



DIFFICULTY LEVEL



J.W. Zuyderduyn

HOW TO MODEL THE **1. PROPELLER** IN SOLIDWORKS®

About the Author

Hi, my name is Jan-Willem Zuyderduyn and I am the founder and owner of YachtTutorial.com. I've been working since 1999 with 3d CAD software and since 2004 with SolidWorks. In that time I've learned a lot about all the possibilities with SolidWorks.

It took me many years to learn everything about SolidWorks. I have been asked many times how to model and render 3D models using SolidWorks. The last few years I've written multiple books and tutorials about SolidWorks. One of them is the SolidWorks Chopper Tutorial <http://www.learnsolidworks.com/chopper> where I teach you how to model a Chopper from A to Z. My goal is to help as many SolidWorkers as possible. That's why I've created the SolidWorks blog www.LearnSolidWorks.com.

The last years I was looking for a successor of the Chopper tutorial. My dream was to model a luxury Yacht in SolidWorks. So I did and when it was finished I couldn't resist to write a brand new SolidWorks course about it. It took me three years but it's finally here!

I am very proud to introduce the SolidWorks Yacht course. In this advanced surface modeling course I will teach you **exactly** how to model the exterior of the Sunseeker Predator 108 in SolidWorks. This is an existing yacht of Sunseeker (I am a huge Sunseeker fan) so the credits for the design will fully go to Sunseeker International.

Despite the fact that it's a sophisticated model it doesn't need to be too difficult with the instructions of this course. Today I will show you how to model part #1 of your yacht in SolidWorks.

Each tutorial starts with a line drawing of the Yacht you will make. You can find the line drawing on the next page. I've highlighted the treated part of each tutorial in **blue**. As you can see we start with the **propeller** of the yacht. Your yacht has 3 of them but I'll show you just one. :)

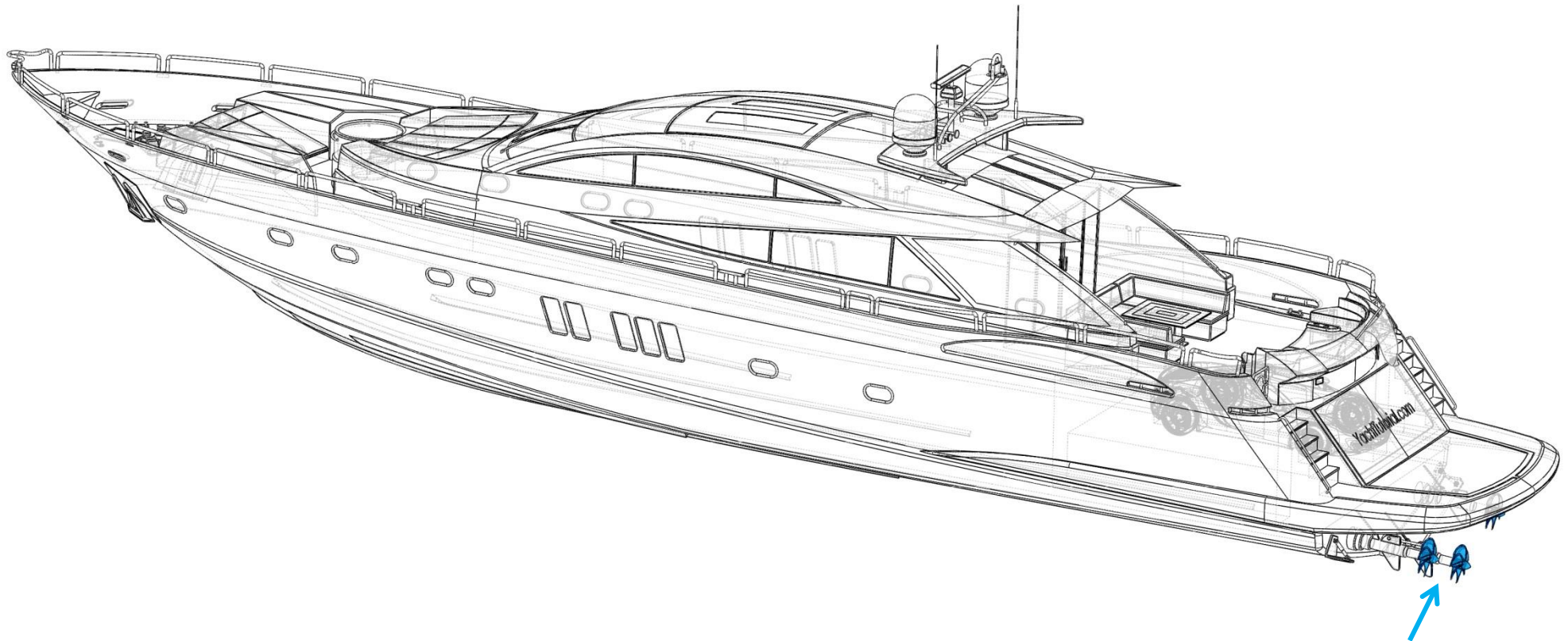
Let's get started! ☺

Jan



[Linkedin](#) Jan-Willem Zuyderduyn

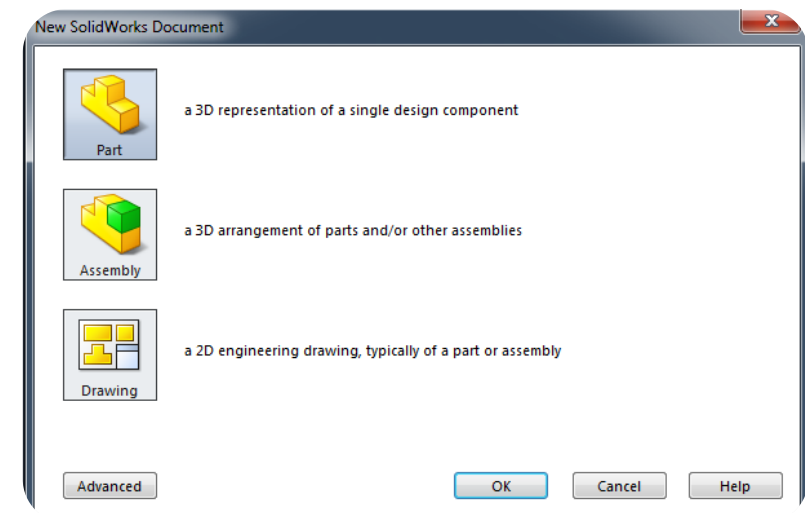




HOW TO MODEL THE 1. PROPELLER IN SOLIDWORKS®?

Open SolidWorks

Go to: **Start > Programs > SolidWorks** 



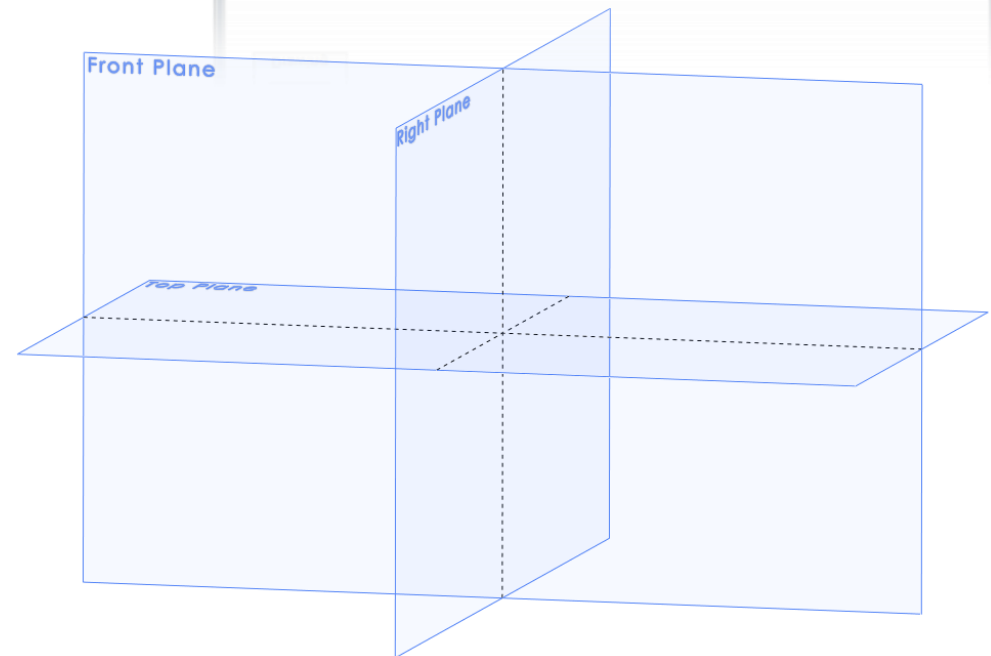
Create a new part

Open a new part with model units set to millimeters.

