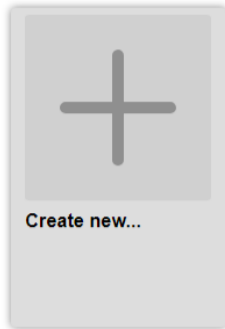# *Modeling the cunning tower*

This is an average level workshop. You will learn some advanced features of Freecad like how to attach a plane surface on a curved surface. 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 tower in about 15 minutes.
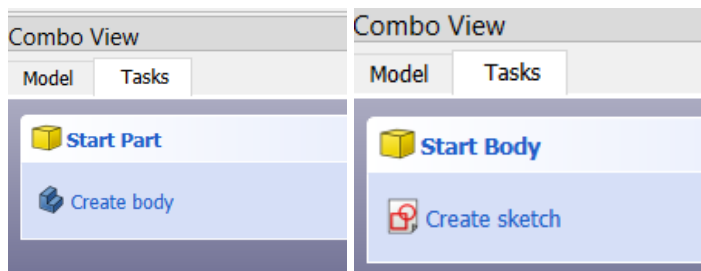

Run FreeCad and create a new document.

Save it now to avoid any loss of data.

- When it's done go in "Part Design".
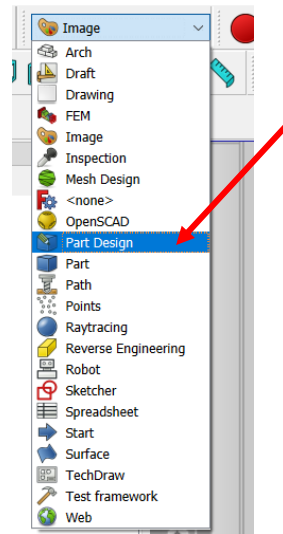
- Create a body then a new sketch in the XZ plane.

NOTICE: A bottle of soda of 1,5L (a cola bottle for example) have a diameter of 88 mm. Its circumference is therefore 276 mm. We don't need all this surface, a surface of 50 mm will be more than enough. With a little mathematics we can easily find that an angle of 65° will intercept a 50 mm string. We show you...
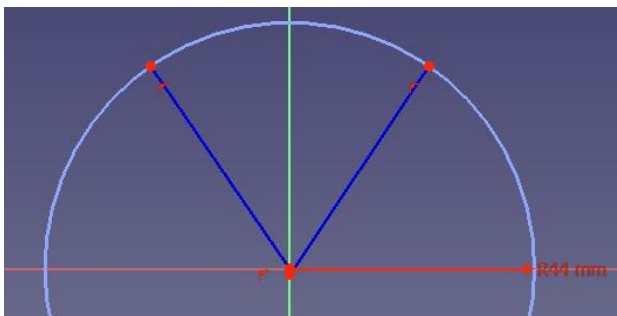
- Click on the construction mode icon. This part will not appear in our model, we just need it to construct the basement of our tower.
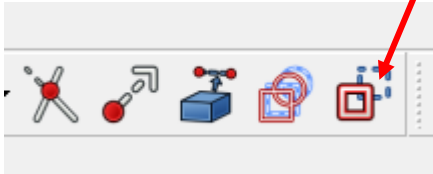

- Draw a circle centered on the center of the axis system. Right click to exit the tool.
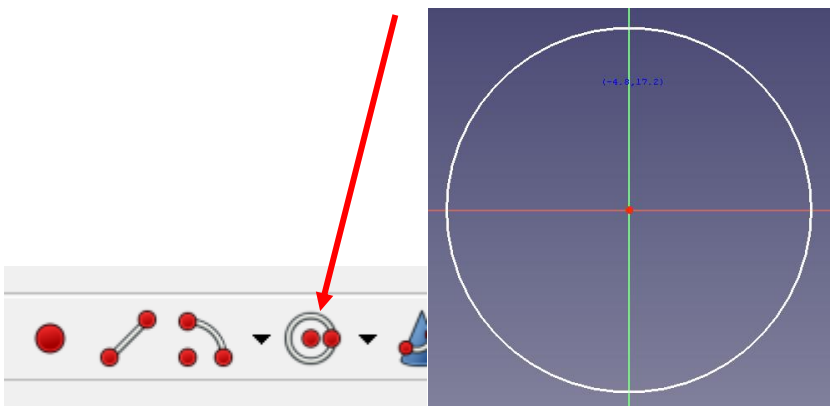
- Constraint the circle with a radius of 44mm.
- Draw to lines like this, starting from the center of the axis (FreeCad indicate it by showing the ✖ icon) and ending on our circles (ⲅ icon appear)

