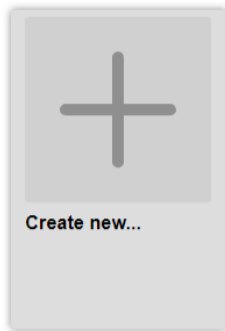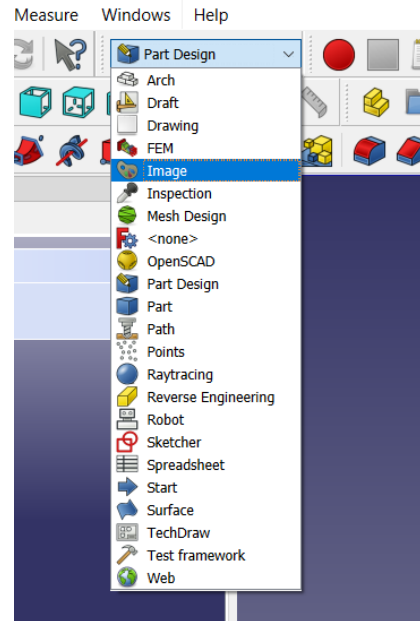# *<u>Modeling a propeller</u>*

We're going to take some time on this piece because the goal is also to teach you modeling. However, you should know that someone who is used to using the software could make this propeller in about 20 minutes.
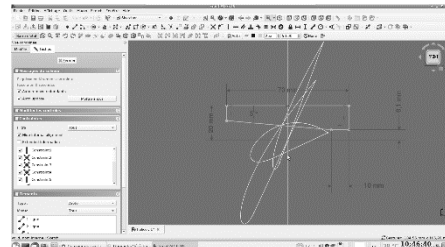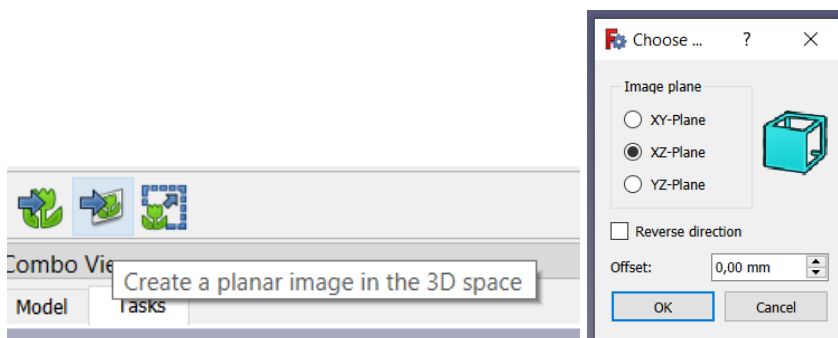
Run FreeCad and create a new document.

Then in the upper scrolldown menu select "Images".

To simplify your life (your happiness is our priority) we will use an image with all the useful measurements. You will just have to redo the drawing in FreeCAD. It is simply the outline of a propeller found on the internet.
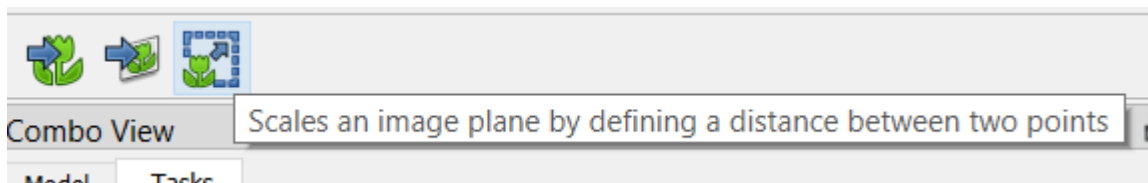
- Import this image into a XZ plane.

- In the following we will need to display the image from the front. This is done by clicking on "front" in the view selector.
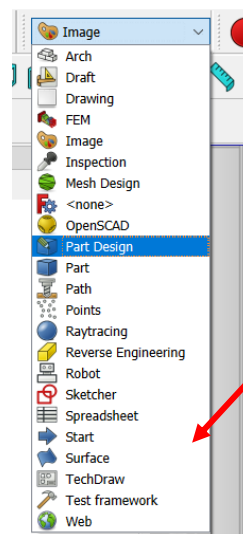
- We will scale the image so that the dimensions in the image coincide with those in FreeCAD. Click on the scale button ().
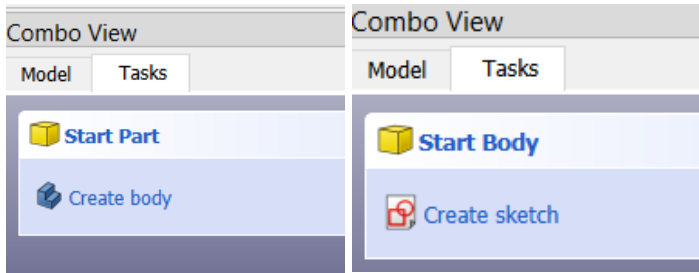

- A dialog box will appear asking you to choose a point then a second. Then the distance between those two point will be asked, enter 70mm.
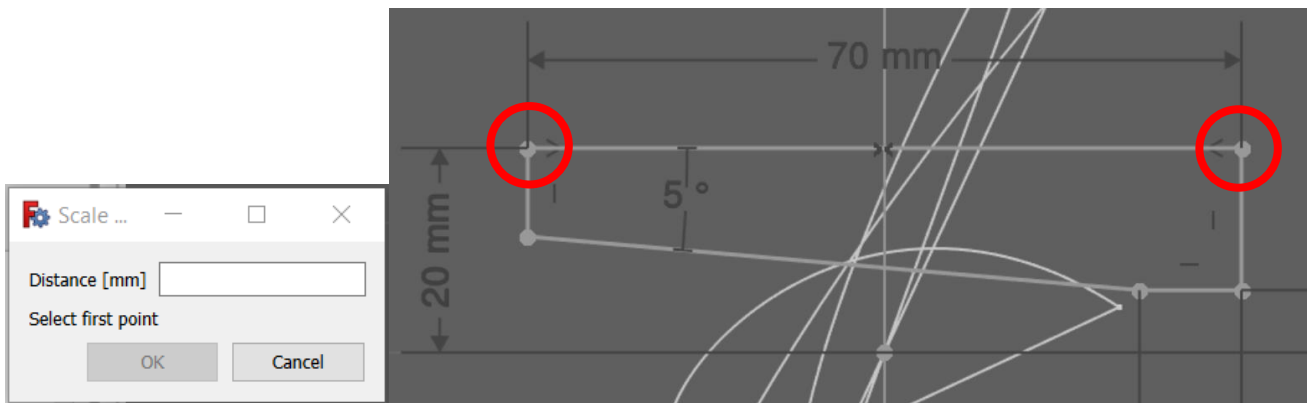
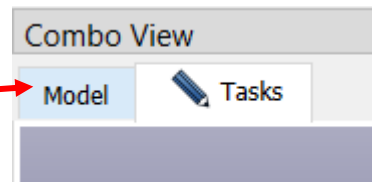- When it's done return in "Part Design".

