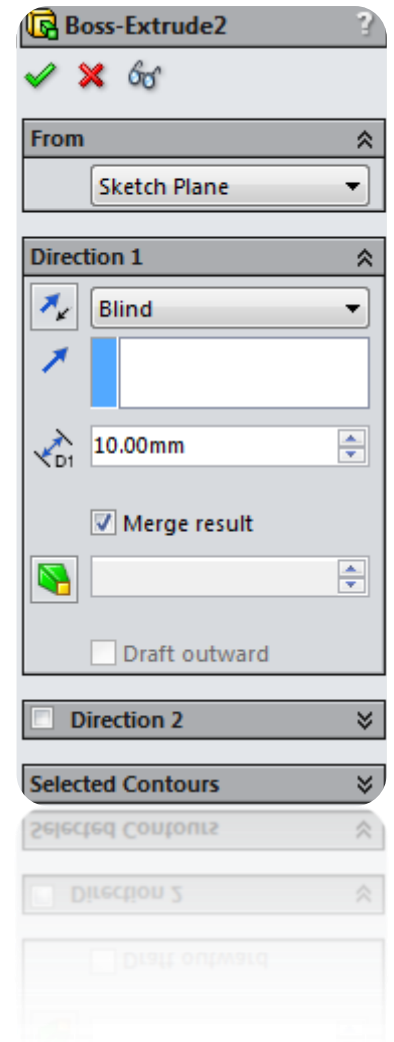
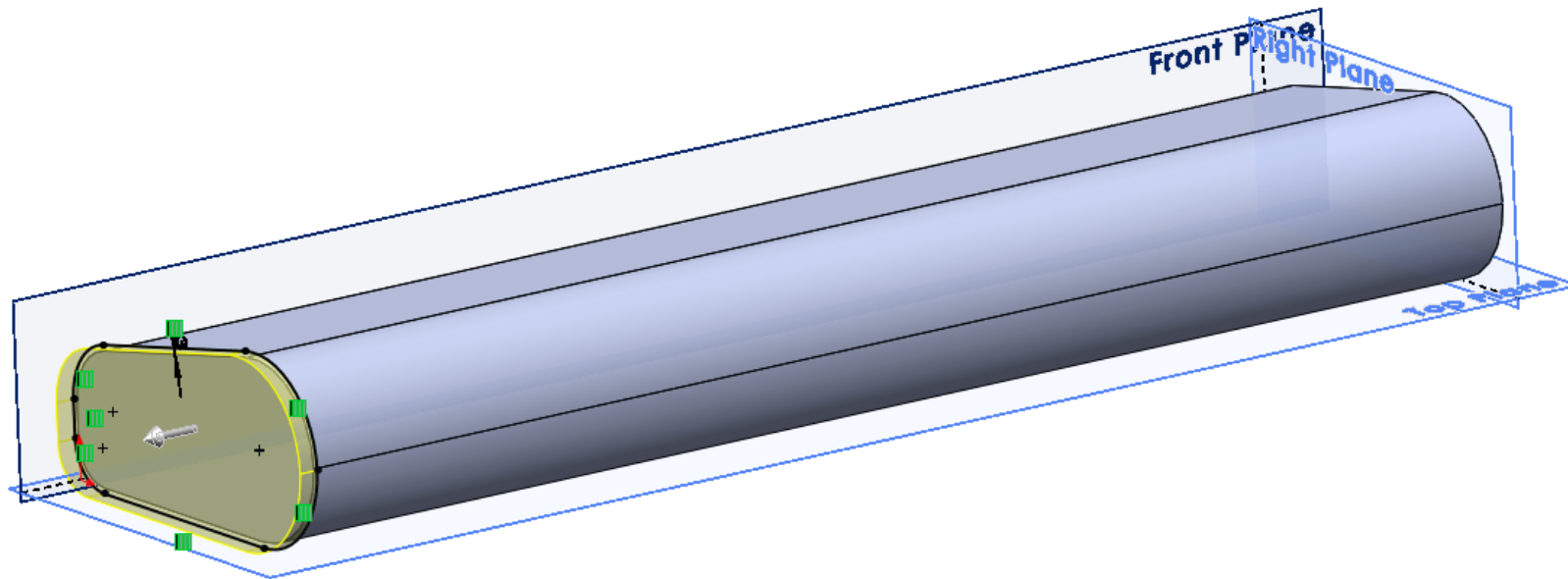


Create an Extrude

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

Change the Depth into 10 mm 

Click OK 



Select the blue surface and click at the **2D Sketch** icon to make a new sketch 

Click at the **Normal To** option to rotate the model normal to the selected surface 

Offset the edges of the blue surface

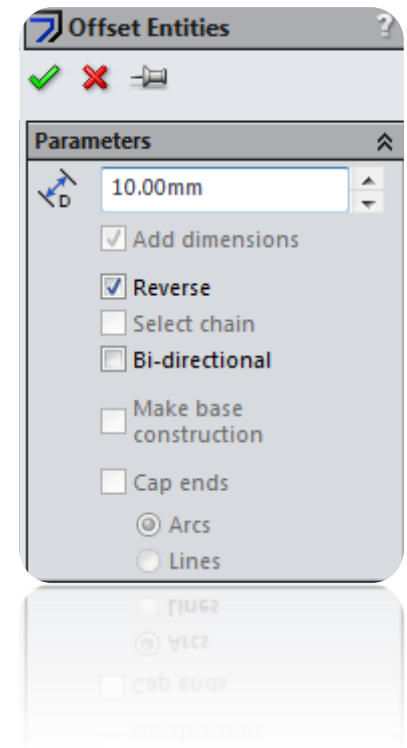
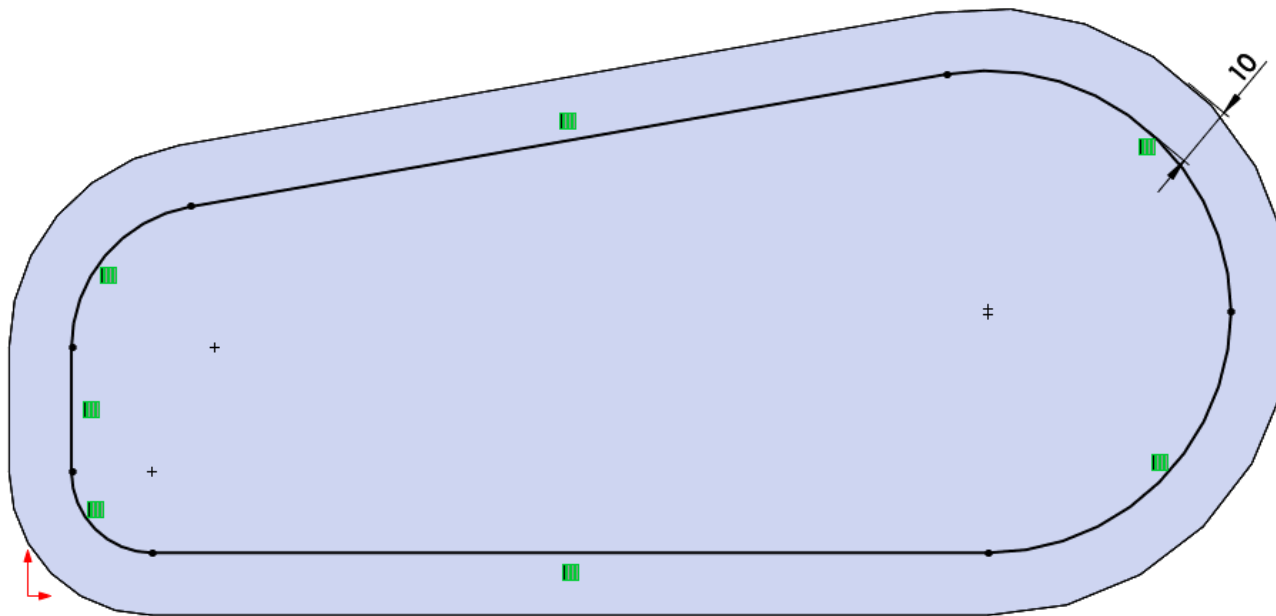
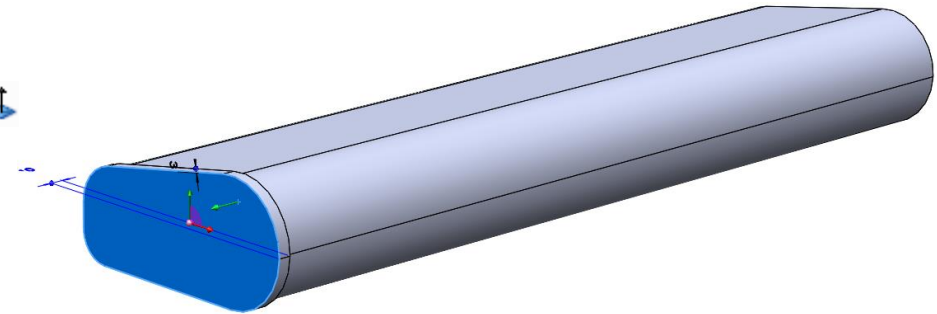
Select the blue surface again

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

Change the **Offset Distance** into 10 mm 

Make sure that the offset is directed **inwards**. If not, click at the **Reverse** button to change the direction.