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



Select the three planes in the feature tree (hold down the **Control** key) and make them visible by clicking on the glasses. 



Create a new sketch on the Right Plane

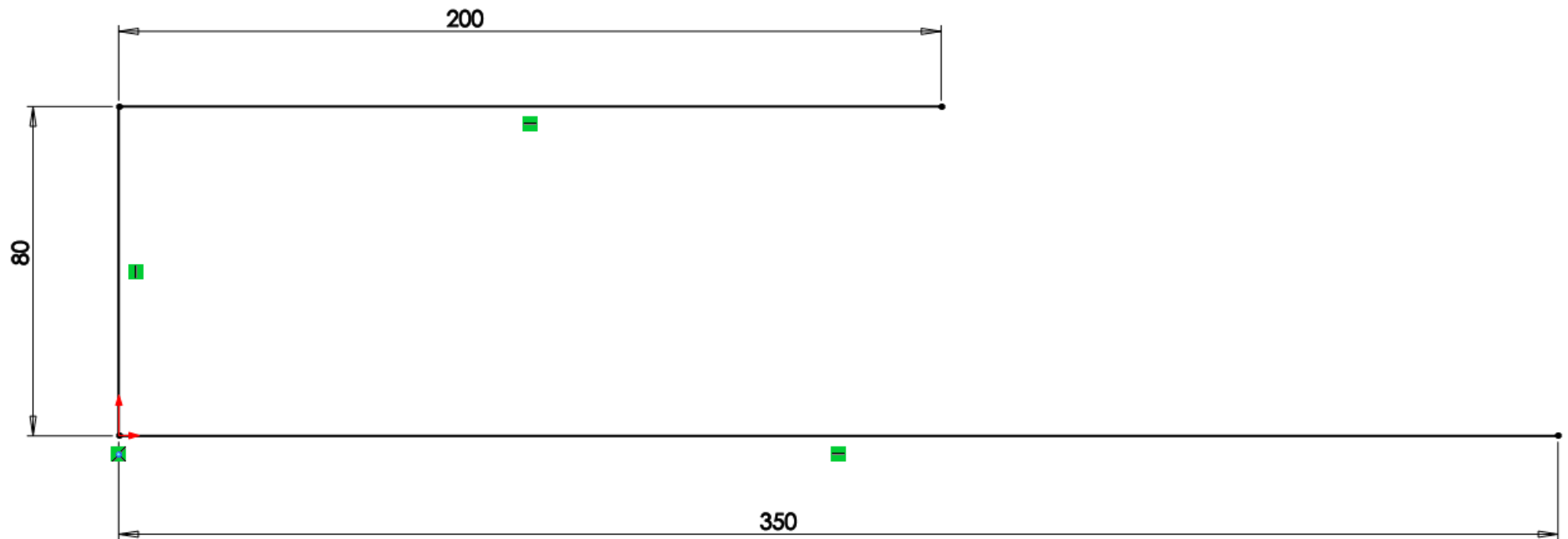
Select the **Right Plane** in the Feature Manager

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

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

Draw the three straight lines  as shown in the picture

Change the dimensions of the lines 

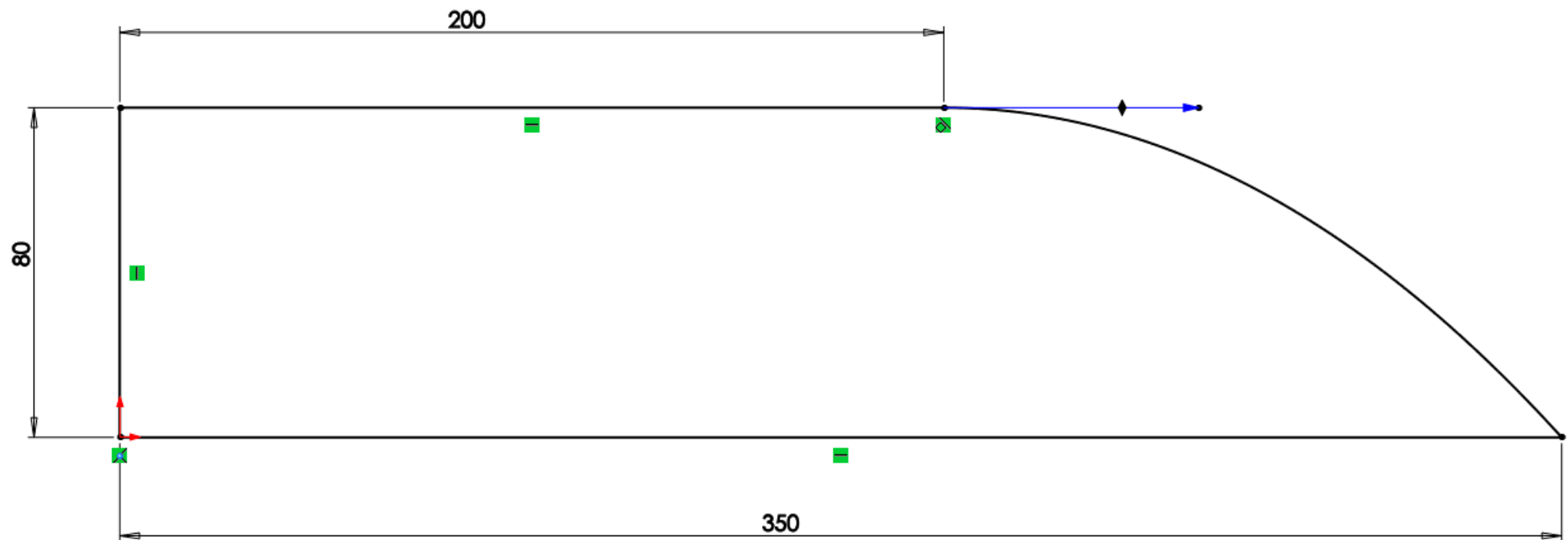
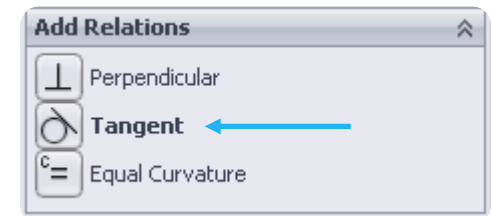


Draw the spline as shown in the picture 

Add a tangency relation on the transition between the horizontal line and the spline

Select the horizontal line, hold down the **Control** key and select the spline. Click at the Tangency icon  in the **Add Relations** bar to add a tangency relation between the line and the spline.