- Create a body then a new sketch in the XZ plane.

Notice that the center of the marker on the imported image does not match the center in FreeCAD. This is not very comfortable, we will fix it.

- **WITHOUT CLOSING** the sketch:
  - Return in the model tab.

  - Go into the data tab of the "Image Plane".

  - By changing the x and z coordinates we can align the two marks. Just click on them to do so. It doesn't have to be to the nearest millimeter ;-)
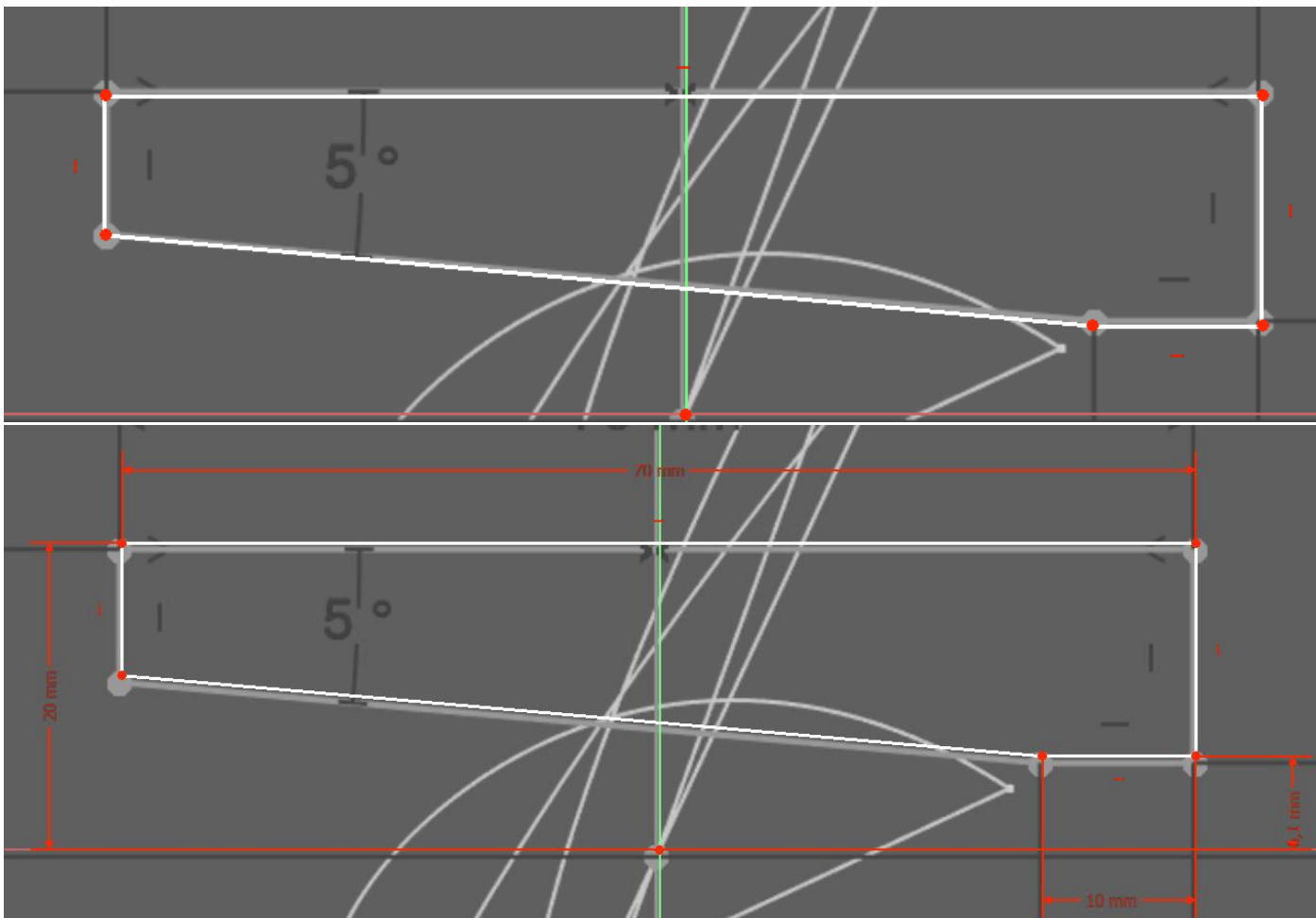
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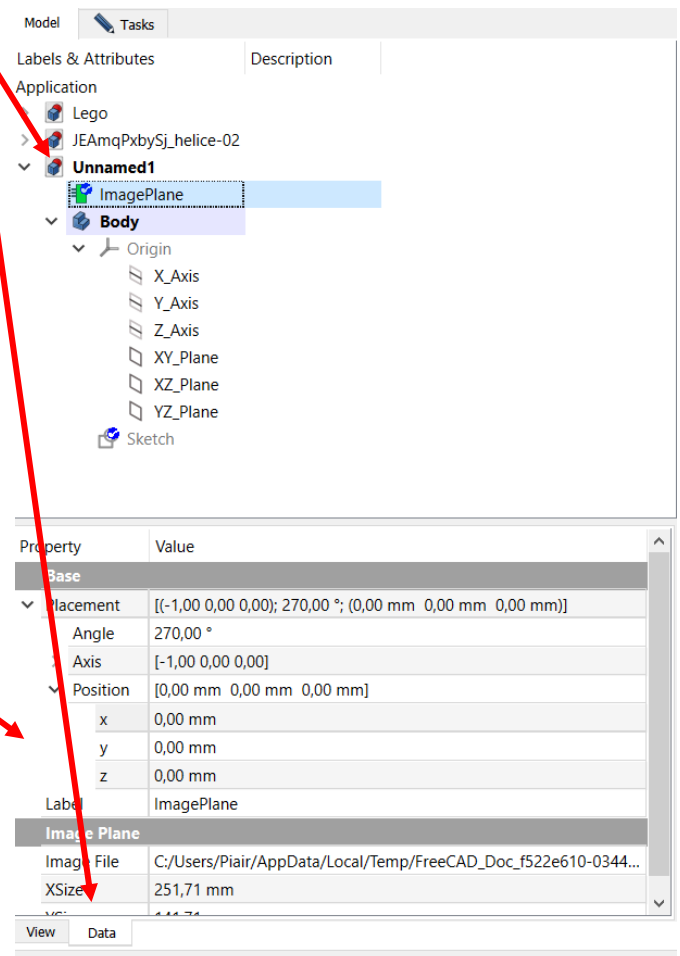
NOTICE: By applying these constraints it sometimes happens that the rectangle is off center. Using some tools is easier if the sketch is well centered.