Click OK 

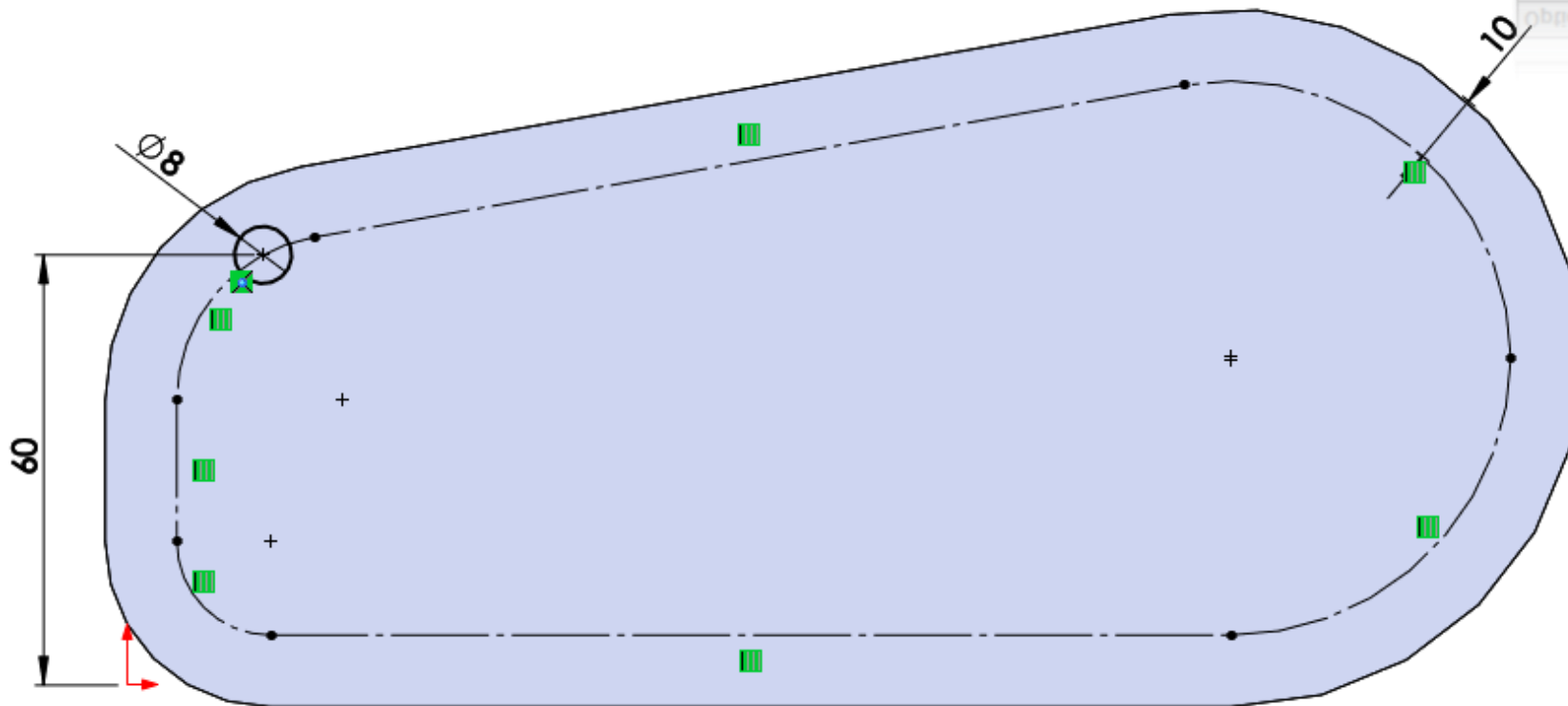
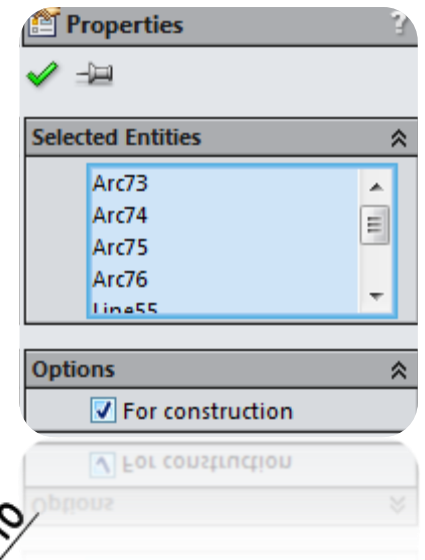


Click and drag your cursor over the sketch to select the offset

Click at the “**For construction**” option to convert the offset line into a construction line

Draw the circle as shown in the picture 

Add the dimension by clicking at the dimension button 

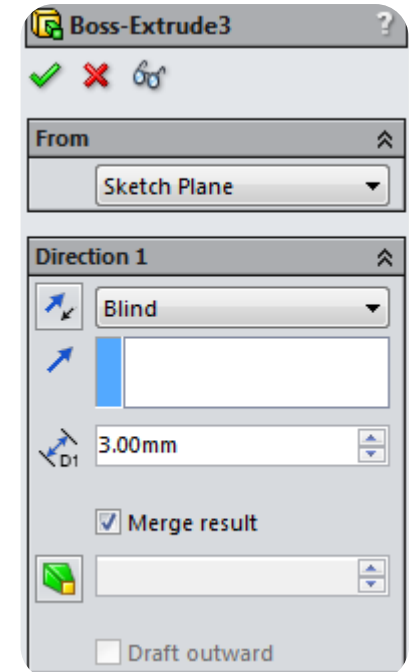
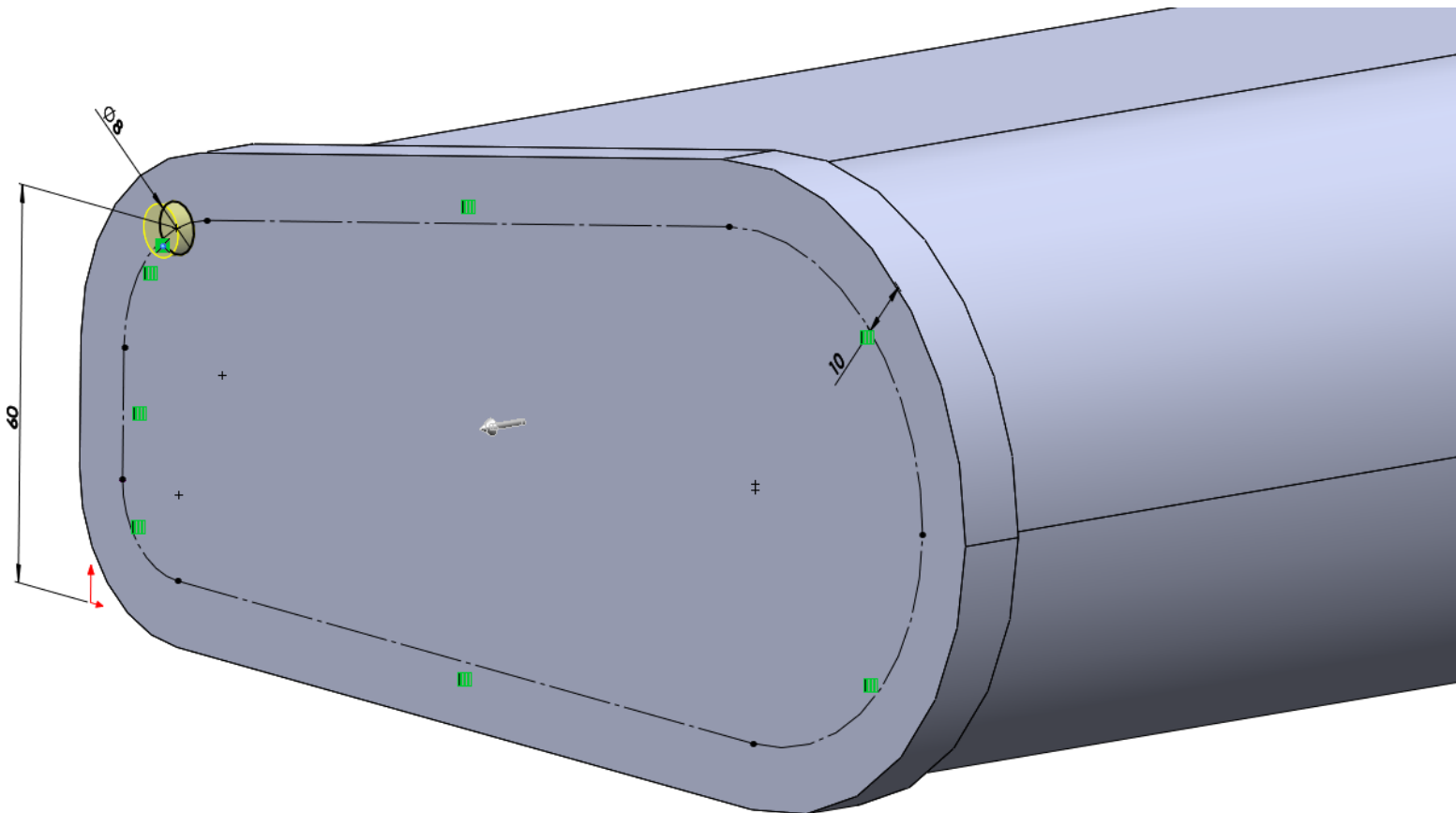


Create an Extrude

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

Change the Depth into 3 mm 

Click OK 



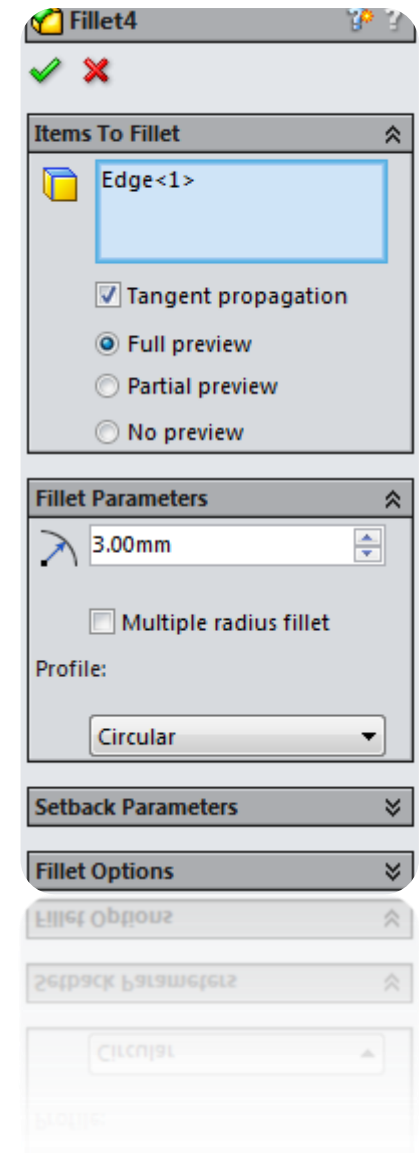
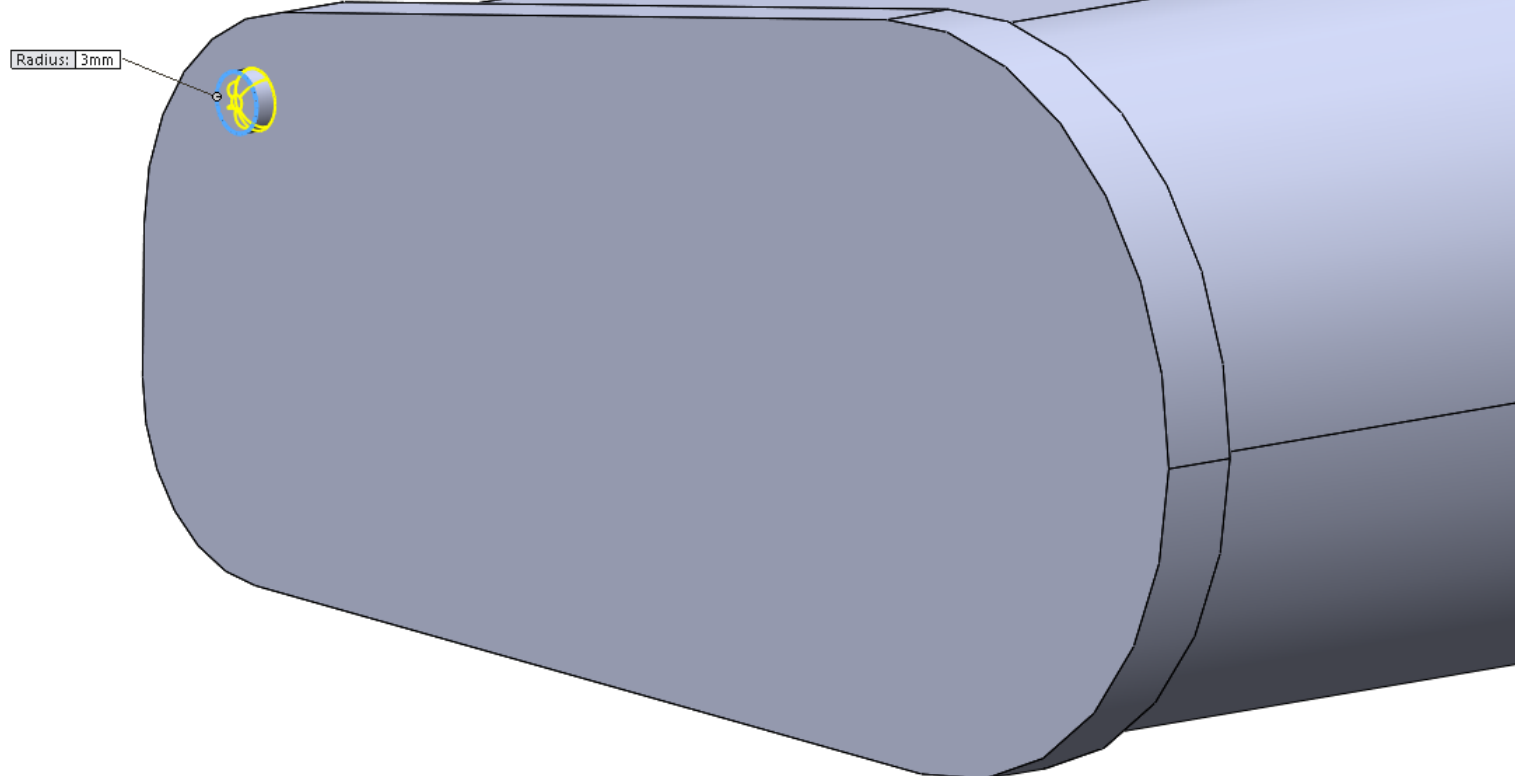
Create a Fillet