Click OK 

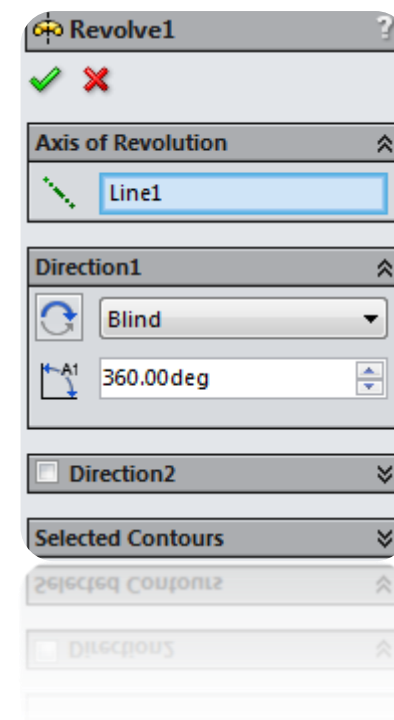
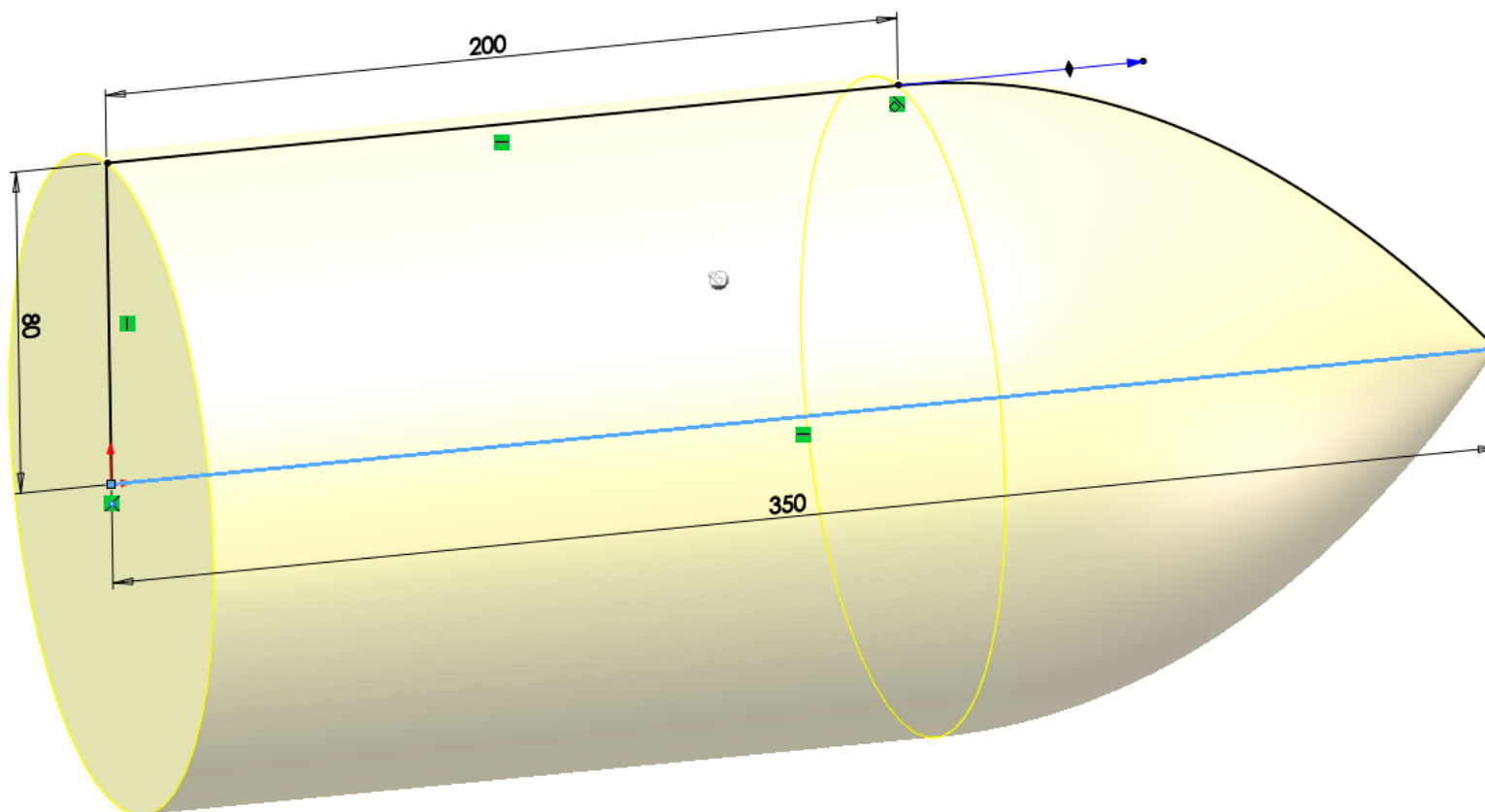


Create a Revolve

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

Axis of revolution : Select the blue line as shown in the picture

Click OK 



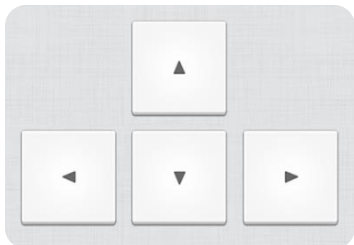
Create a new sketch on the Top Plane

Select the **Top Plane** in the Feature Manager

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 Plane. 

NOTE: You can also use the arrow keys of your Keyboard to rotate your model



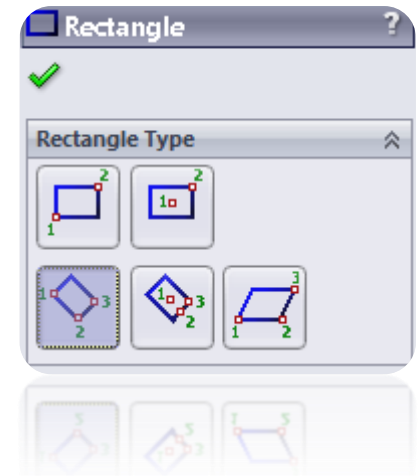
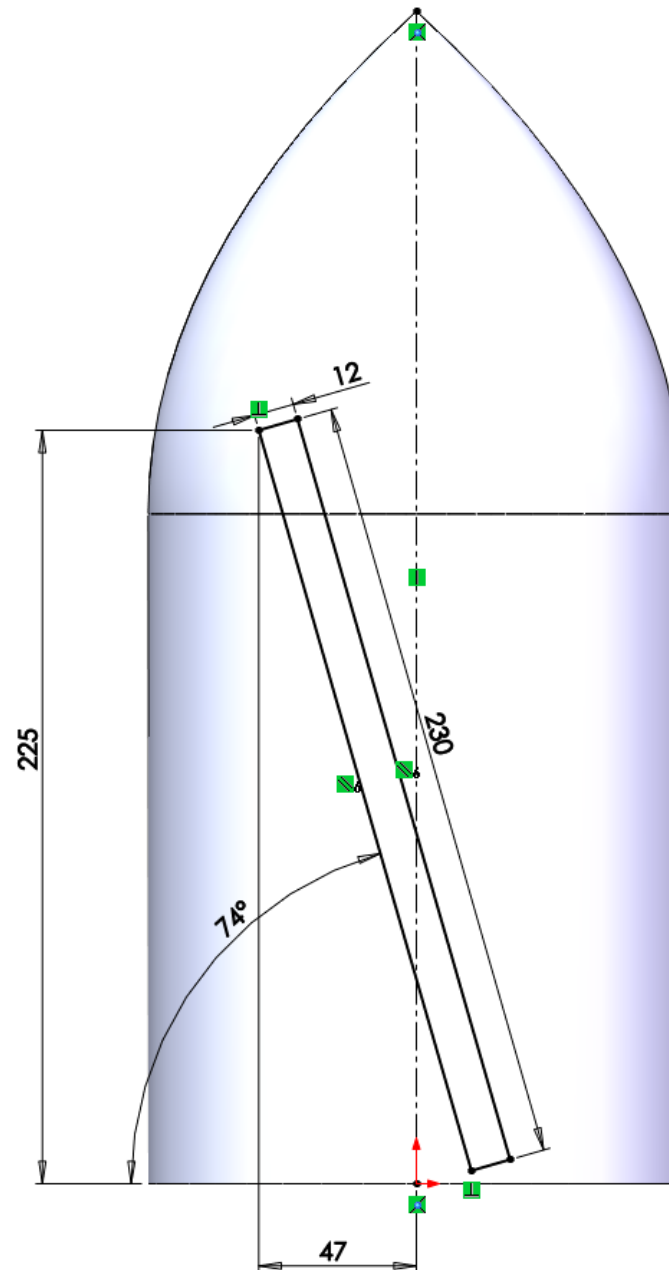
Create a 3 Point Corner Rectangle

Go to **Tools > Sketch Entities > 3 Point Corner Rectangle** or click at the 3 Point Corner Rectangle icon 

Draw the rectangle as shown in the picture

Change the dimensions of the rectangle 

Close the sketch 



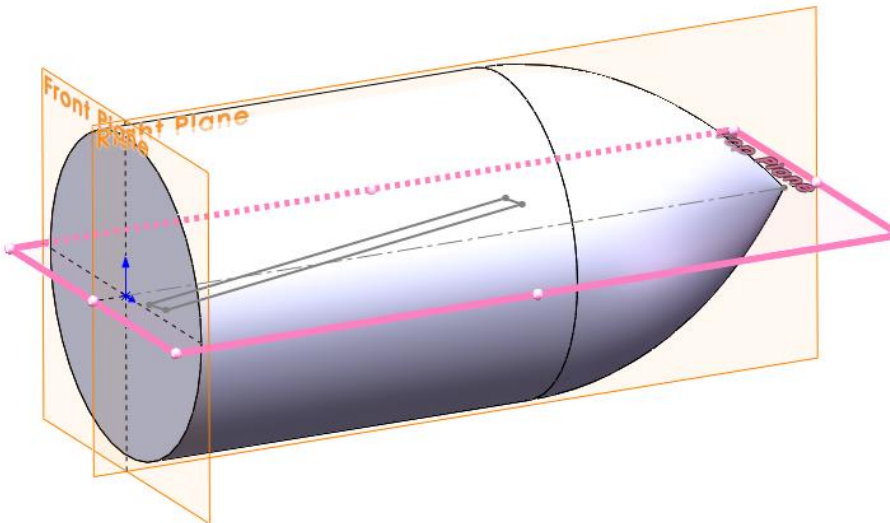
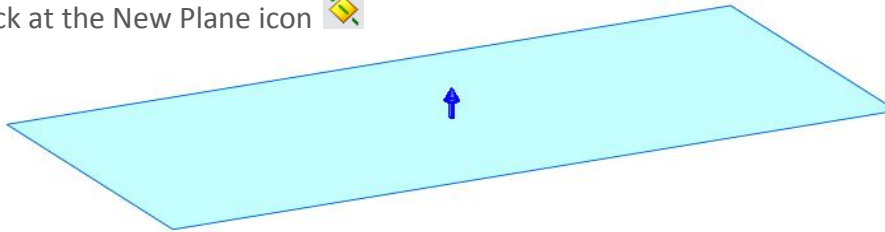
Create a New Plane

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

First Reference : Select the Top Plane

Change the Offset Distance  into: 360 mm

Click OK 



 Plane1 ?