- Right click somewhere on the screen to open the "quick menu" select "polyline".
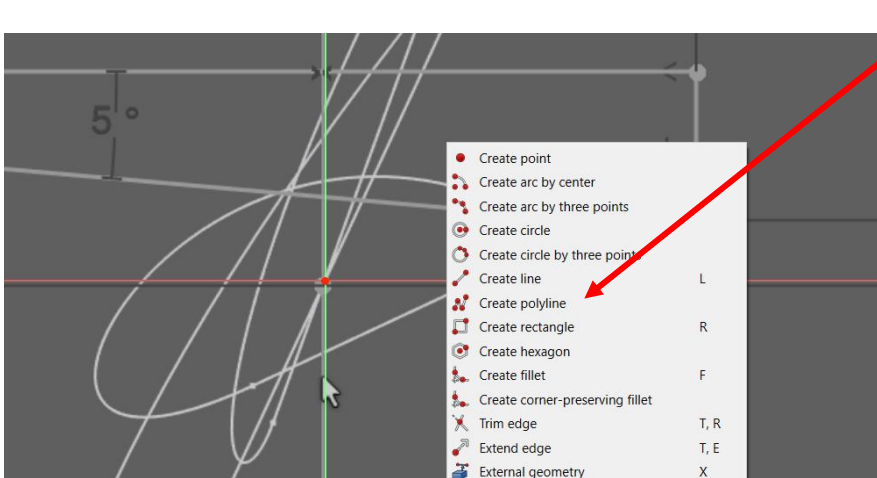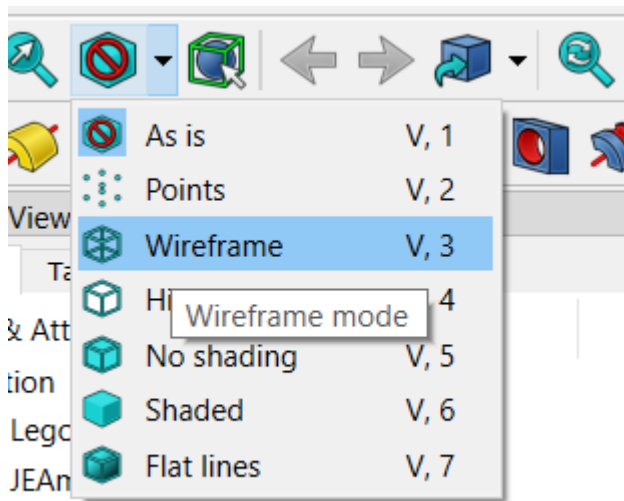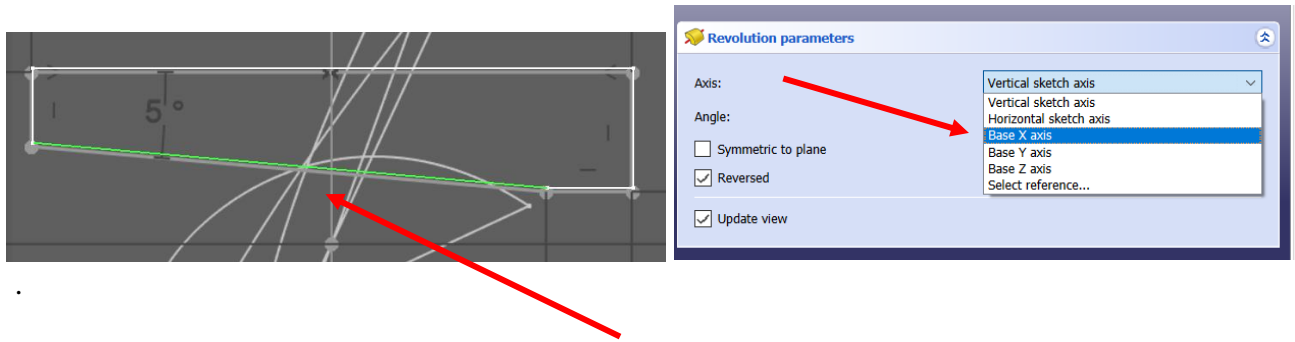

- Go over the outline of the axle as below. Give the same constraints.

(Don't forget the amplitude between the upper and bottom sides)

- Close the sketch (Return in the "tasks" tab if needed.)

- Select a segment of our figure and clik on the "revolution tool" around the X axis.

.

Now that the axis is complete, let's move on to creating the propeller blades. To do this we should go back to the design we imported at the very beginning of the project. Unfortunately it is now hidden by our 3D model.

We will switch to "wireframe" mode to correct this. We can go back to normal mode later.

- Create a new sketch in the XZ plane ().