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

Click at the blue edge as shown in the picture

Change the Radius into 3 mm 

Click OK 



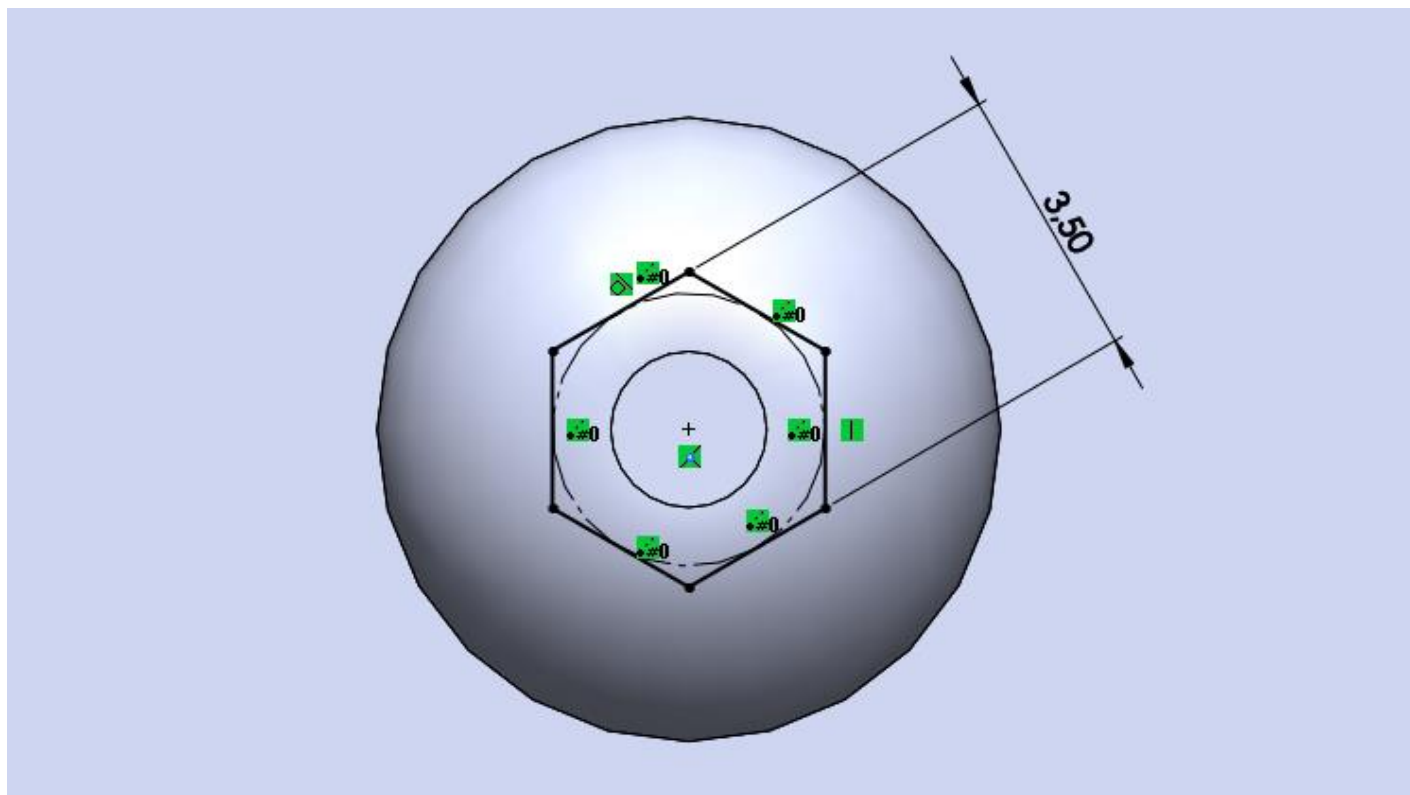
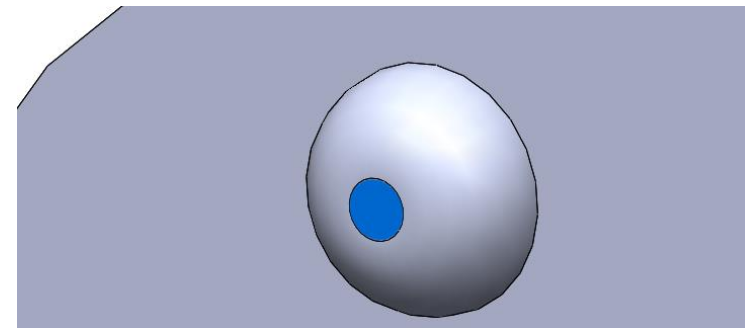
Select the blue surface and click at the **2D Sketch** icon to make a new sketch 

Click at the **Normal To** option to rotate the model normal to the selected surface 

Draw a Polygon

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

Change the **Width** into 3,5 mm 

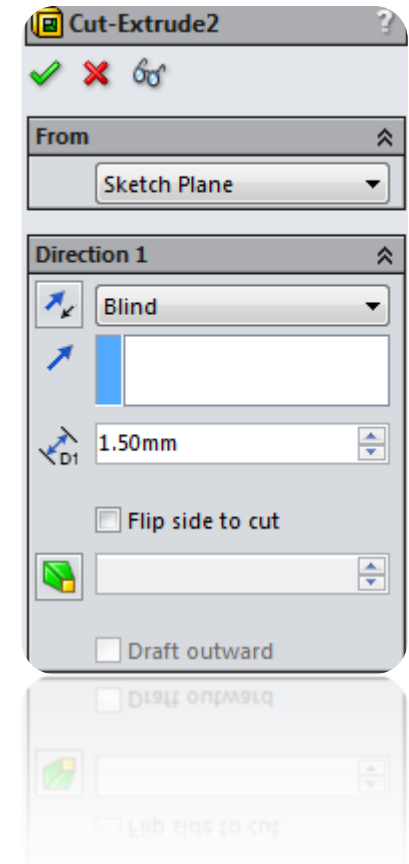
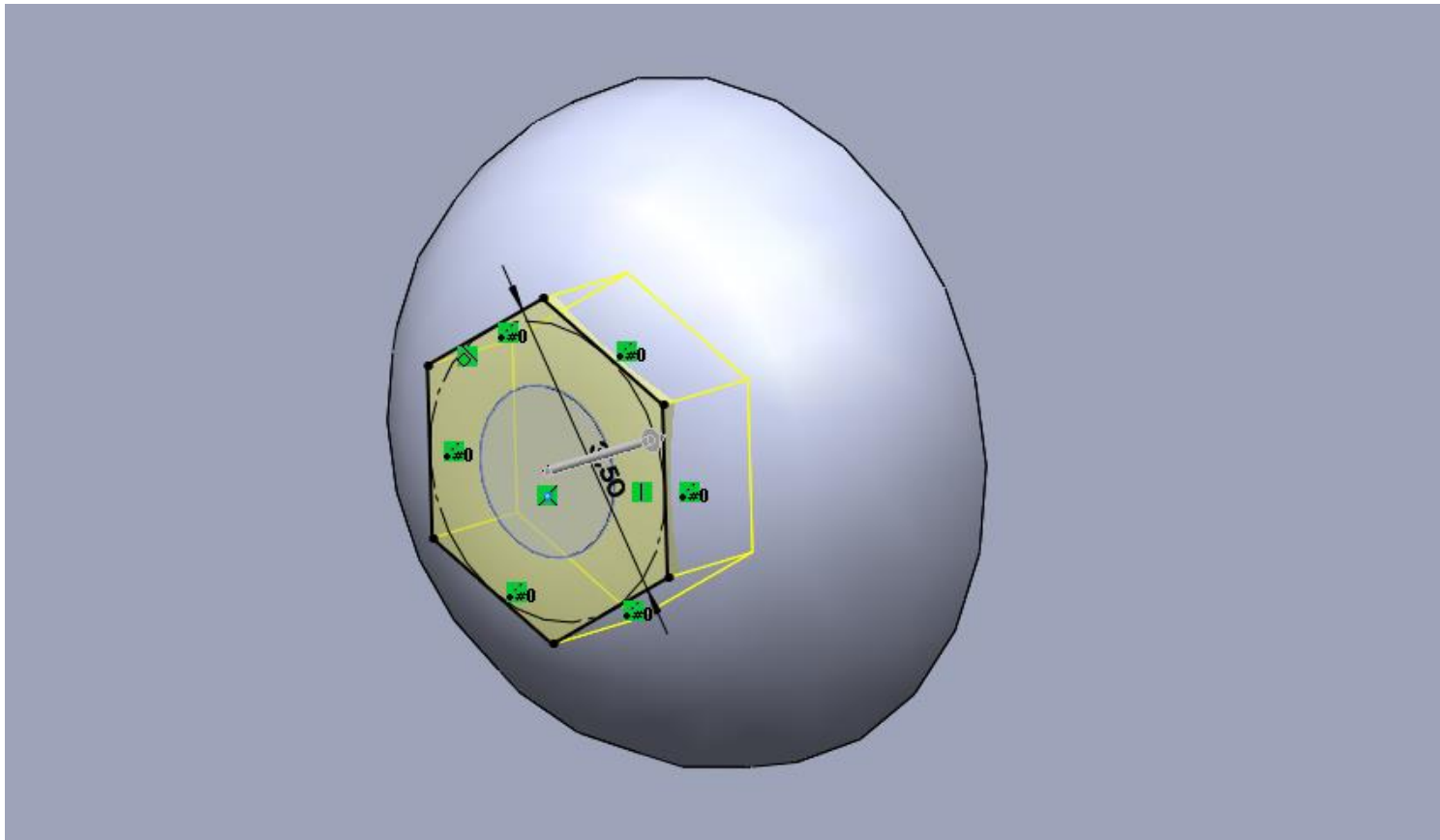