Message 

Fully defined

First Reference 

 Top Plane

 Parallel

 Perpendicular

 Coincident

 0  

 360.00mm  

☐ Flip

 Mid Plane

Second Reference 



Third Reference 



Options 

☐ Flip normal

☐ Flip normal

Obsolete 



Third Reference 



Create a Solid Loft

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

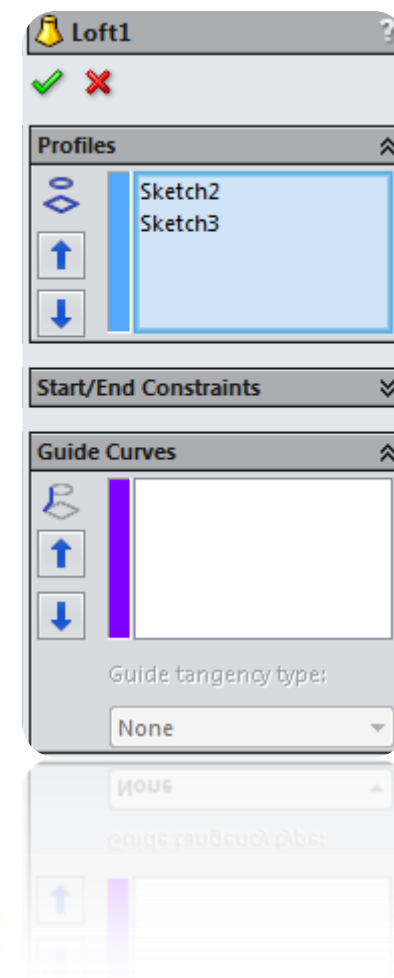
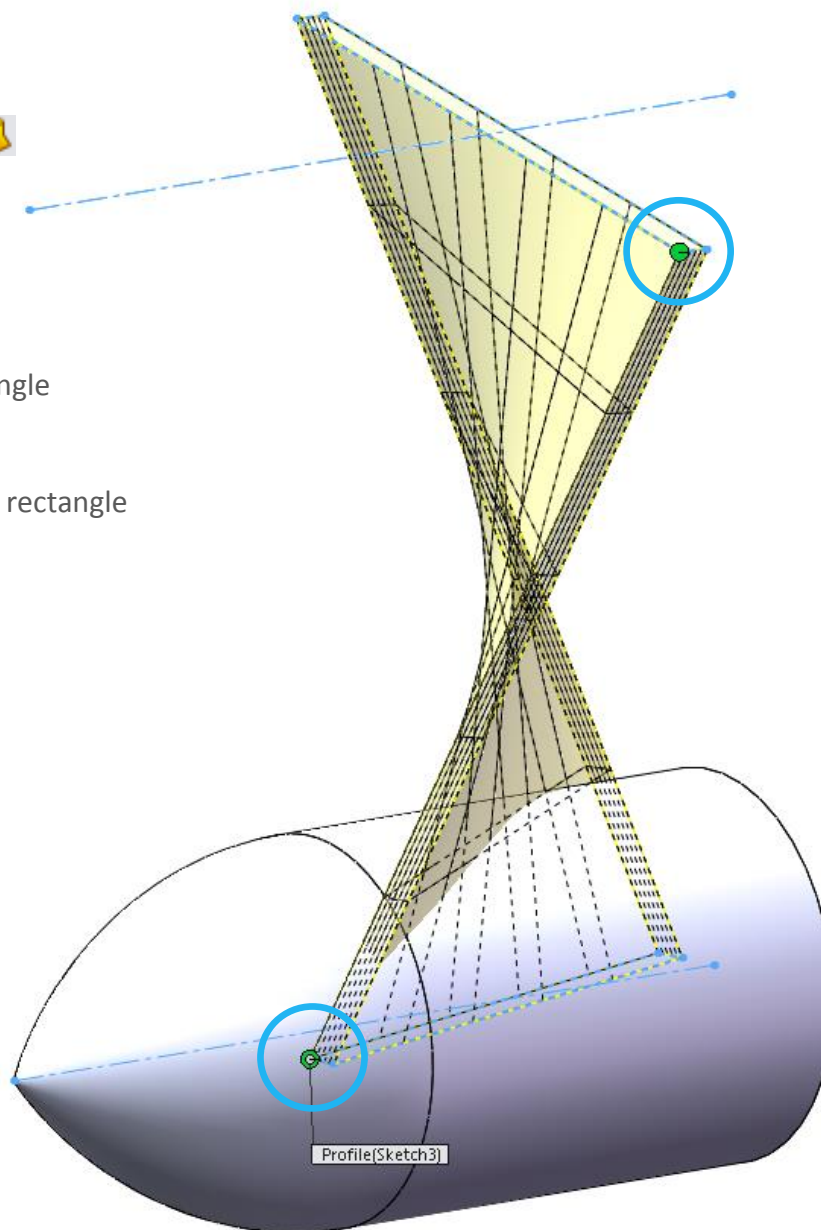
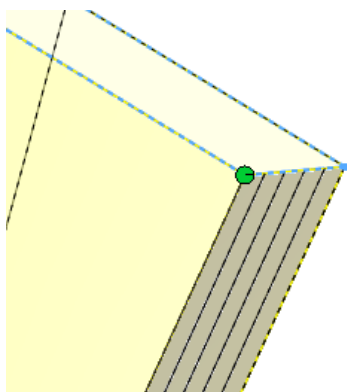
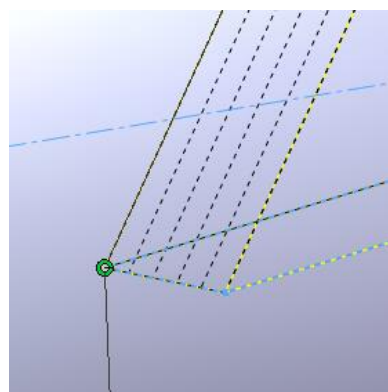
Click in the Profiles box 

Select **Sketch2** and **Sketch3** as shown in the picture

Make sure that the green balls are both in the same rectangle corner as shown in the picture and blue circle details