Be sure that everything was drawed in construction mode, if it's not you can select each of your lines afterwards and click on the construction mode icon then.
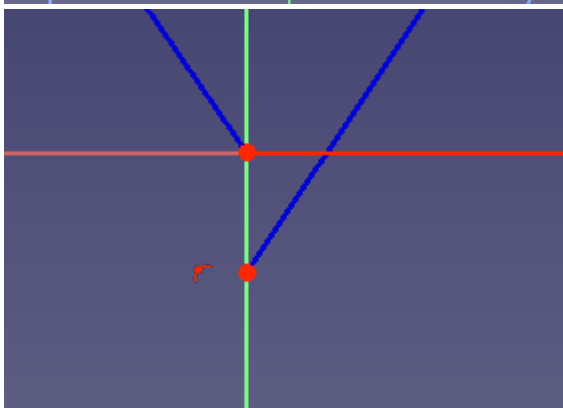

If you notice that your starting points are not aligned with the center of the marker, don't worry, just select the offset point and the center then the coincidence constraint. ✖

If your upper points doesn't fit on the circle you can select the point, then the circle and finally the "fix onto an object" constraint. ⲅ

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What's really practical with Freecad is that you can nearly do anything at anytime!

- To be sure our two points on the circle are at the same distance from de Z axis we will constraint them. Holding the Shift key all the way, select the two points and THEN the z axis. Then click on the symmetry constraint icon.
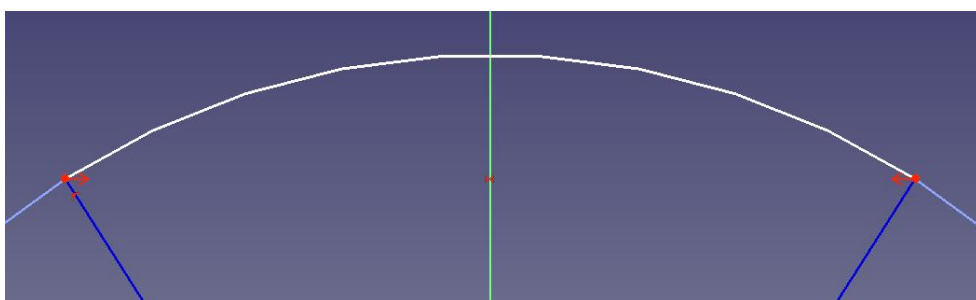
- Now, as we seen before, constraint the angle between the two lines to 65°.
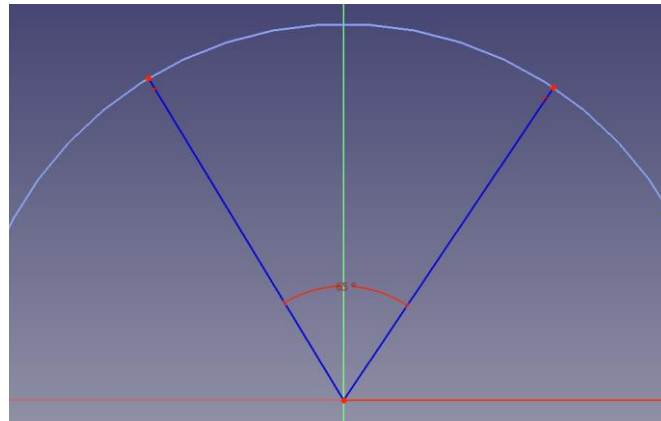
- Exit the construction mode.
- We will now draw an arc on our circle by using the arc tool. You will have to first select the center of the circle and THEN each point on our circle.
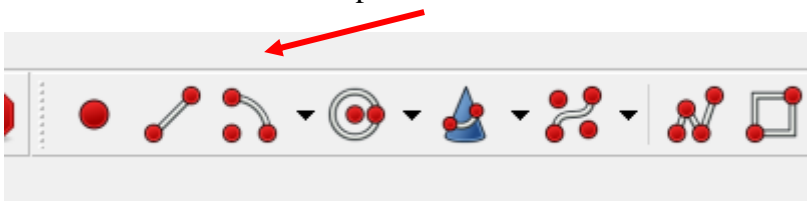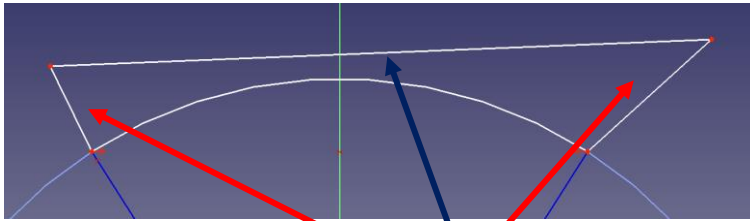
NOTICE: Be sure that the coincidence icon appear when you select your points.