To create our blade we will need a reference axis which does not yet exist. It is not one of the axes of the reference frame but an axis that we will create ourselves. This is called a "construction axis".
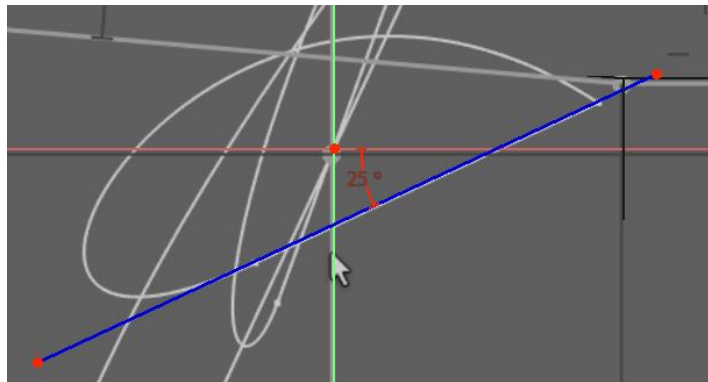
- Toggle the construction mode. The color of the drawing tools change.
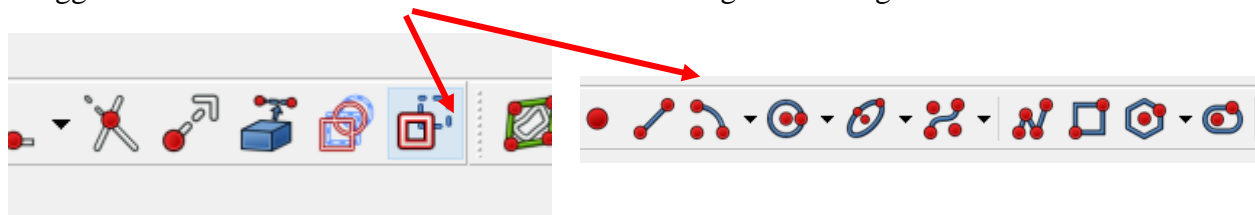
- Draw the construction line like this. If you want to move this line by using one of its points unselect the tool (right click) the drag and drop the point.
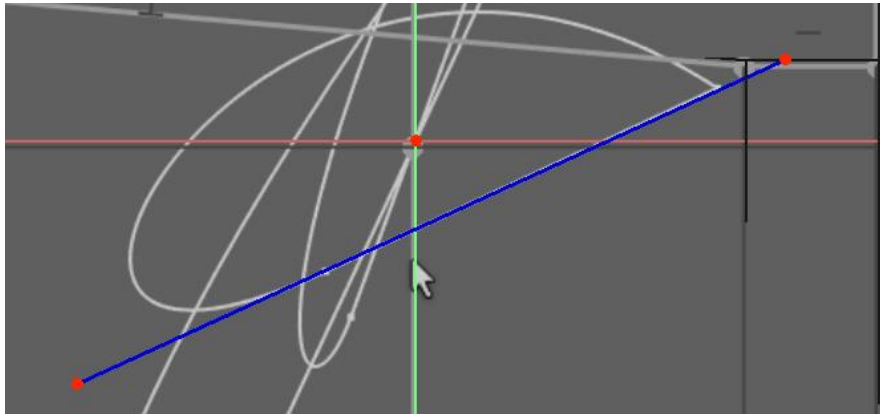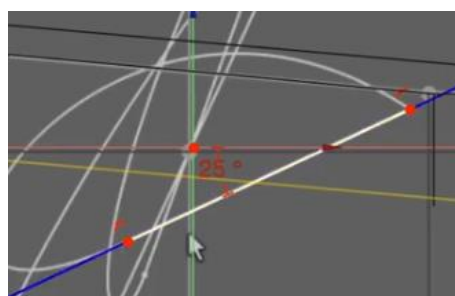

- By holding the shift button select the construction line and the x axis. Constraint an angle of 25°.
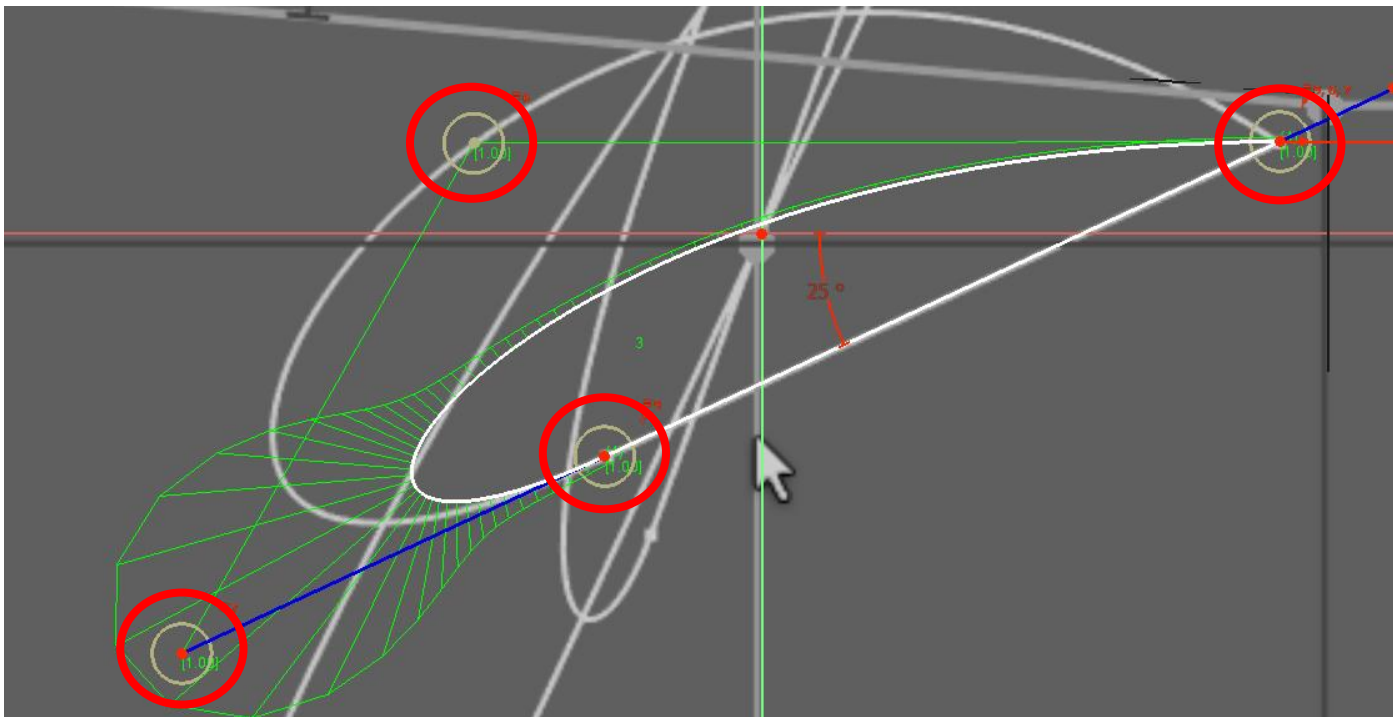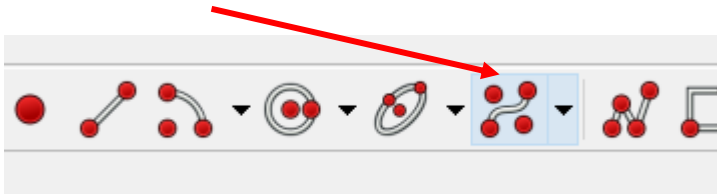