Create an Extruded Cut

Go to: **Insert > Cut > Extrude** or click at the Extruded Cut icon 

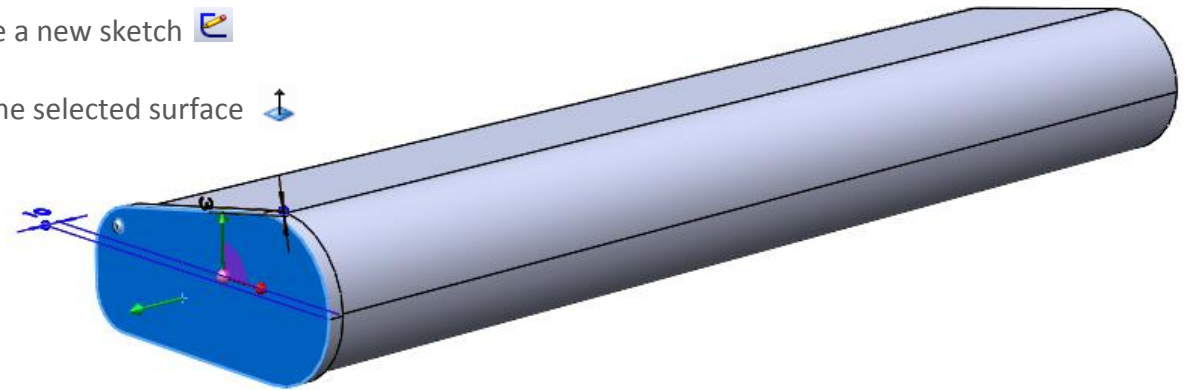
Change the Depth into 1.5 mm 

Click OK 



Select the blue surface and click at the **2D Sketch** icon to make a new sketch 

Click at the **Normal To** option to rotate the model normal to the selected surface 



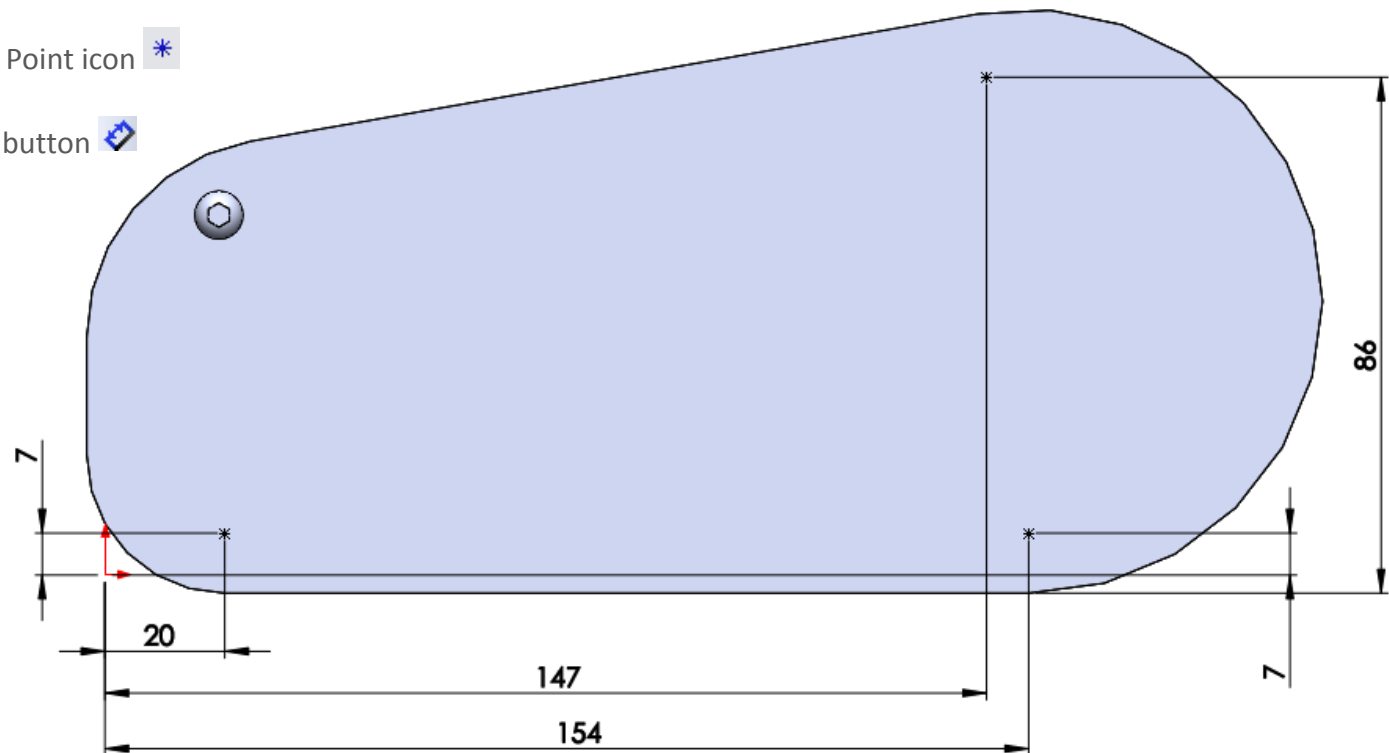
Draw the three points as shown in the picture

Go to **Tools > Sketch Tools > Point** or click at the Point icon 

Add the dimensions by clicking at the dimension button 

Click OK 

Close the sketch 



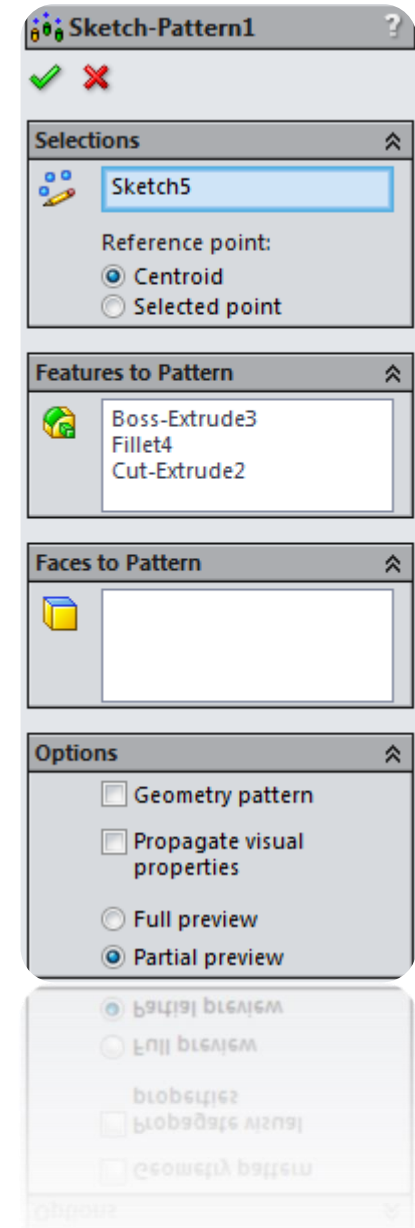
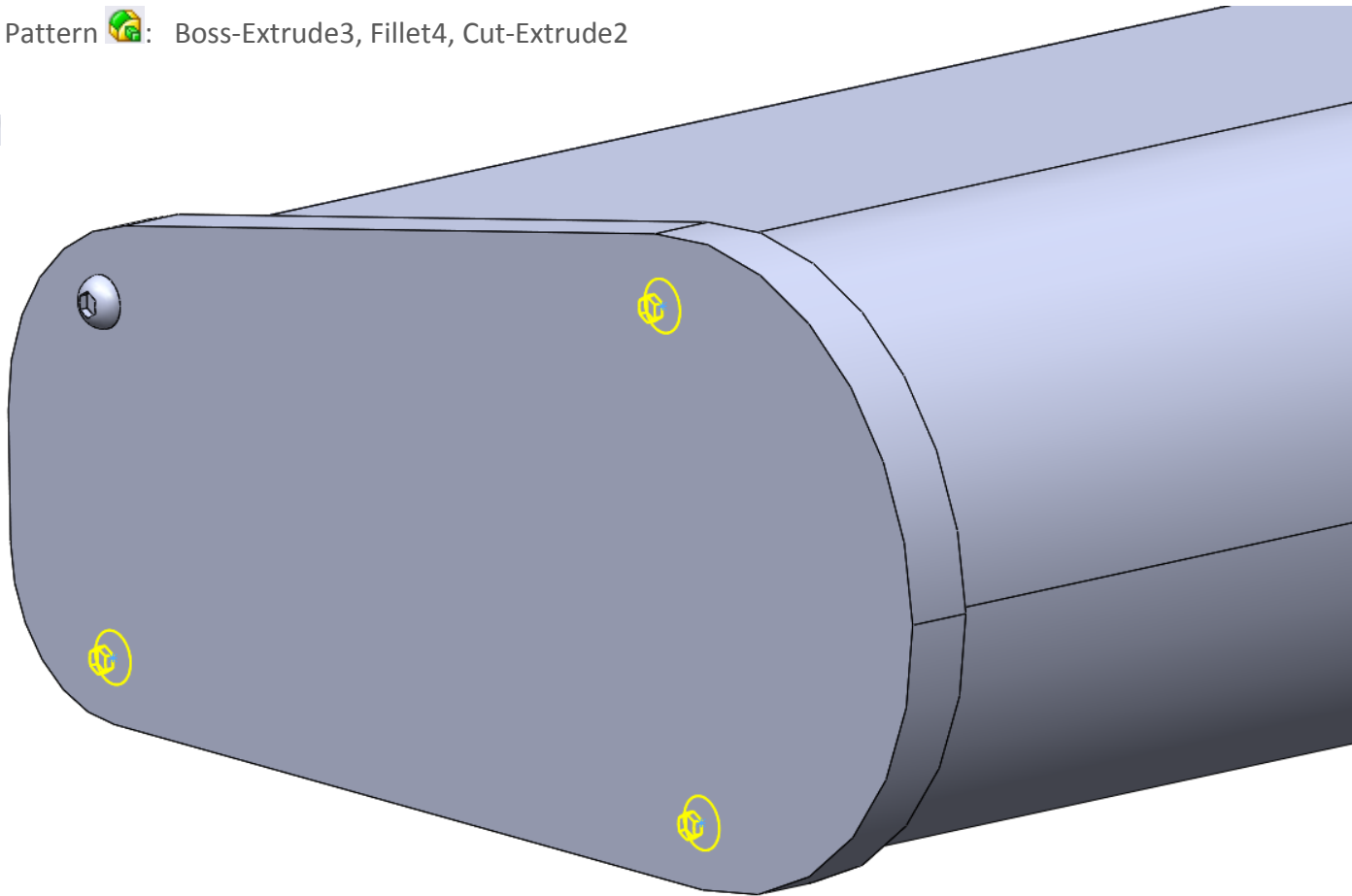
Create a Sketch Driven Pattern