- We will now complete our sketch with the multiline tool ( ) When you reached the last point exit the tool by right clicking.
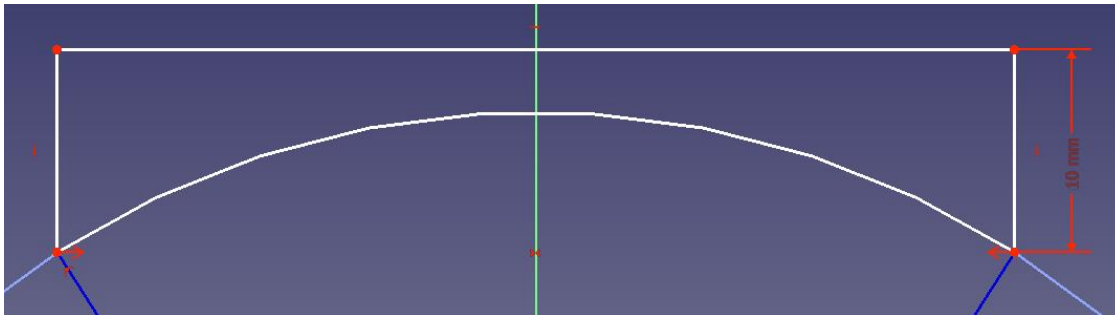
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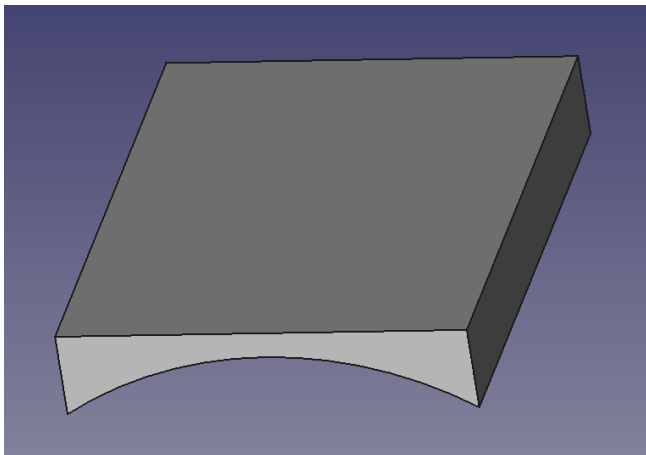
- Try to draw something like this.

Use a vertical constraint ▮ here.

Use a horizontal ▬ constraint here.

Give a height ⌶ of 10 mm to one of the vertical lines (no need to do it to both of them because we previously defined a symmetry constraint).

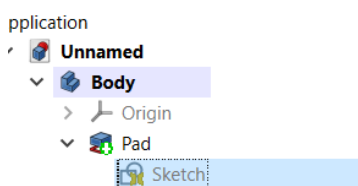- This is it! Now close the sketch and pad it 🗇 (a height of 60 mm will be enough)

Do you see this ? It's awfull!!!