If not, **click and drag** one ball to the another corner of the rectangle

Click OK 



Create two Fillets on the edges of the loft

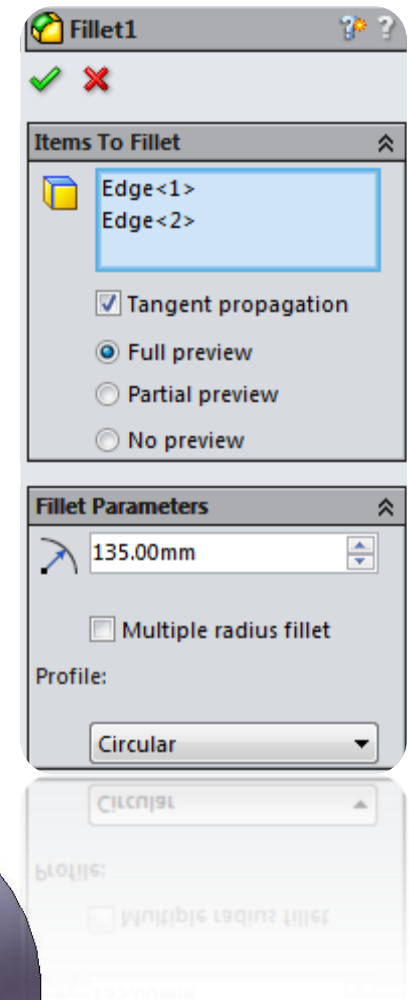
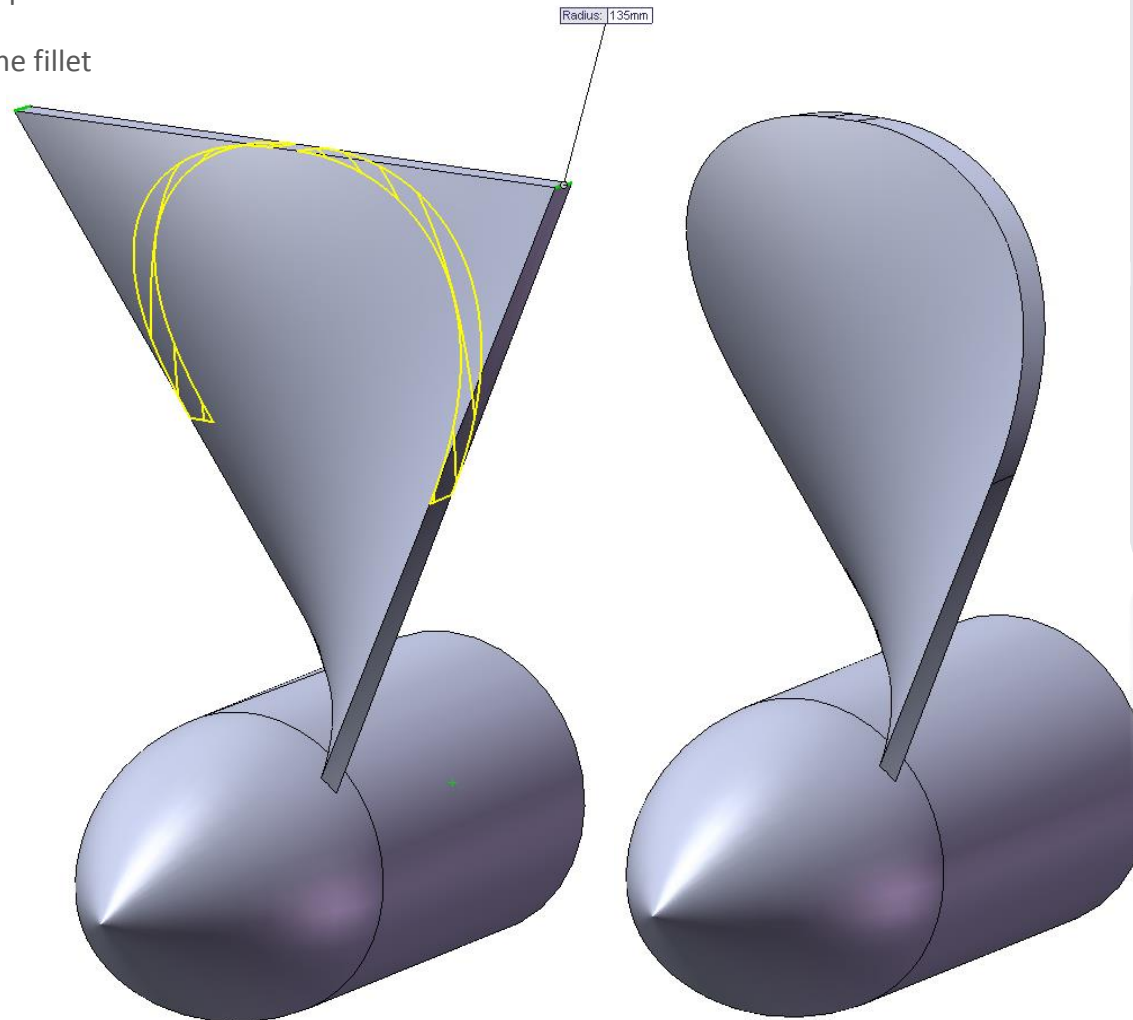
Go to: **Insert > Features > Fillet/Round** or click at the Fillet icon 

Select the two edges as shown in the picture

Select *Full preview* for a preview of the fillet

Change the Radius into 135 mm 

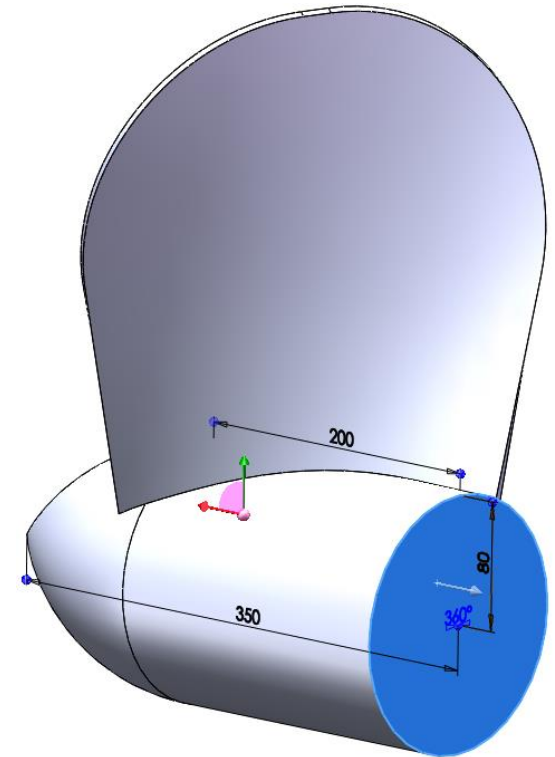
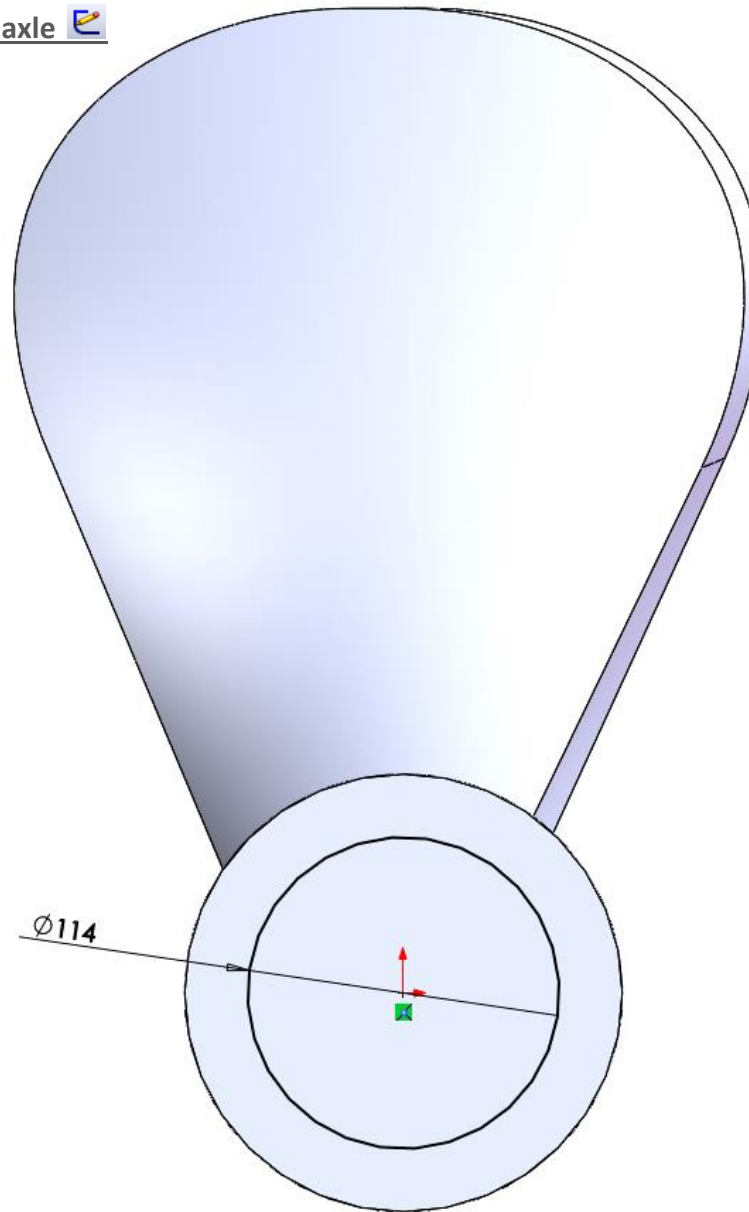
Click OK 