Go to: **Insert > Pattern/Mirror > Sketch Driven Pattern** or click at the Sketch Driven Pattern icon 

Reference Sketch : Select Sketch5 as reference sketch

Features to Pattern : Boss-Extrude3, Fillet4, Cut-Extrude2

Click OK 



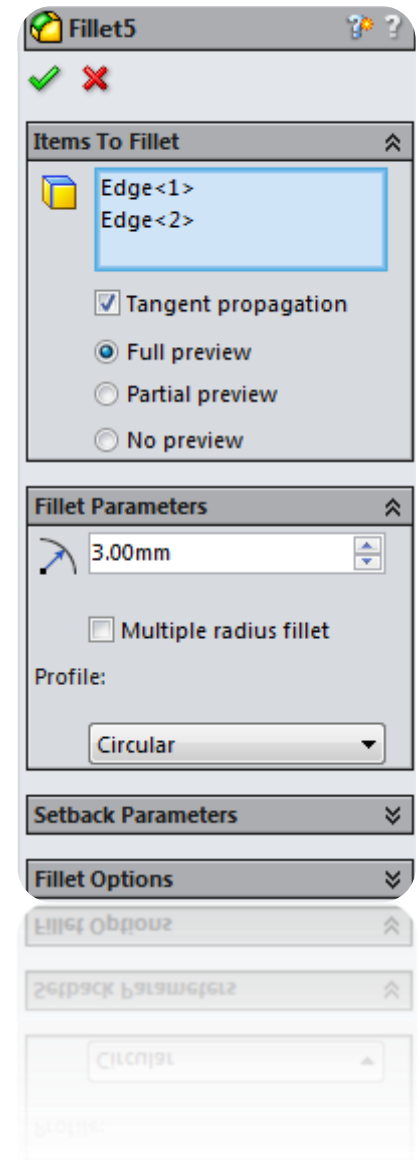
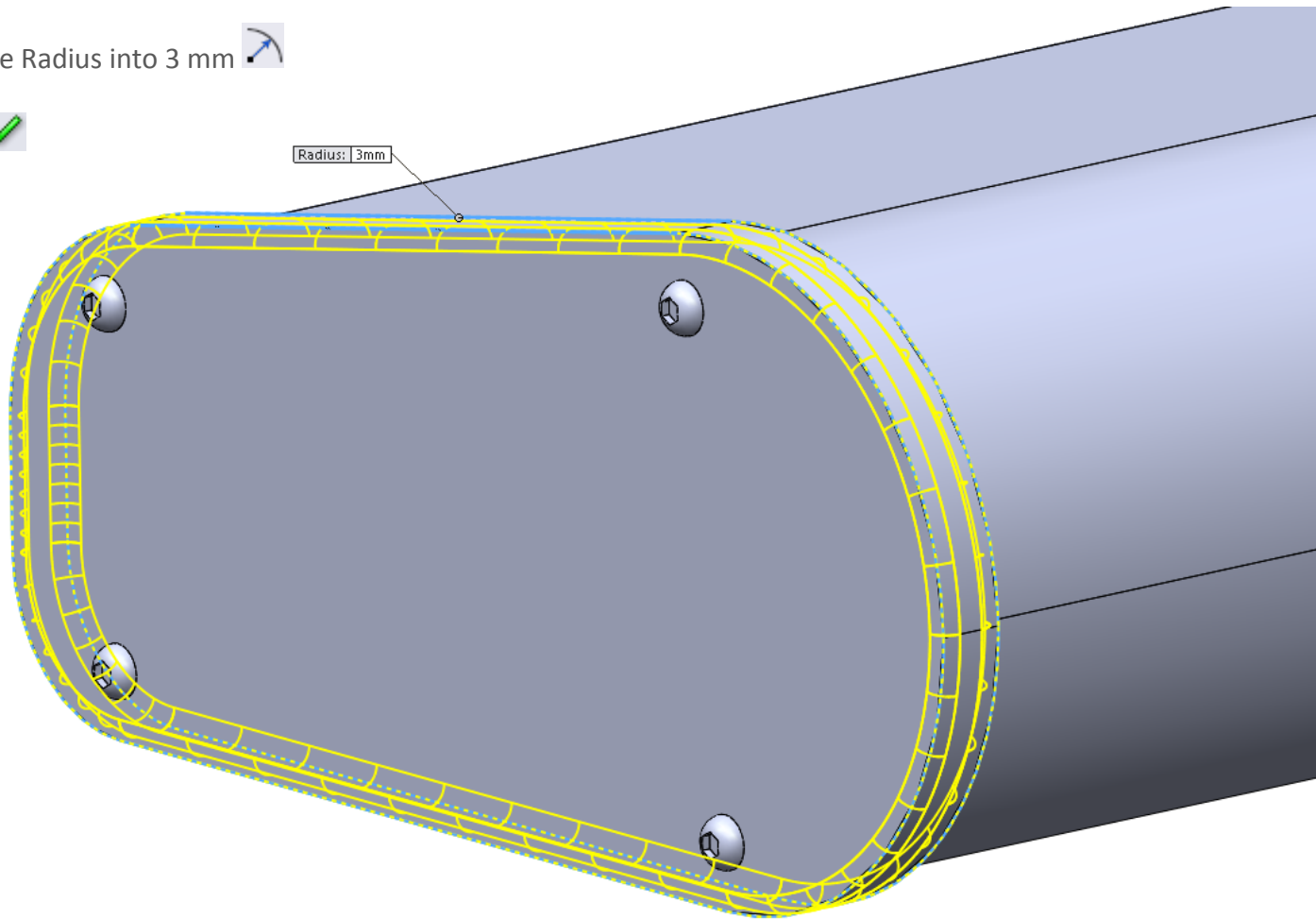
Create a Fillet

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

Click at the two blue edges as shown in the picture

Change the Radius into 3 mm 

Click OK 



Mirror the radar

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

Select the Right Plane as **Mirror Face** 

Select the **Bodies to Mirror** option 

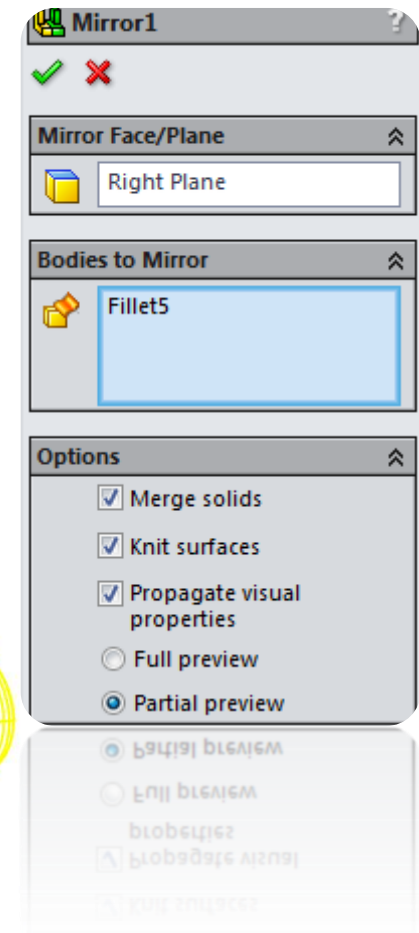
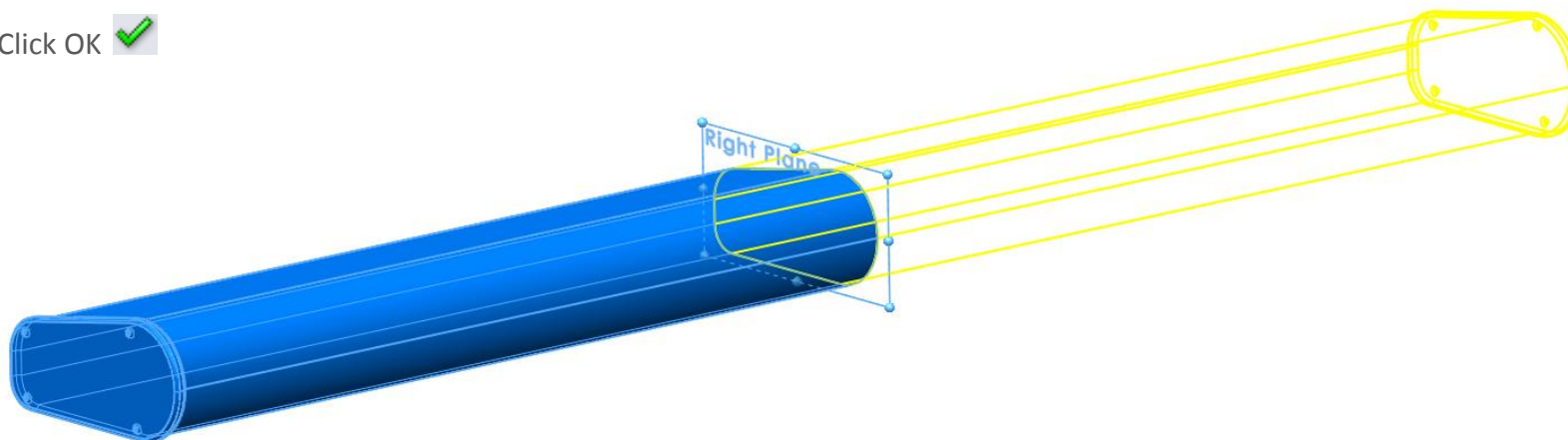
Click at the blue body as shown in the picture