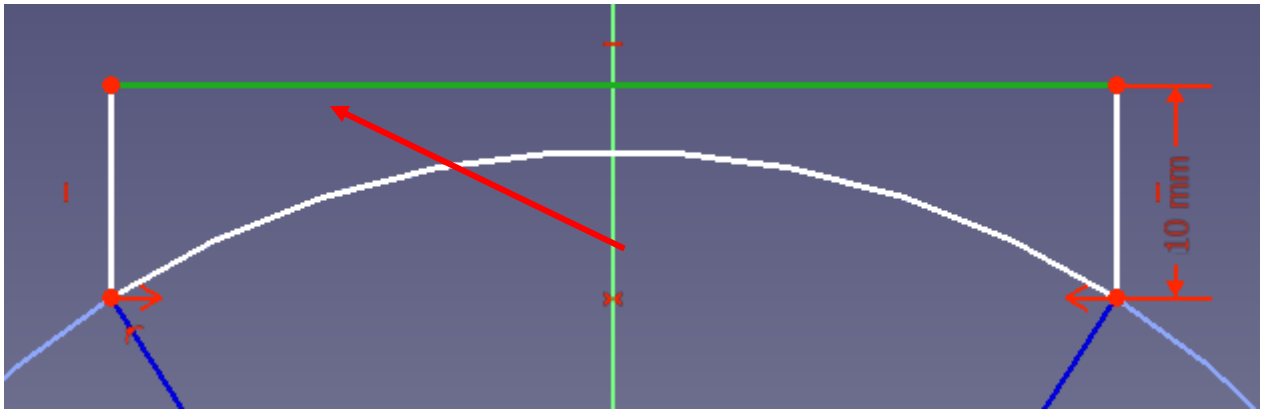We cannot afford to put such flat surfaces on our submarine. First because we are STEM!

Secondly because it is not really aerodynamic...

FreeCAD allow us to do anything at anytime, do you remember ? Let's correct this bad mistake.
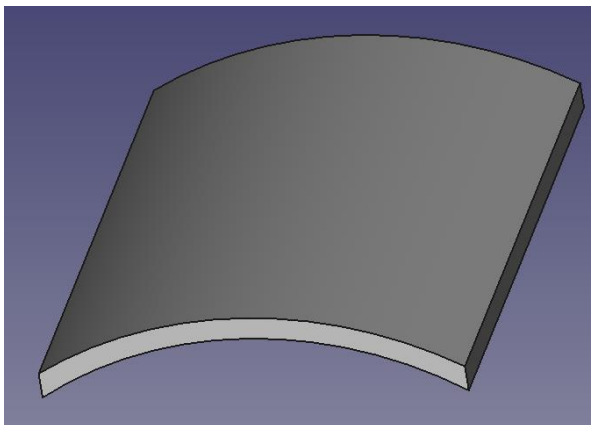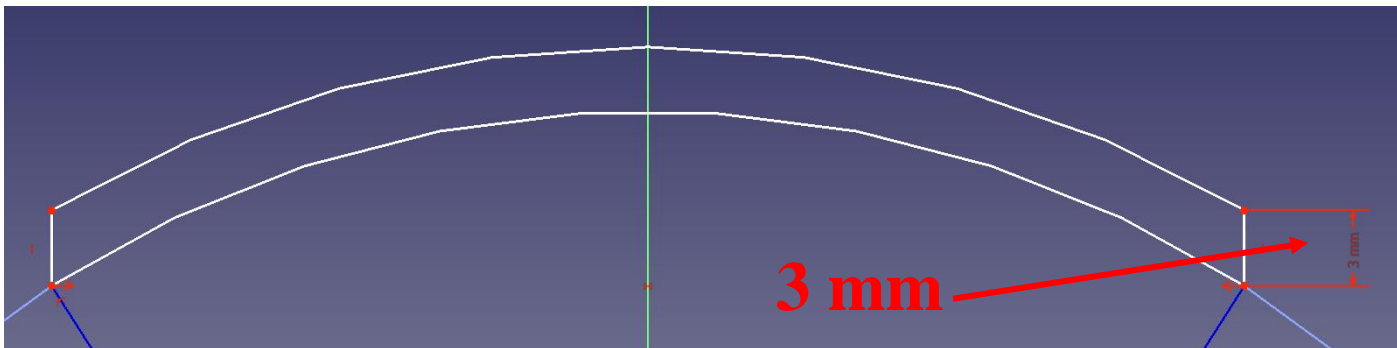

In the model panel double click on the sketch to edit it.