- Exit the construction mode by clicking again on the toggle construction mode icon. The drawing tool become light again.
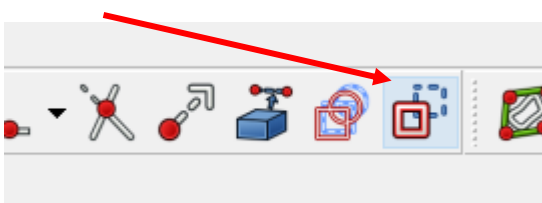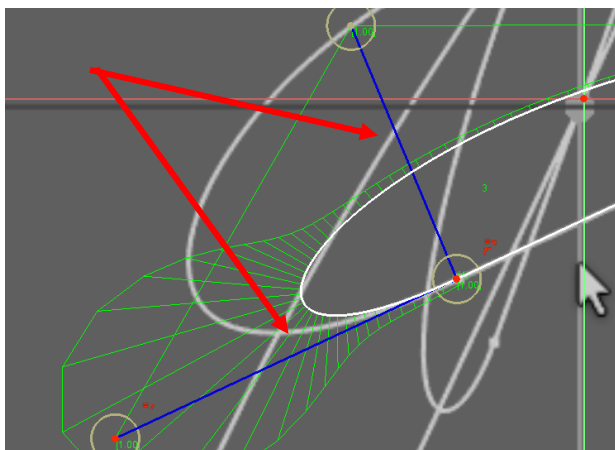

- Now that we have a construction axis we can begin to draw the sketch of the pale. Draw a first line like this:
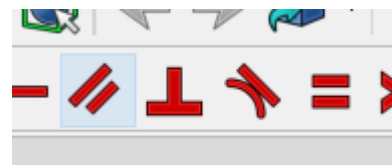
- To draw the curved part we will use the b-spline tool. Click on this icon then draw for points (see below)

- When it's done **right click to exit the drawing tool**.
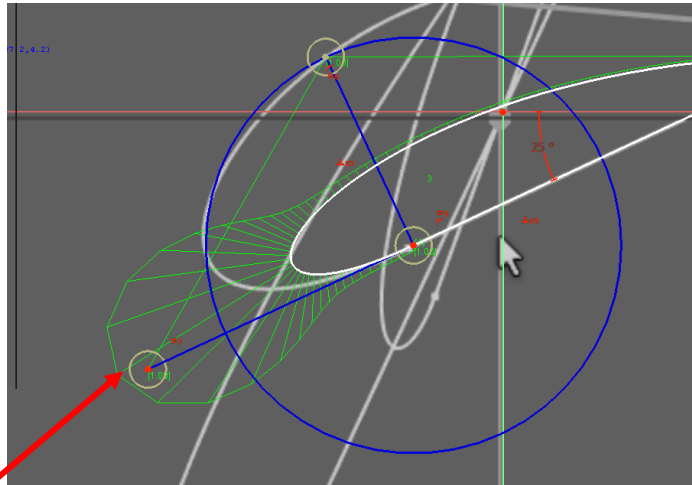- We will need one more construction line. Toggle the construction mode and draw it like this.

- Select the two construction lines and constrain them so that they become perpendicular.

Take a breath… it's nearly done ;-)

- We will draw a construction circle (check that the drawing tool is blue, it means that you are in the construction mode) between those two points:


- Select this point and the circle by holding the shift button.

- Click on the constraint "Fix a point onto an object" and attach the point on the circle.
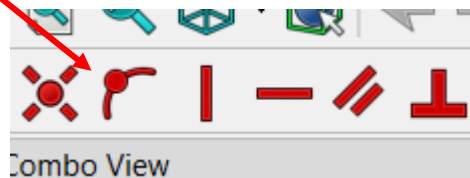
But why on earth do we have to do all this ?? We don't understand anything!

I'm sorry to put you through this :-(This is high level math! Every geometric object has characteristics (centers, radius, etc.) To constrain our helix we needed to go through this.
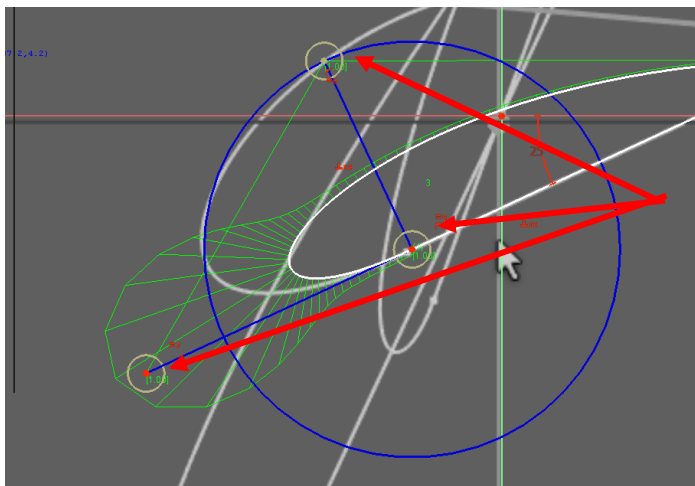
Thanks to the constraints we have defined we now have three points that we can move. By doing this we can make our blade take the shape that was on our reference drawing. The goal was this... If you feel the need to understand why, come back to me in a few years. I would be proud to know that you took a degree in mathematics just for that :-)

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- Move the points to have something like this. It don't have to be perfect, something like this would be nice.
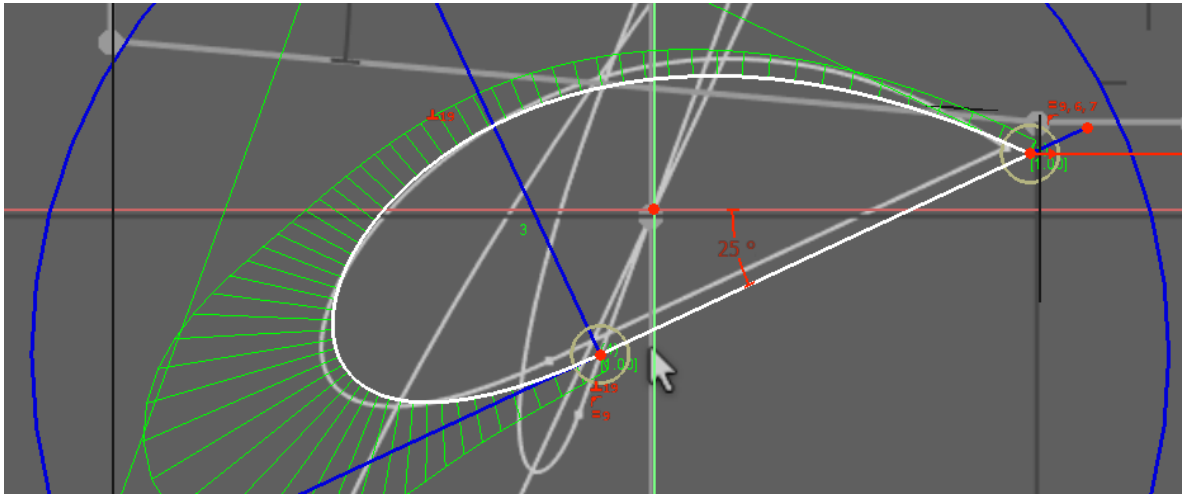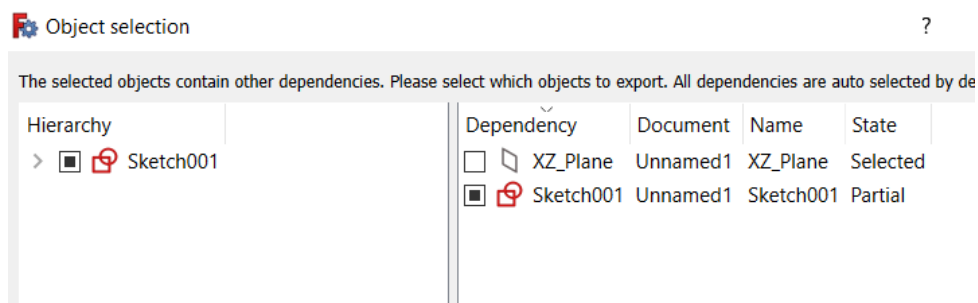