Create a new sketch on the blue surface of the axle 

Draw the circle as shown in the picture 

Change the size of the circle by
clicking at the dimension button 



Create an Extruded Cut

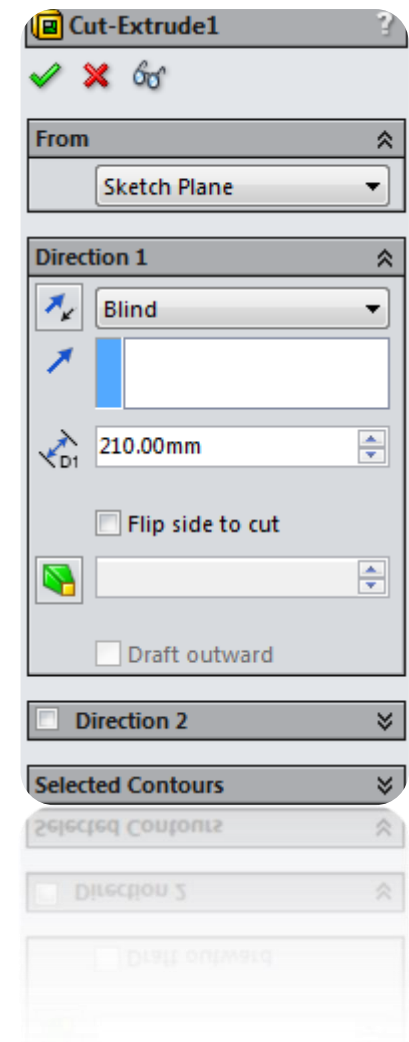
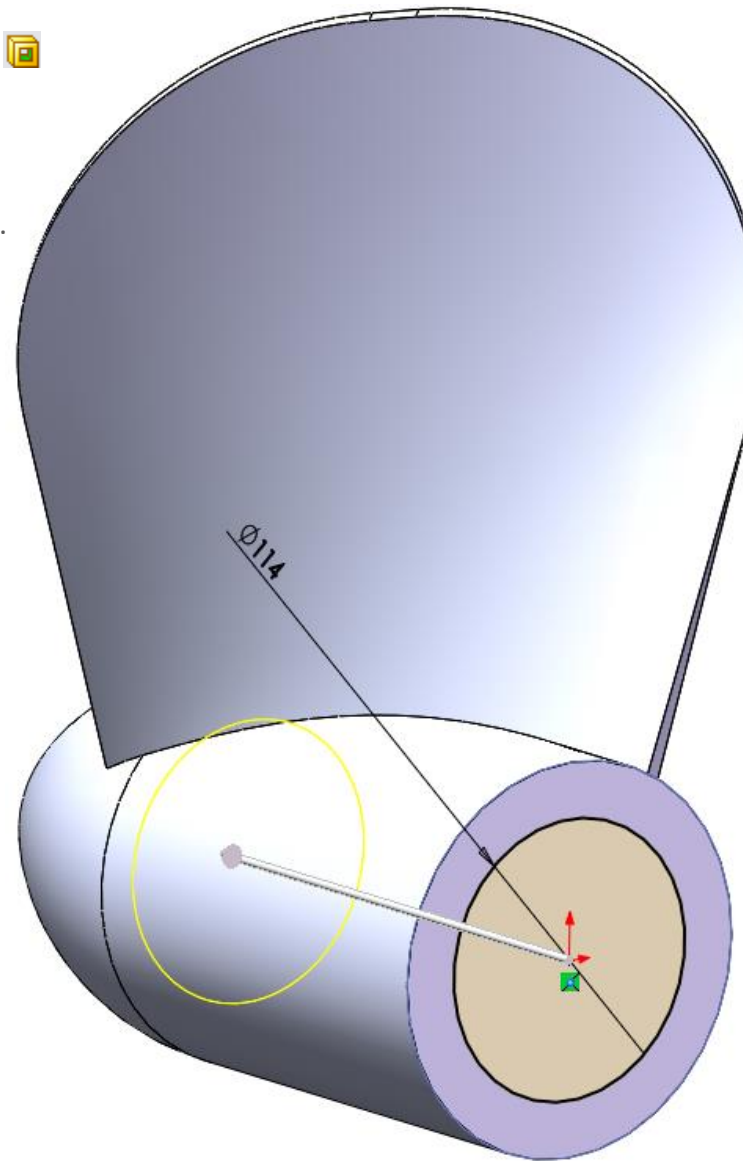
Go to: **Insert > Cut > Extrude** or click at the Extruded Cut icon 

Change  into 210 mm

Make sure that the Extruded Cut direction is pointed **inwards**.

If not click at the **Reverse** icon 

Click OK 



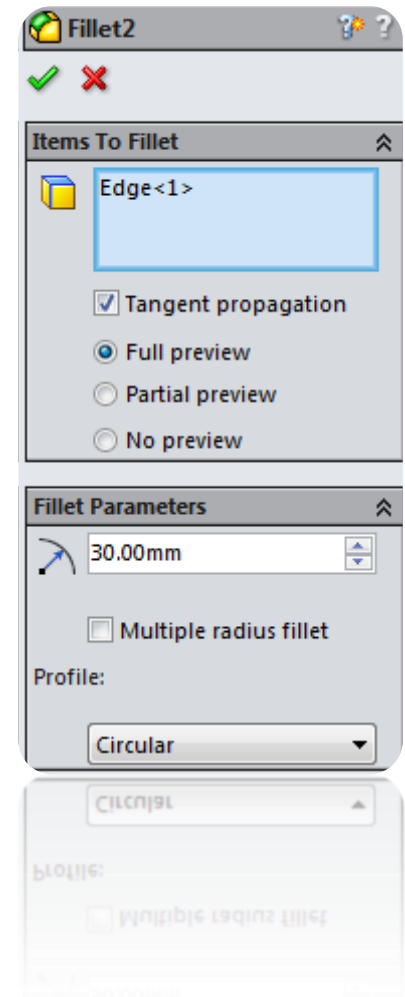
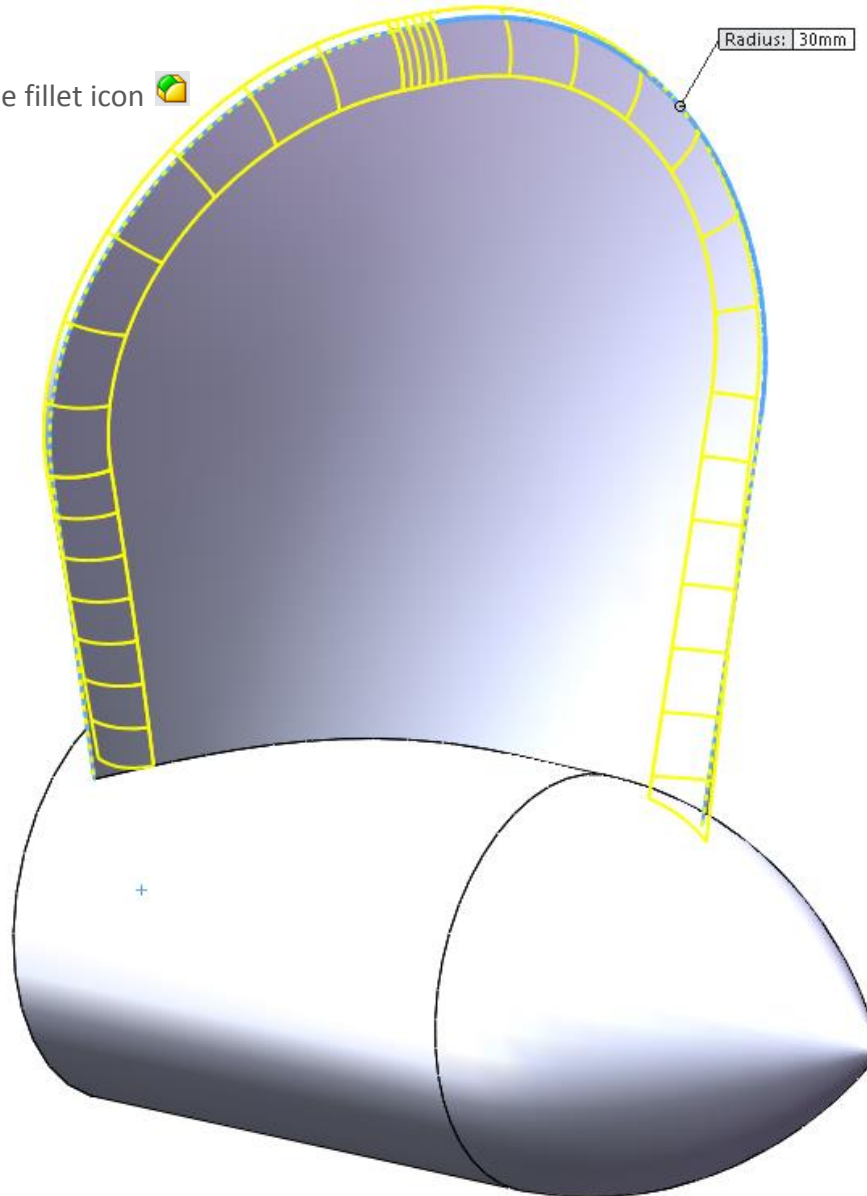
Create a fillet on the edge of the blade