Enable the **Merge solids** option

Enable the **Knit Surfaces** option

Enable the **Propagate visual** properties

Click OK 

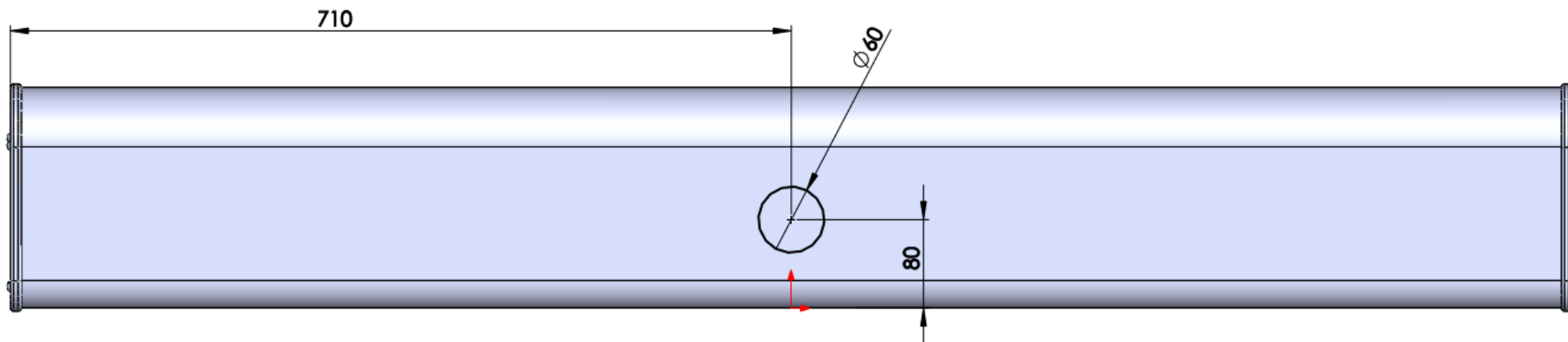
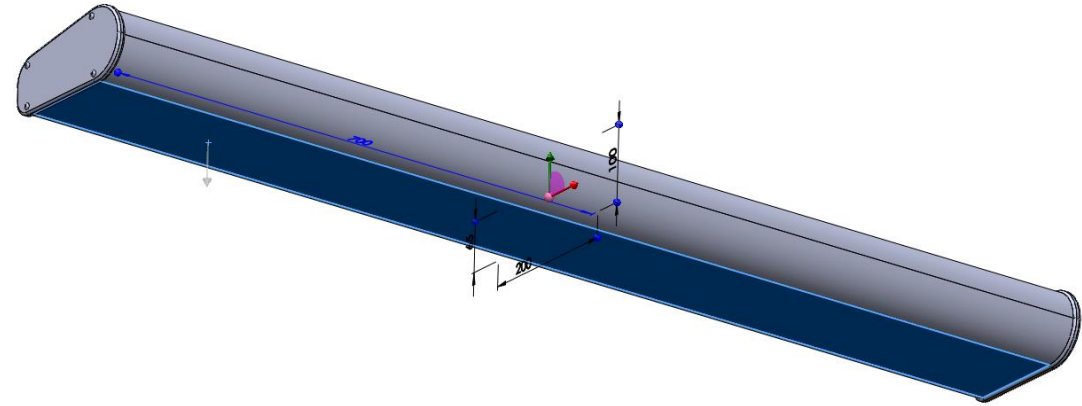


Select the blue bottom surface and click at the **2D Sketch** icon to make a new sketch 

Click at the **Normal To** option to rotate the model normal to the selected plane 

Draw the circle as shown in the picture 

Add the dimension by clicking at the dimension button 

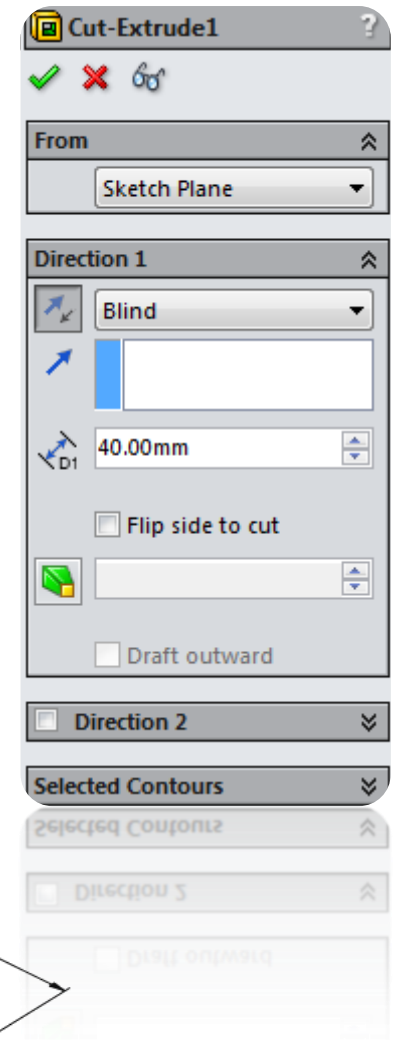
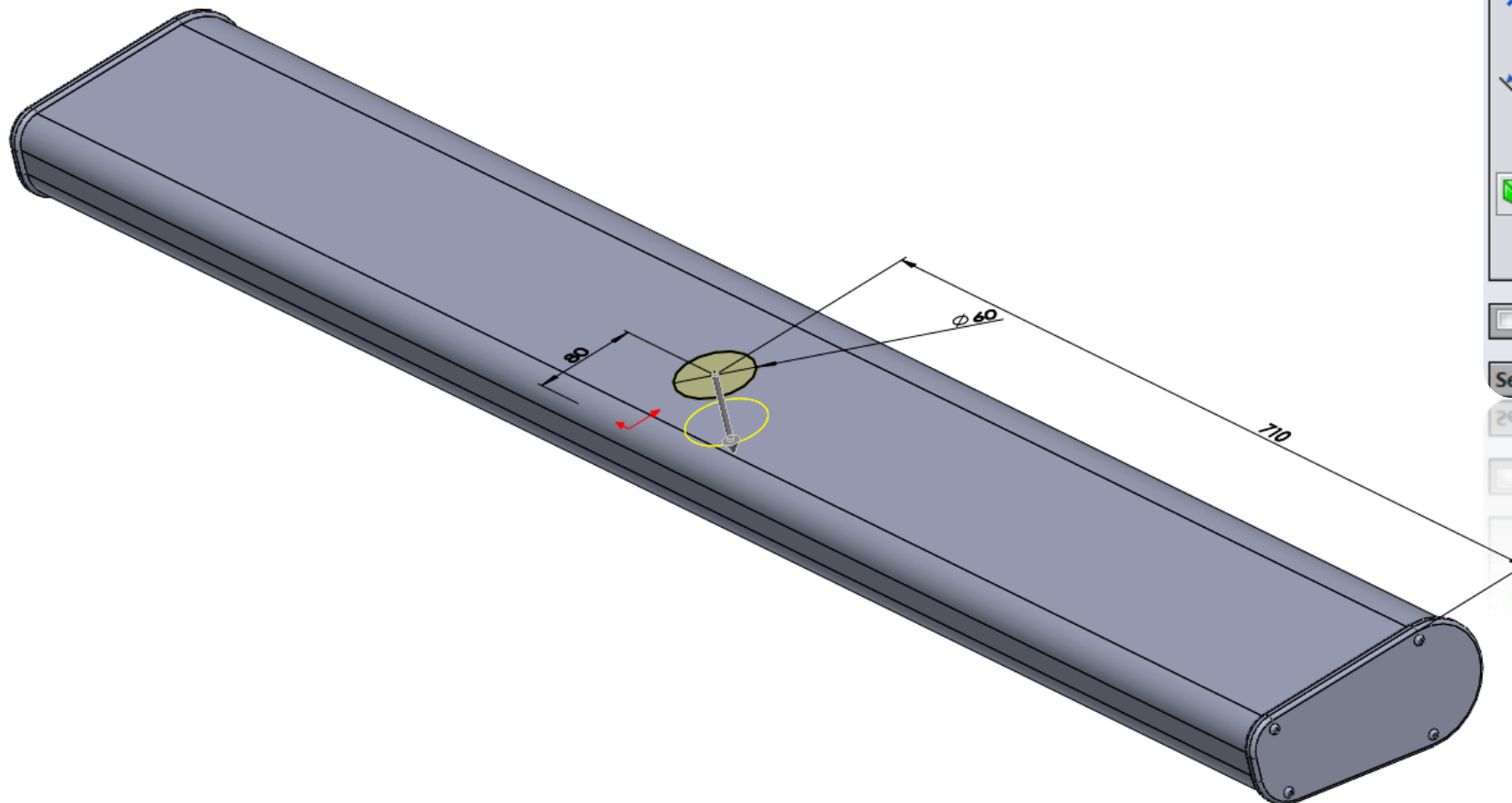


Create an Extruded Cut

Go to: **Insert > Cut > Extrude** or click at the Extruded Cut icon 

Change the Depth into 40 mm 

Click OK 



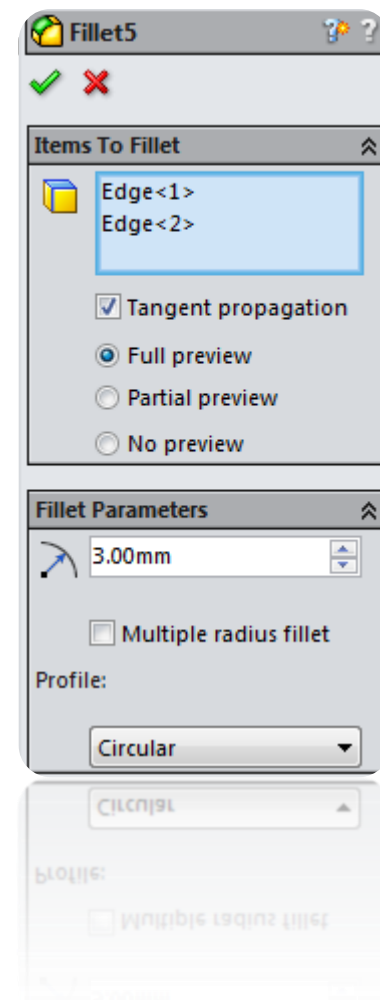
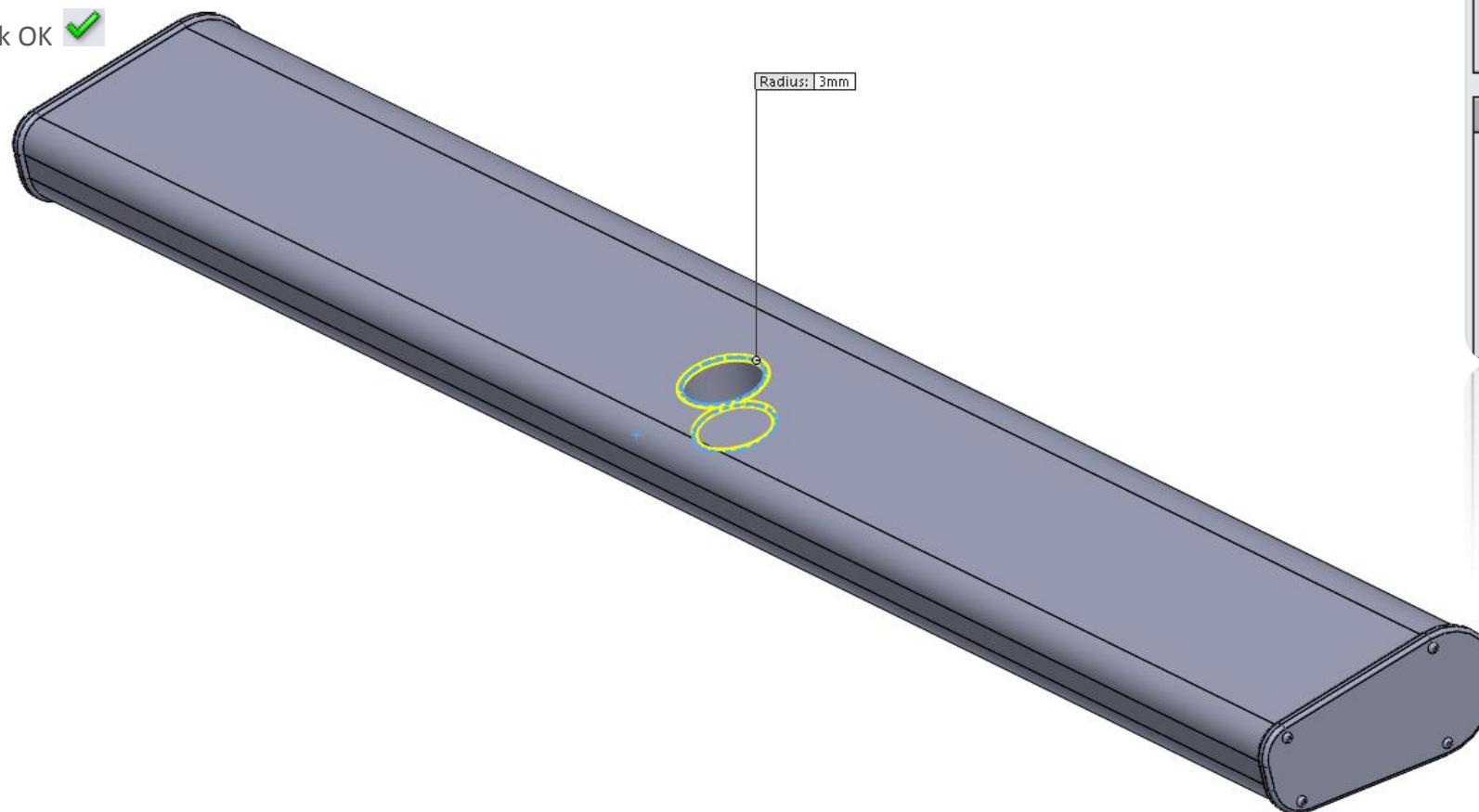
Create a Fillet