- Close the sketch.

    We want three blades. Don't worry, we won't have to go through all this again for each blade. We will copy our sketch and modify what we need to get the following blades.

- In the model panel right click on our sketch and select the copy tool.
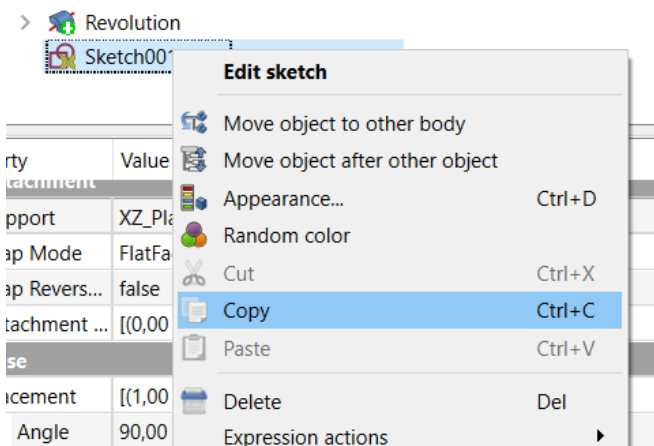
- A new window pop up. Unselect the XZ_plane, we just need the sketch not the attached plane. Click OK.

- On the keyboard push CTRL + V to paste our sketch.
- In the model panel, drag and drop the new sketch on the body.
- Now double click on this pasted sketch to edit it. We will change some things to create our second blade.
- Select the 25° constraint (it becomes green) and on the keyboard push "del".
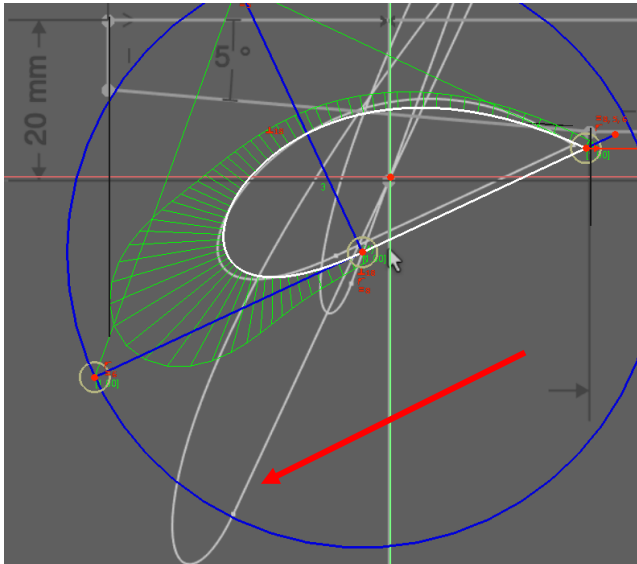
Our second sketch have to be here.

By holding shift select the construction line and the x axis to constraint an angle of 65° then play with de constraint points to superpose the reference drawing and our sketch.
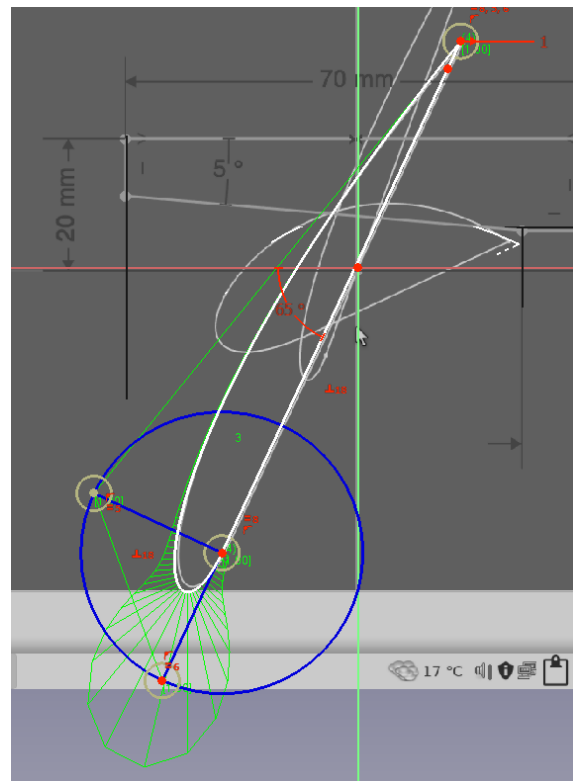