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

Select the edge as shown in the picture

Change the Radius into 30 mm 

Click OK 



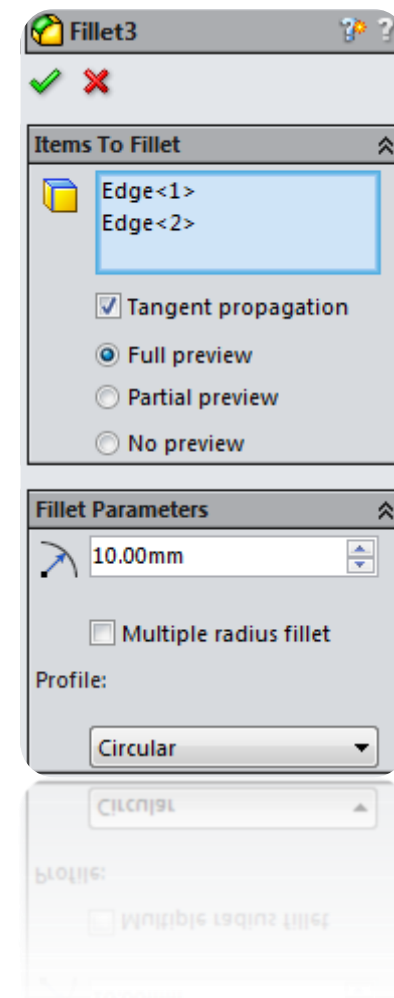
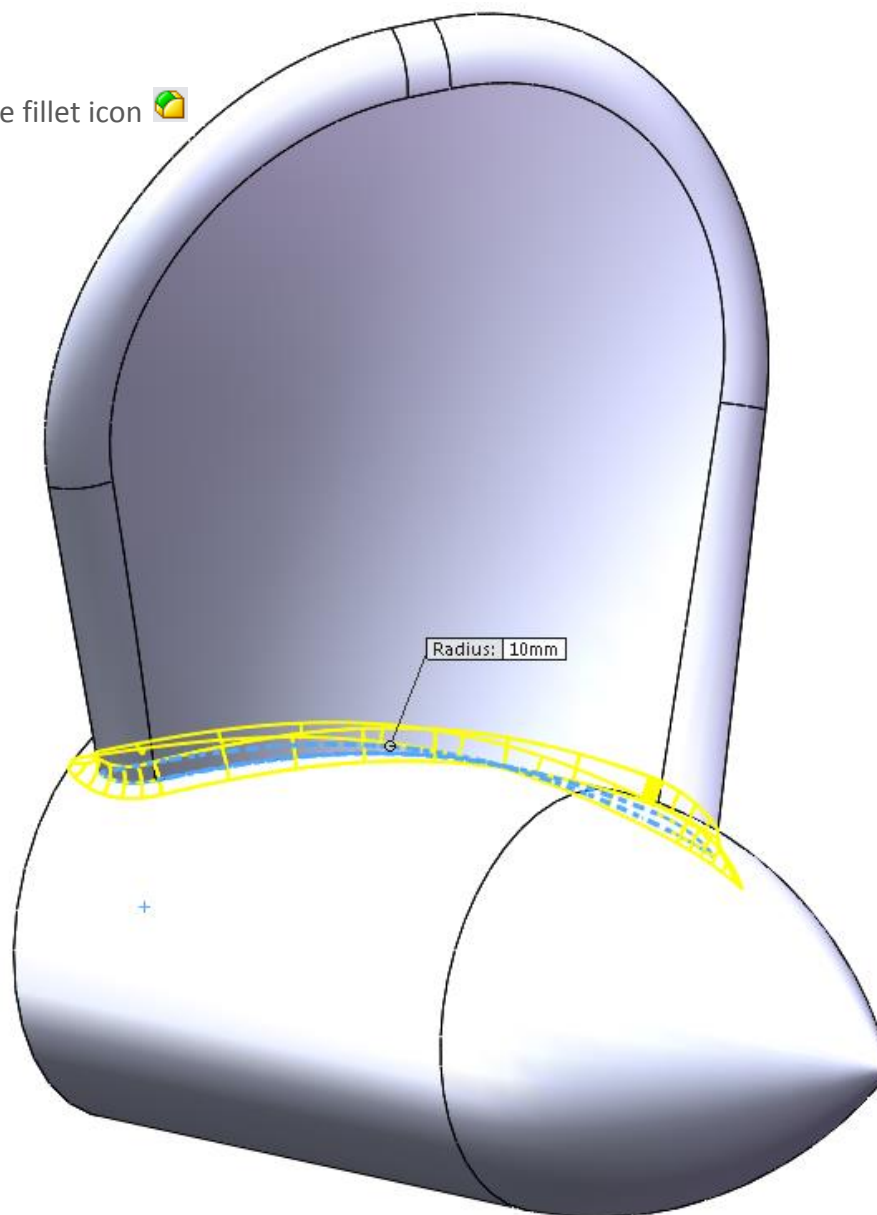
Create a fillet on the edge of the axle and blade

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

Select the two edges as shown in the picture

Change the Radius into 10 mm 

Click OK 



Create a Circular Pattern

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

Pattern Axis  : Select the blue surface as shown in the picture.

Total Angle  : 360 deg

Number of instances  : 6

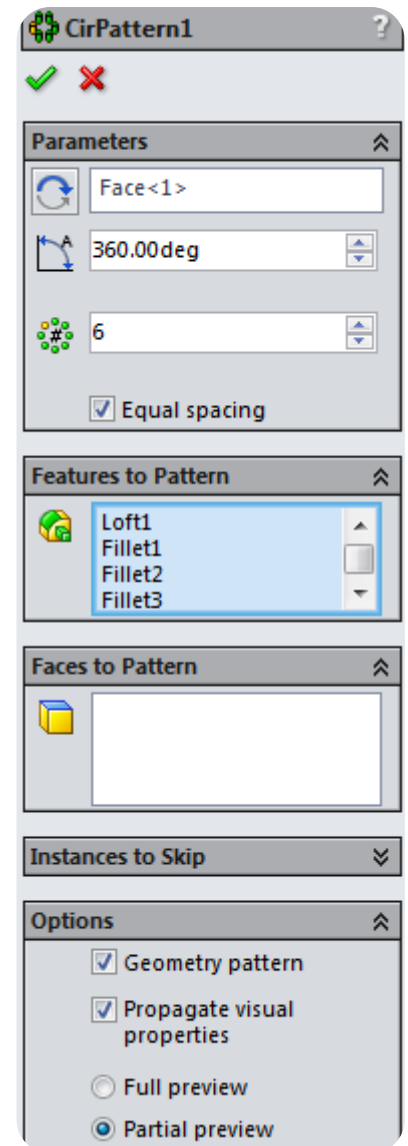
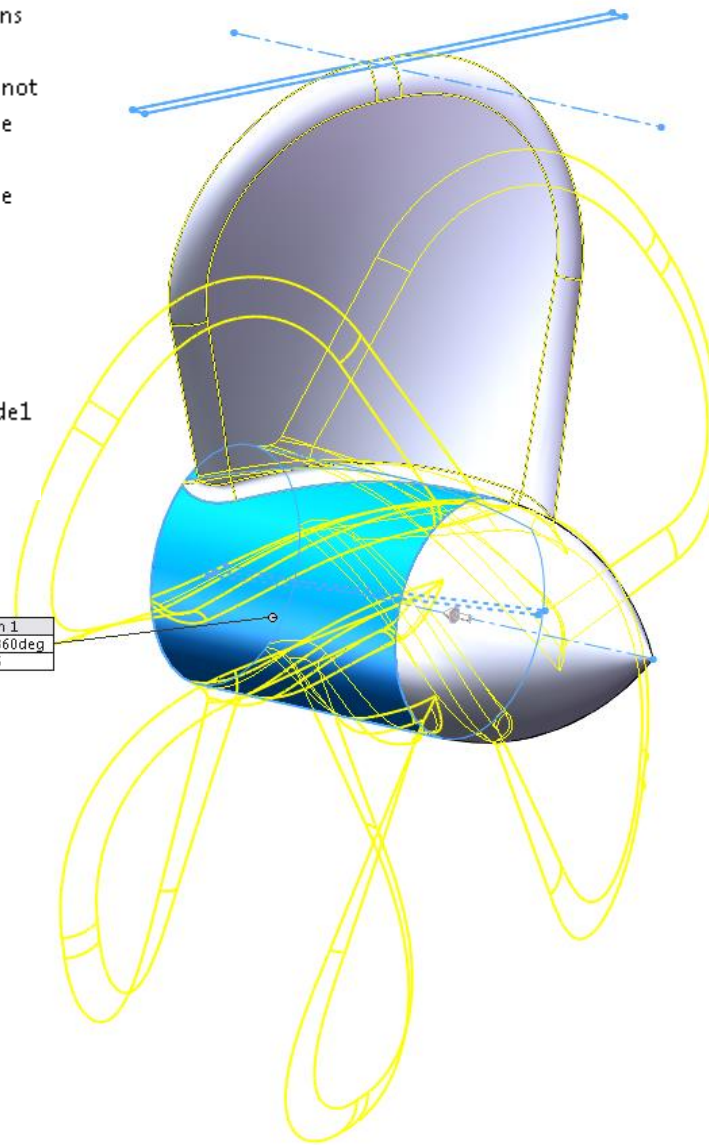
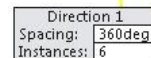
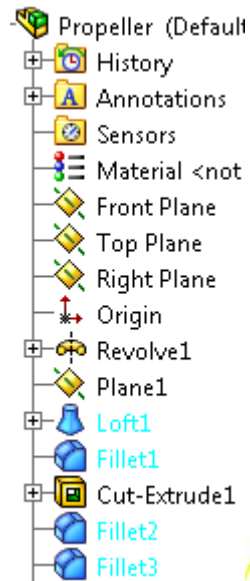
Select the **"Equal Spacing"** option.