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

Click at the two blue edges as shown in the picture

Change the Radius into 3 mm 

Click OK 



Create a new 2D sketch on the blue surface of the radar

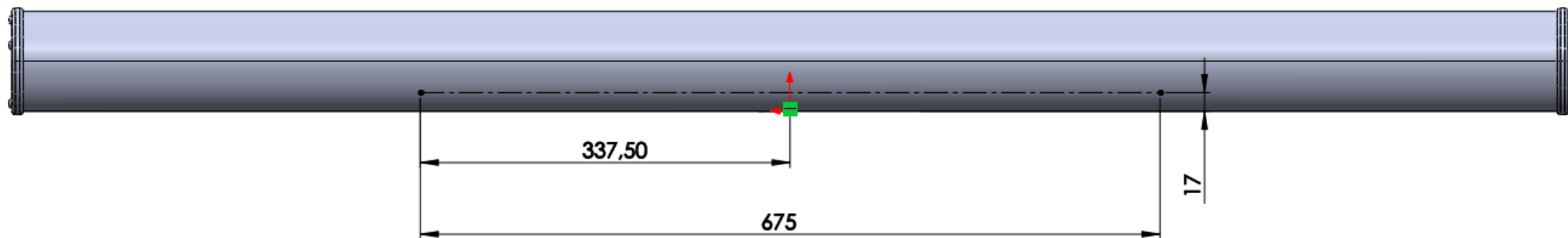
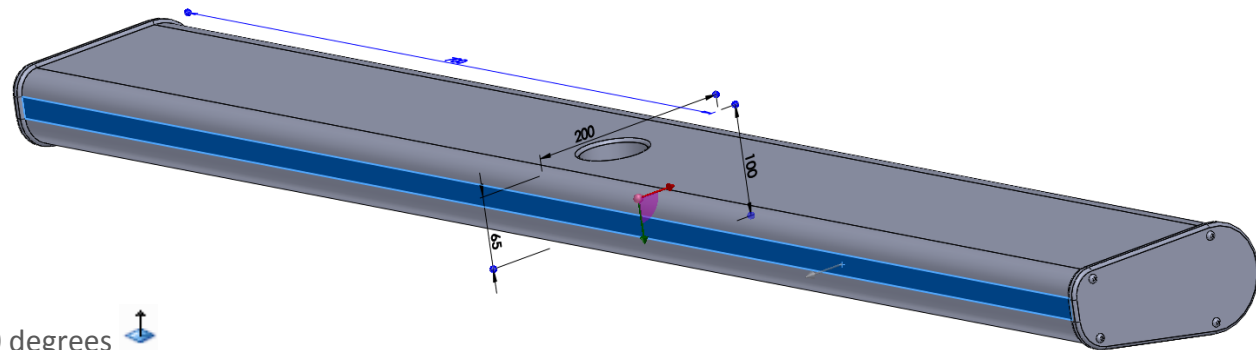
Select the blue surface as and click at the **2D Sketch** icon to make a new sketch 

Click at the **Normal To** option to rotate and zoom the model normal to the selected surface 

Click again at the **Normal To** button to rotate the view 180 degrees 

Draw the Horizontal centerline as shown in the picture 

Change the length and position of the line by clicking at the dimension button 



Add a Text on the Radar

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

Click at the horizontal centerline as reference Curve 

Type a text to project on the radar in the Text box: YACHTTUTORIAL (or anything else)

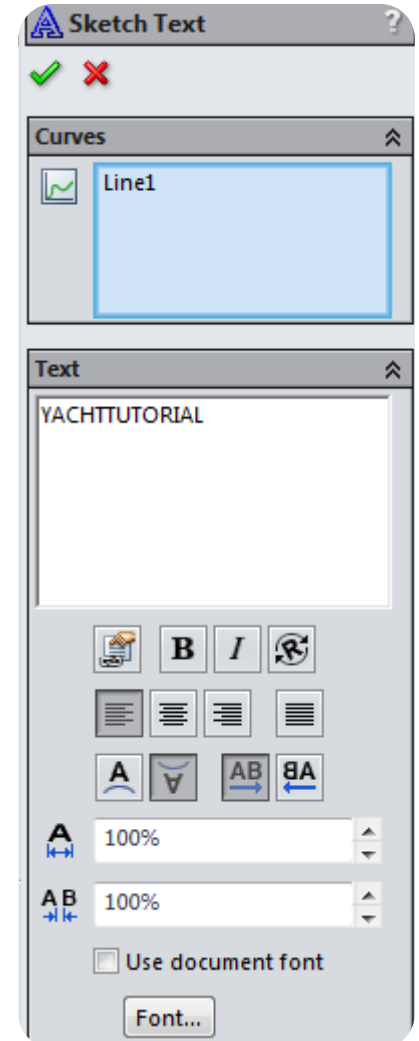
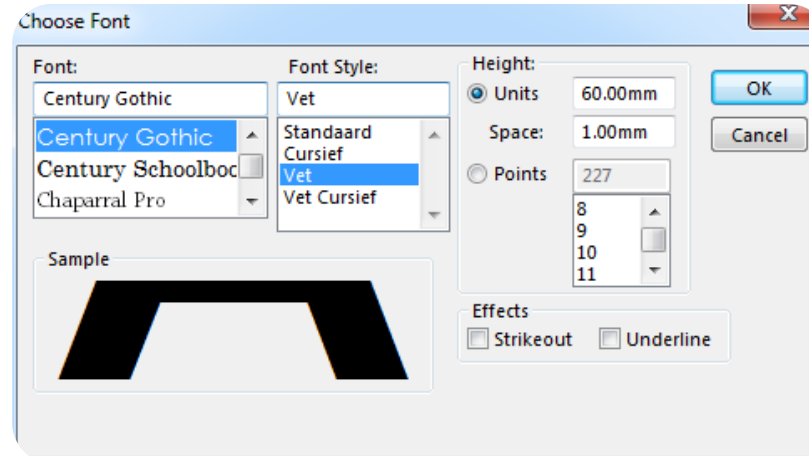
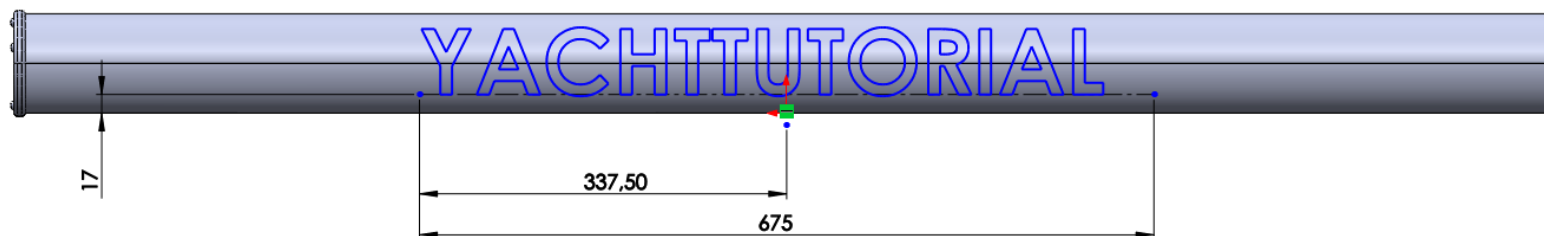
Click at the Font button 