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- Select the upper flat line and delete it.
- Now let's go for a little exercice. You followed all the previous instructions carefully, isn't ? It's okay then, try to draw this:


The magic of FreeCAD works... We modified the sketch, the software modified the protrusion accordingly.

It will now perfectly fit to the bottle. Wan't to use another kind of bottle? No problem, edit the previous sketch and change de radius of the circle according to the radius of your bottle. It's pretty easy!

NOTICE: You will now use one of the more important tricks in 3D modeling: the use of a personalized reference plan. Indeed we can directly draw a plane surface on a curved one.

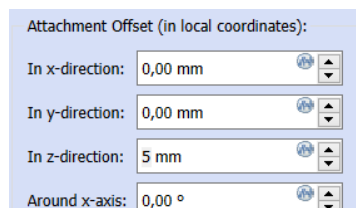- First select the upper face of our curved solid.
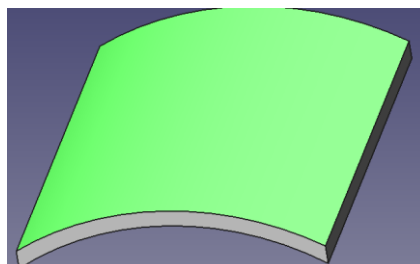
- Now click on the "datum plane" icon.
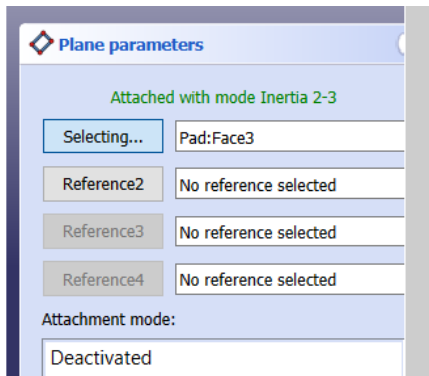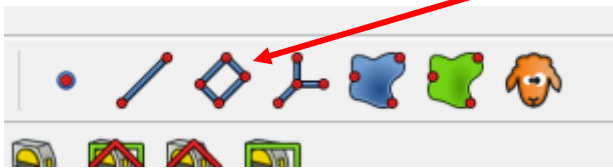
Be sure that the attached mode is "selecting" your pad.

NOTICE: At this point, if we let our plane go through our model we will not be able to see what we draw on it. So we will first give an offset in the z-direction of 5mm.


- Click ok to leave the datum plane tool.
- Select the datumplane in the model panel. It becomes green.
- Click to draw a new sketch (because we selected the datum plan before FreeCAD know we want to work with it as reference).

- We will now do a little exercise with the "slot" tool.
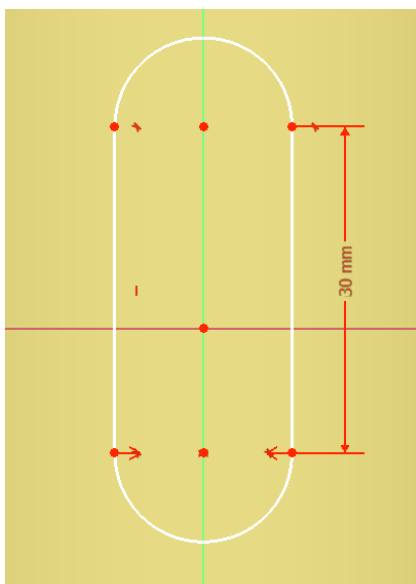
By using the predefined constraints of the slot tool try to draw something like this.

TRICK 1: Begin by defing a symmetry constraint with the y axis.
TRICK 2: Then give a length of 30 mm to the line as shown.