Features to Pattern  : Loft1, Fillet1, Fillet2, Fillet3

Select the **"Geometry Pattern"** option.

Select the **"Propagate Visual Properties"** option.

Click OK 



Change the color of the propeller:

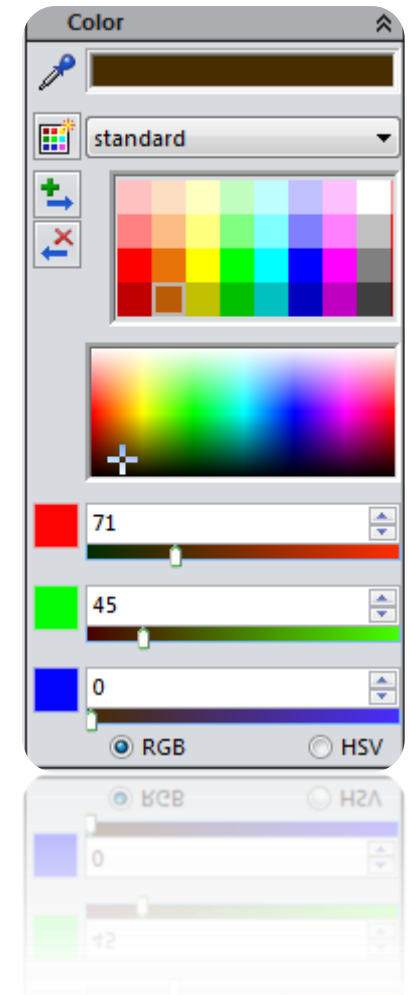
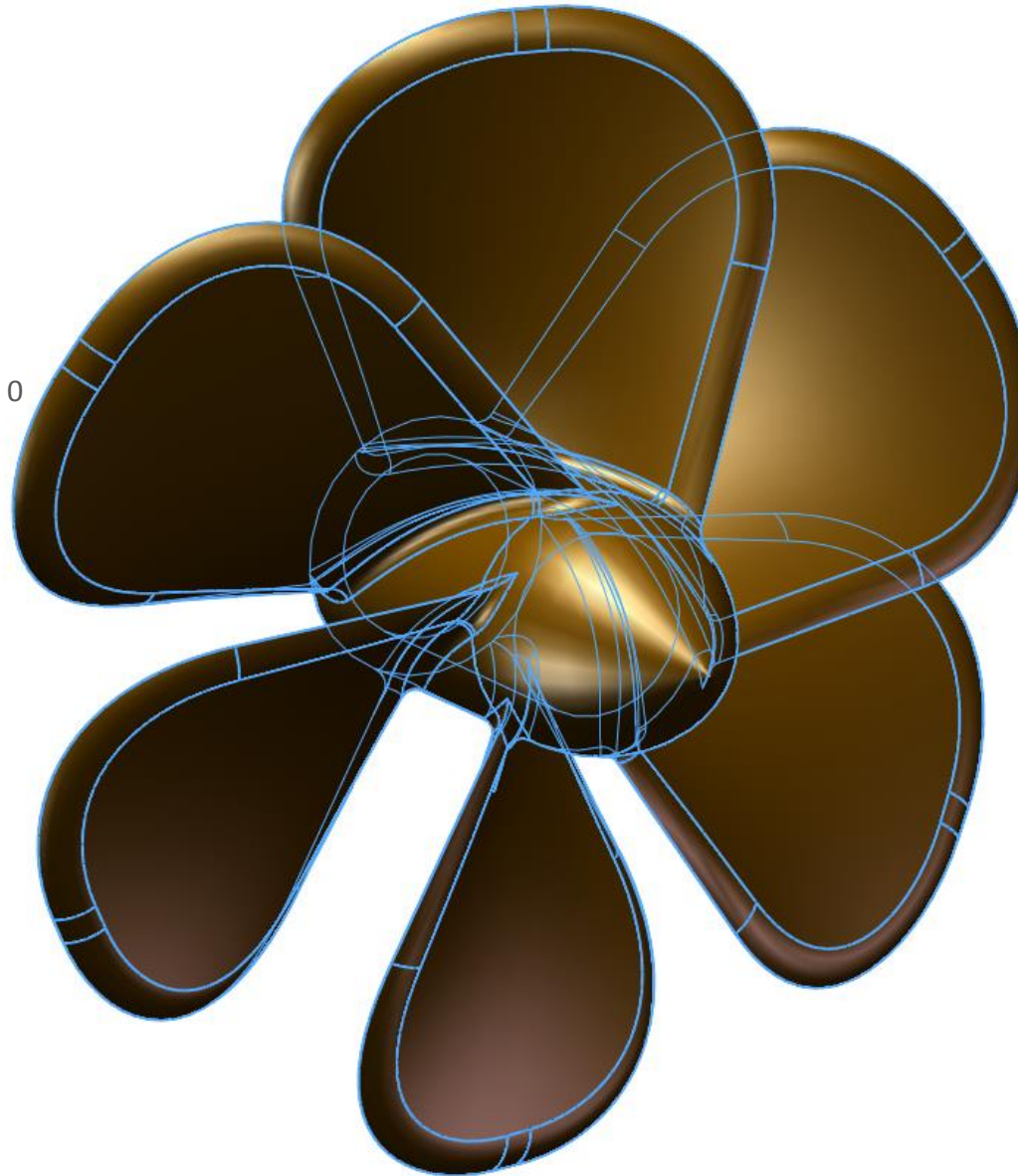
Click at the Propeller model

Click at the appearances button 

Click at the Body option 

Change the color into a custom brown
by changing the RGB numbers into 71, 45, 0

Click OK 



Hide  all planes and sketches

Save  the file with the following name: **Propeller**

Well done!

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

That's why you're going to love my [full SolidWorks Yacht course](https://www.YachtTutorial.com/course). In this tutorial package I will show you **exactly** how to model this complete yacht in SolidWorks to make you a SolidWorks Pro in the shortest possible time.

This step for step training course literally takes you by the hand and walks you step by step into becoming a SolidWorks Pro fast.

If this sounds good to you, simply click the link below to find out more:

www.YachtTutorial.com/offer.html

Stay tuned,

Jan



In the next tutorial you will learn how to model the Radar of your yacht in SolidWorks.