- When it's done close the sketch and copy paste it into the body like we did before.
- Edit it.
- Suppress the angle constraint and create a new one of 110°.
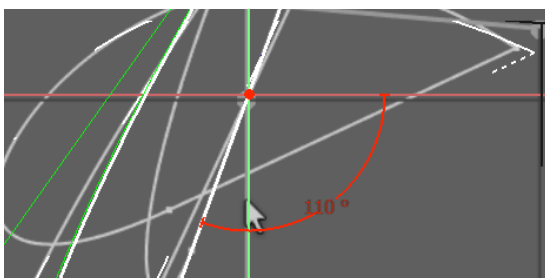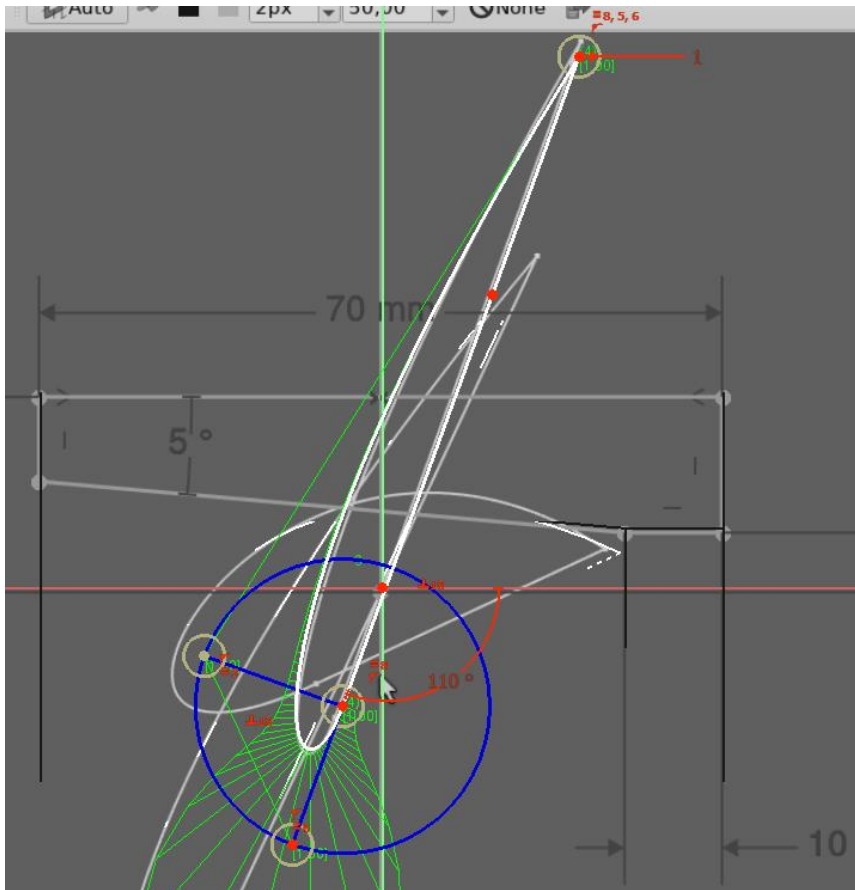
By moving the construction points try to have something like this then close the sketch.

- Copy the last sketch again, move it into the body and without deleting the angle constraint try to have something like this at the edge of our former sketch.
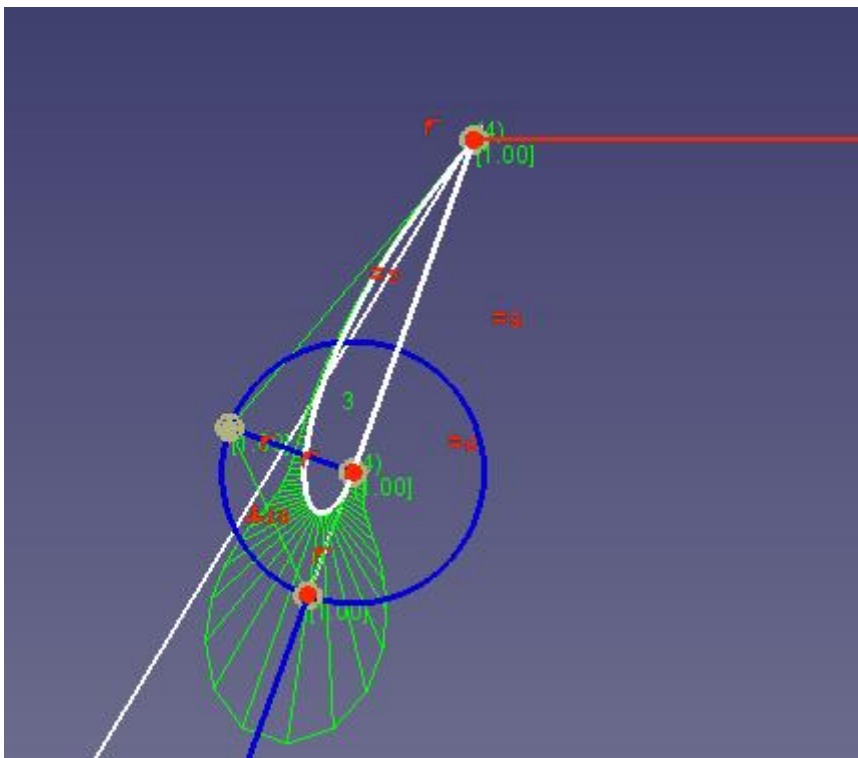
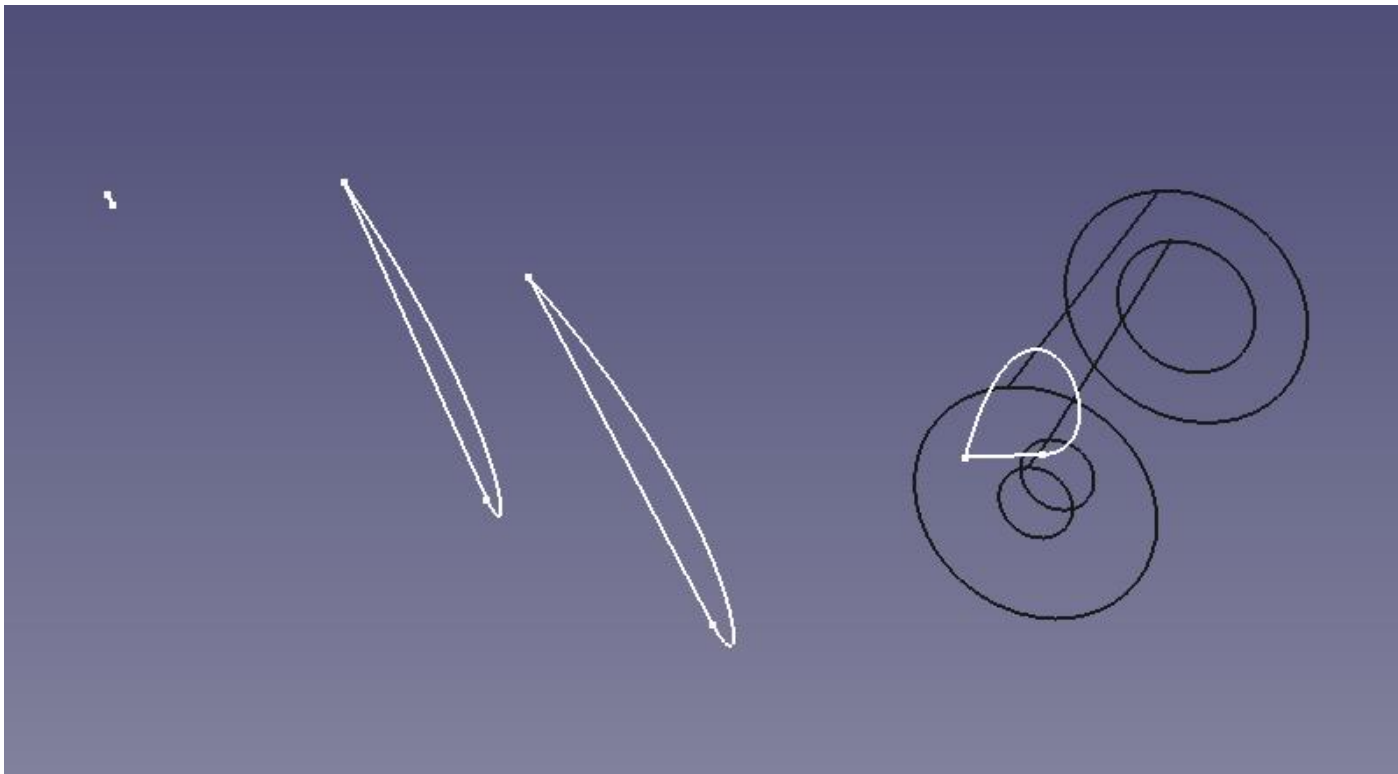This part must be very small because it is the tip of our blade.