Change the font into: Century Gothic

Font Style: Bold

Change the Text Height into 60 mm

Click OK 



Create Split Lines of the Text

Go to **Insert > Curve > Split Line**
or click at the Split Line icon 

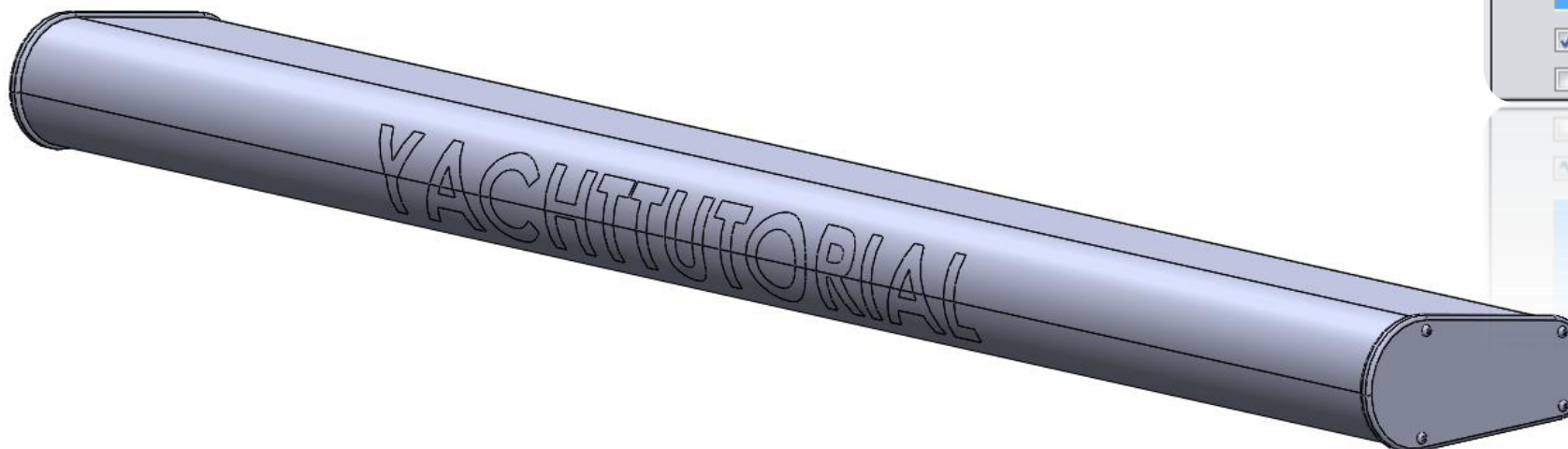
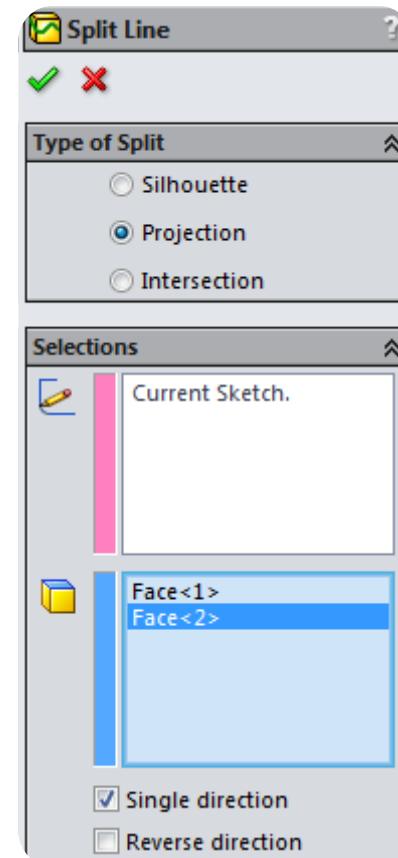
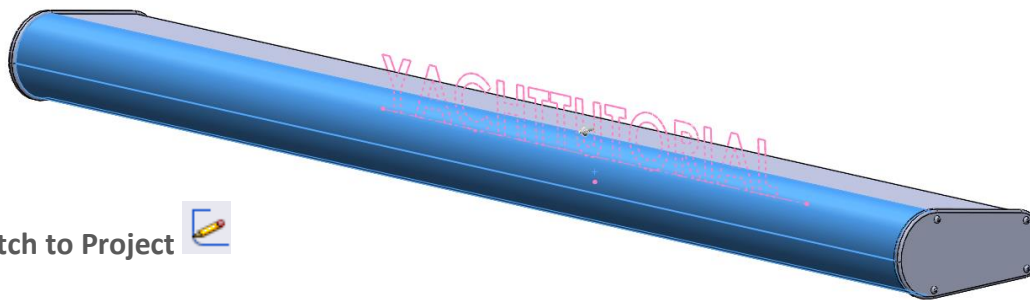
Select the **Projection** option

Select Sketch7 (current sketch) as **Sketch to Project** 

Select the two blue surfaces as **Faces to Split** 

Select the **Single direction** option

Click OK 



Change the color of your radar

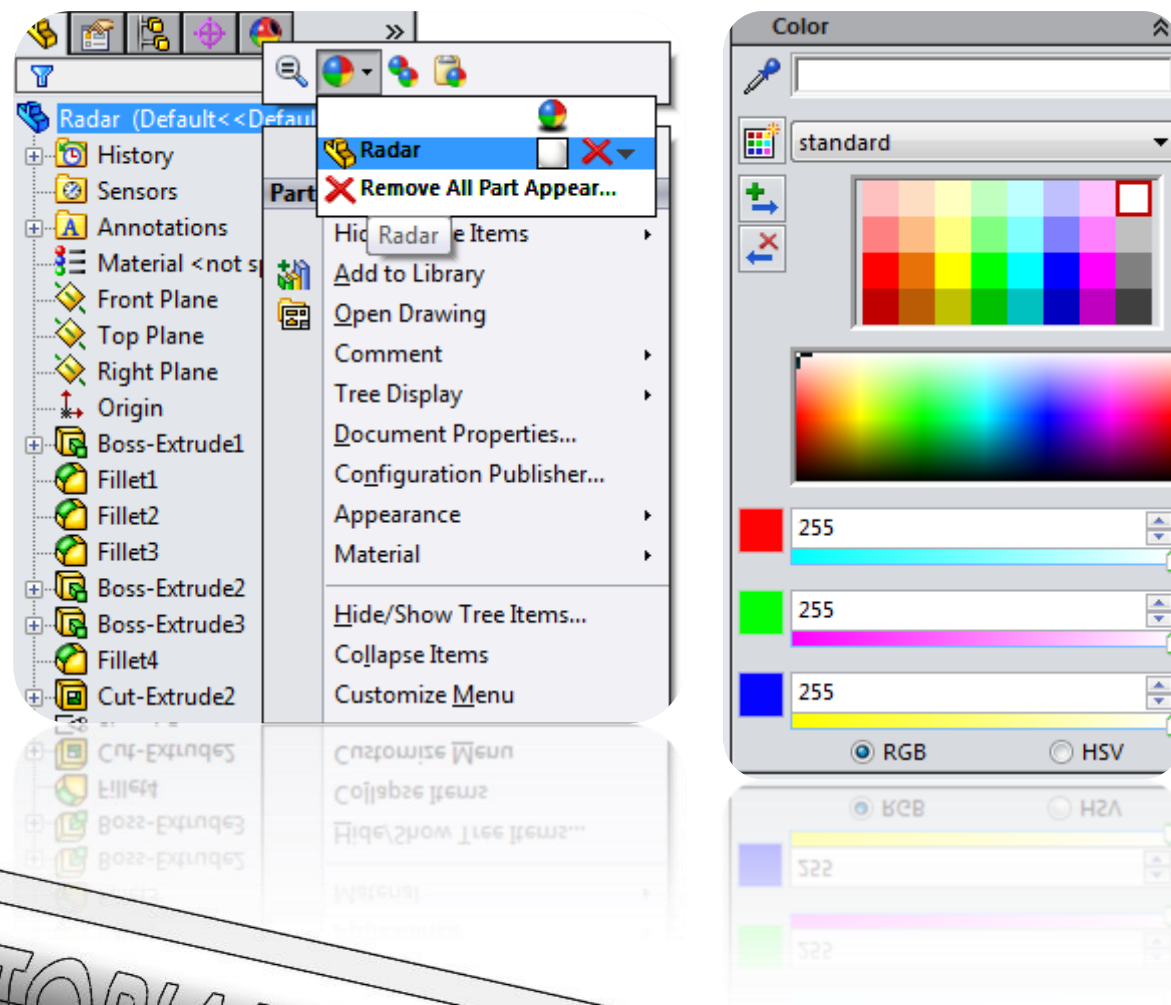
Right click at **Radar** in the feature tree

Click at the appearances icon 

Click at Radar 

Change the color into white

Click OK 



Change the color of the Text

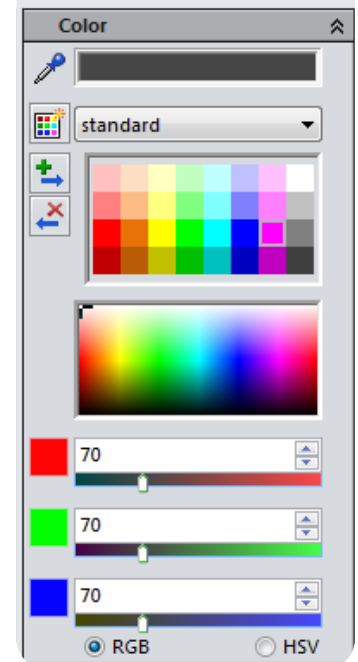
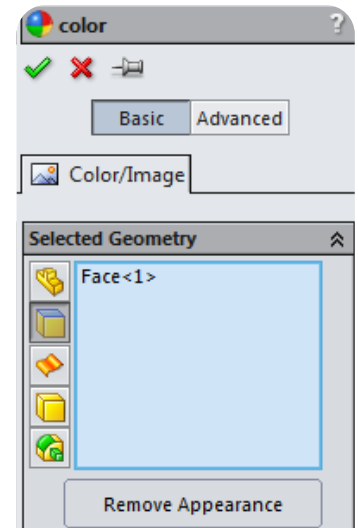
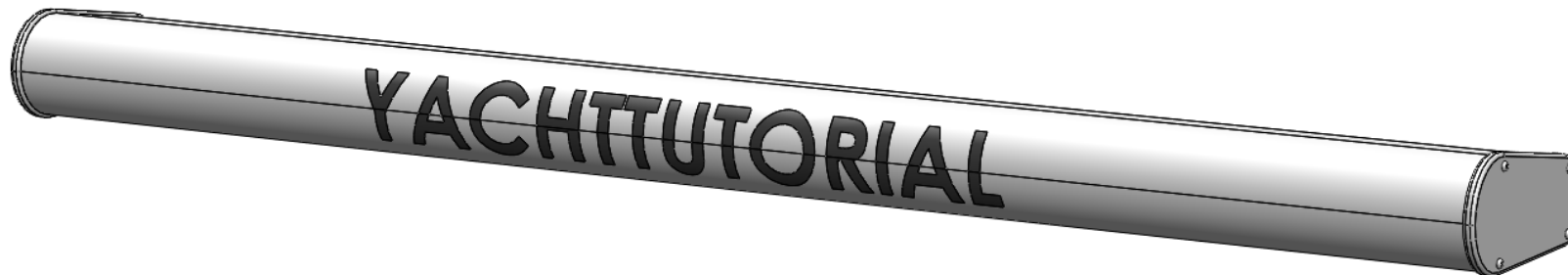
Click at one of the Text surfaces, hold down the **Control** button and select all the other Text surfaces as well

Right click at the appearances button 

Click at the Face option 

Change the color into a custom dark grey by changing the RGB numbers into 70, 70, 70

Click OK 



Hide  all planes and sketches

Save  the file with the following name: **Radar.sldprt**



That's all!

You have just read this second eBook, you've modeled your own Radar in SolidWorks and you know you can dramatically improve your SolidWorks modeling skills by step by step SolidWorks eBooks like this one.

If you thought this eBook was helpful, you're really going to like my complete SolidWorks yacht course. Click this link to find out more:

www.YachtTutorial.com/offer.html

And keep an eye at your Inbox because I will send you Part 3 of this step-by-step modeling course in **two more days**.

Stay tuned,

Jan

In the next eBook you will learn how to setup the blueprints of your yacht in SolidWorks.