It's done ? Close the tool!
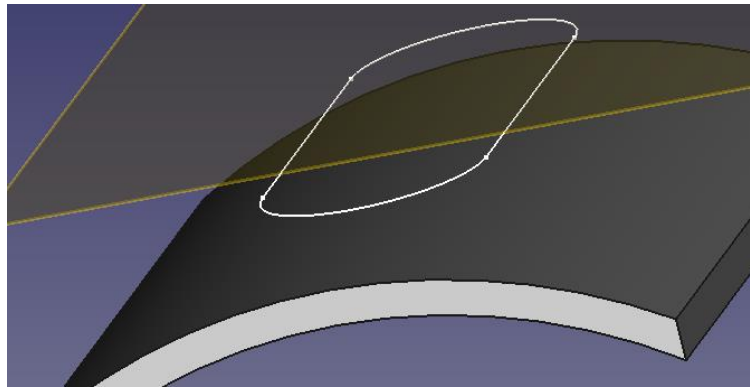
We could try to immediately use the protrusion tool but it will not work. FreeCAD don't allow multiple solid parts in the same document. So we first want to "fix" this sketch on the upper face of our solid base.

- Select the upper face of the base.
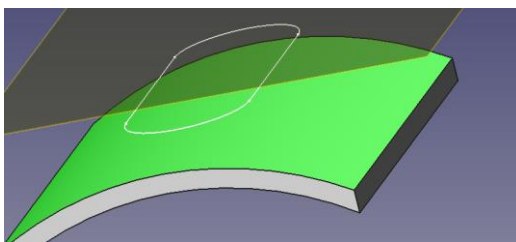
- Click on the "support of a sketch" tool and ask to attach your sketch into the contextual menu. Another menu will appear don't worry, simply click ok again.
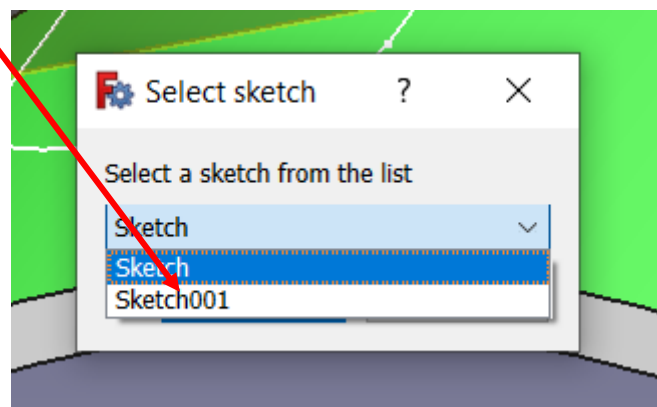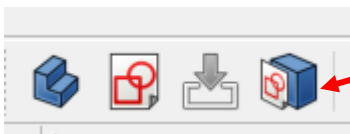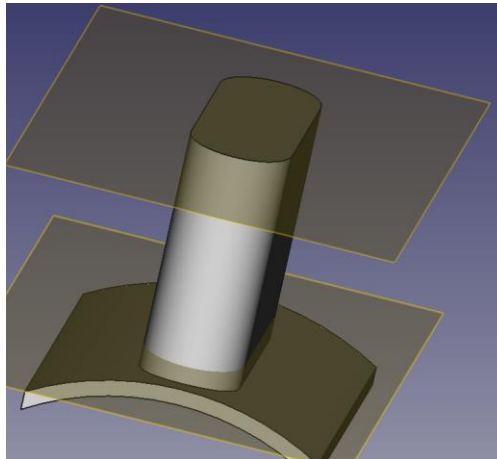

OH MY GOD!!! Our sketch disappeared!!

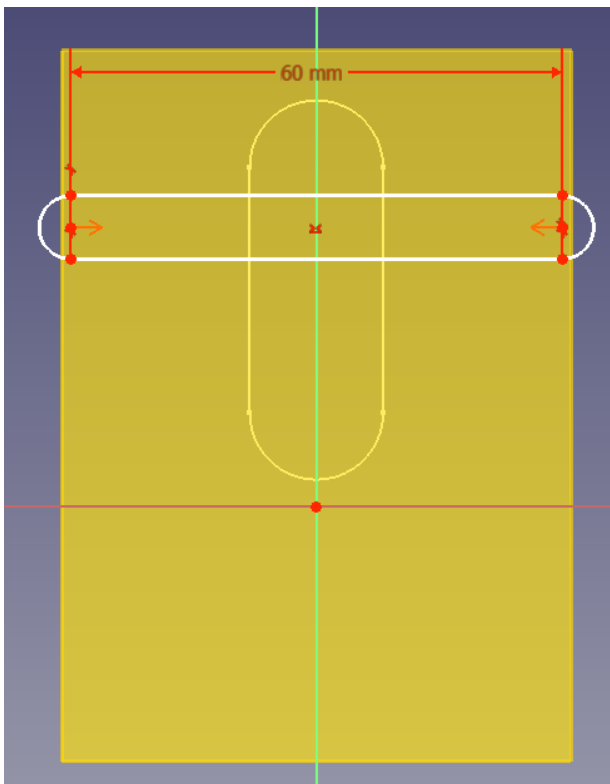No it doesn't… It's simply merged with our solid. Select it into the model panel, use the pad tool (height of 40mm)