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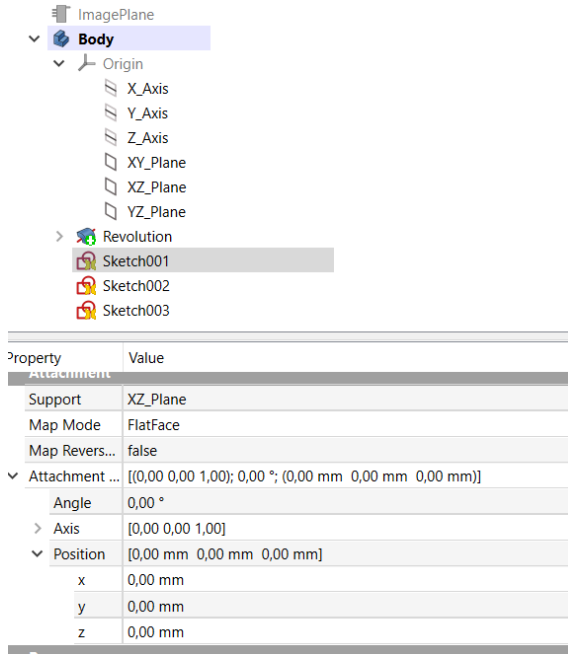
Select our first sketch and in the "attachment" property give a z position of -15 mm.

For the next one give a z position of -85mm.

For the third one a z position of -110mm.
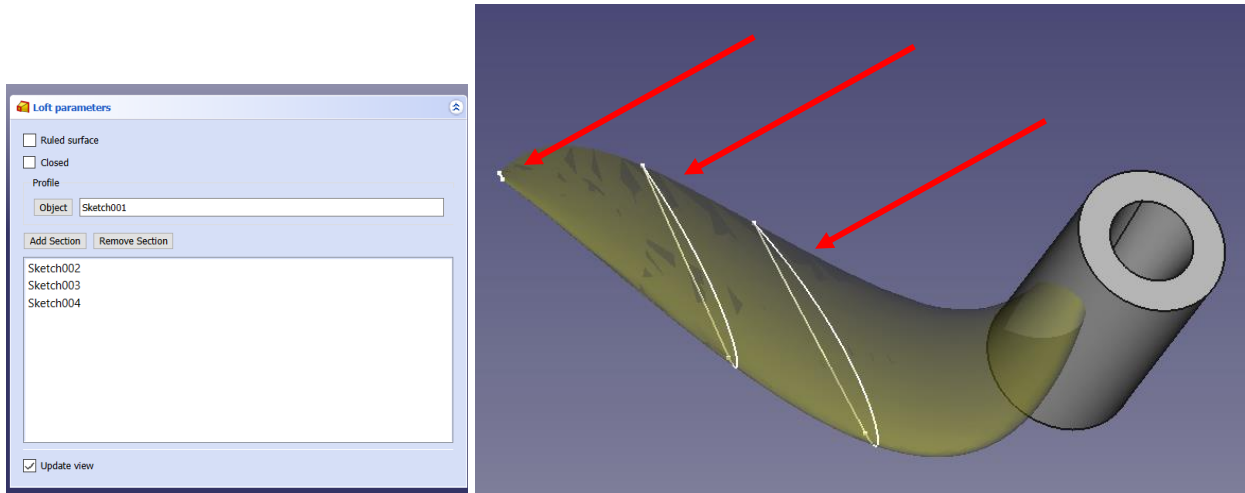
For the last one a z position of -150mm.

See… that's why we worked so hard! Now… let's create our first blade.

• Change the view mode to "as is" then select our first sketch and click on the loft tool.
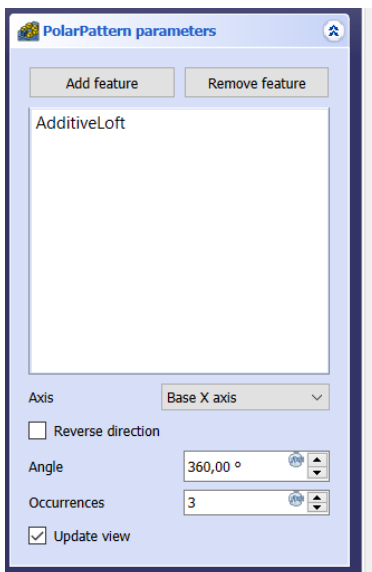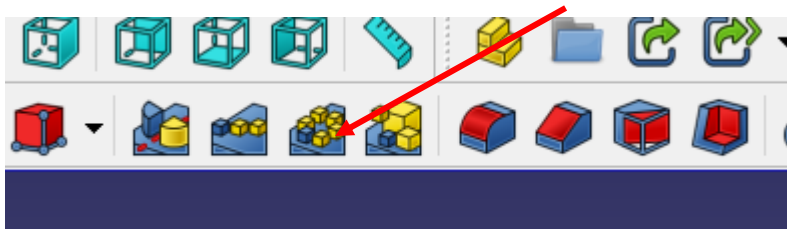
- A new panel will open. Click on add section then add the three sketch we created in the main screen.

- Click the "OK" button.

  Job done! Our first blade is modeled!

- Select the additiveloft in the model panel and select the "polar pattern" feature.

Make three occurrences around the base x axis and click ok.

We have a real propeller now!