Job's done!

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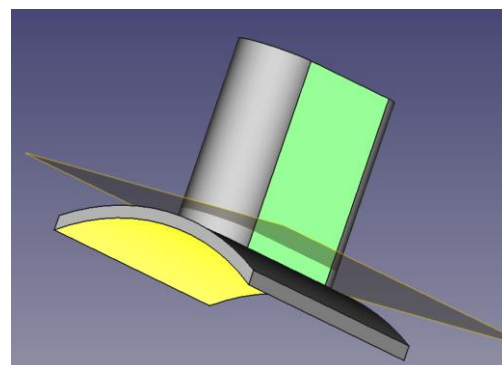
- We will now attach a new datum plan on the upper side of our tower.

- Select it and draw a new sketch on it.

You know how it works now. By using the slot tool and some constraints (horizontal distance and symmetry) try to reproduce this sketch then click ok to leave the sketch.

You can pad it (Height of 3mm) directly because the datum plan fit perfectly with a plane surface.
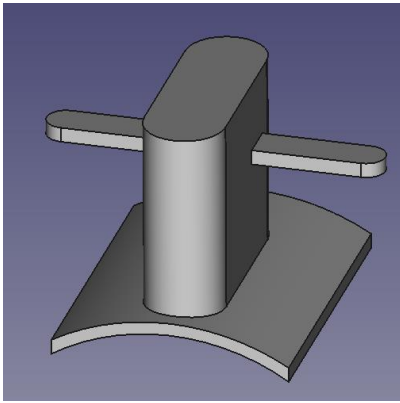
Why using a datum plan then? Because by doing this we can change the height of our part!

In the attachment property of your datum plane 2 you can easily modify the z position to set the part at the height you wish.

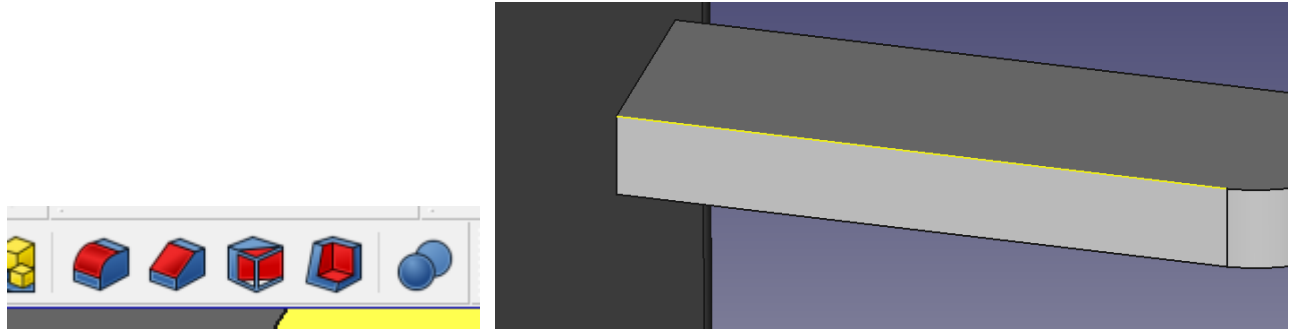Because those "wings" are related to the datum plan they will follow!

Let's have a look to our model… If you want to hide the datum planes like here, simply select them in the model panel then push the space bar.

Our model is already nice but contains too many straight lines. We'll have to round it up a bit...

- Select a straight line and then click on the fillet tool.

By doing this on all straight lines we finally get this…