

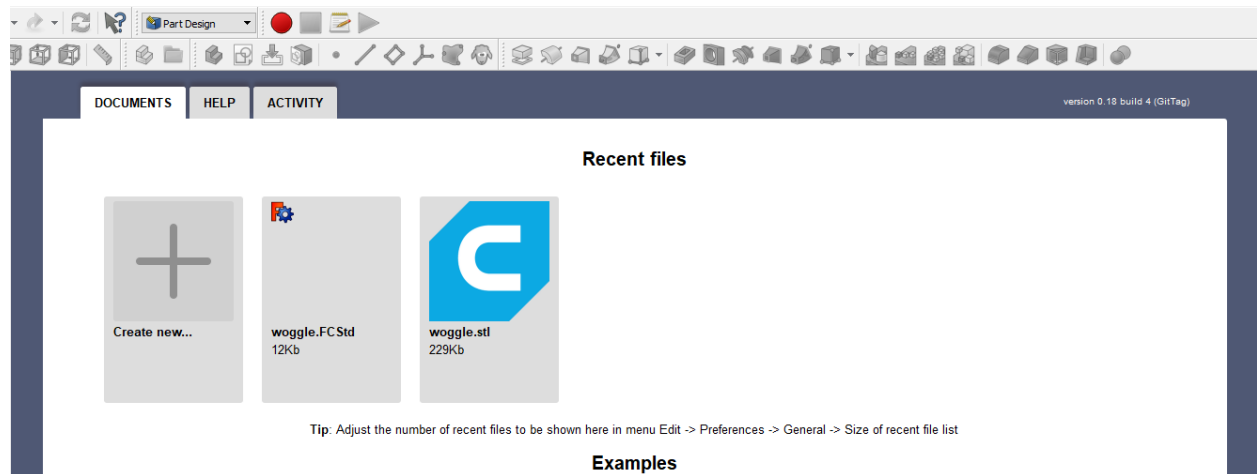
# FreeCAD Tutorial

Recommended video tutorial:

<https://www.youtube.com/watch?v=sxnij3CkklU>

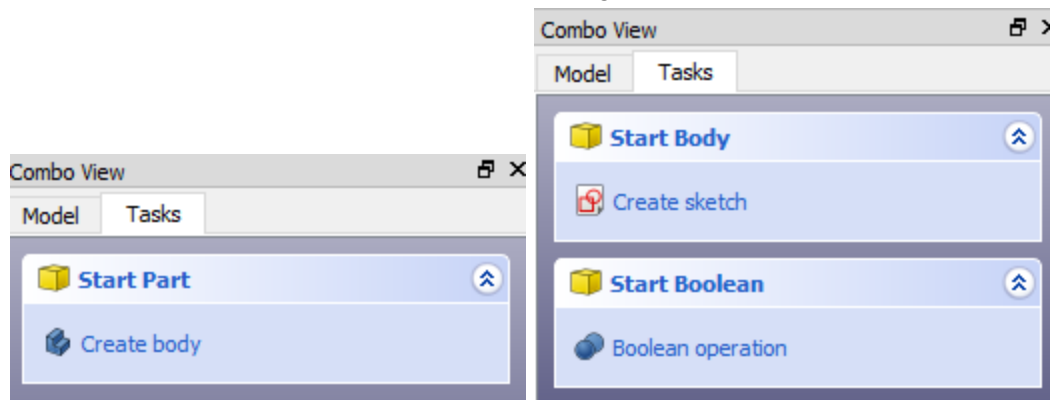
## Tutorial Cube

To create a part you must first create a new file. To do this select “Part Design” from the drop down box by the red record button (you do not need to know how to use the record button for this tutorial). Then press create new... to make a new file.

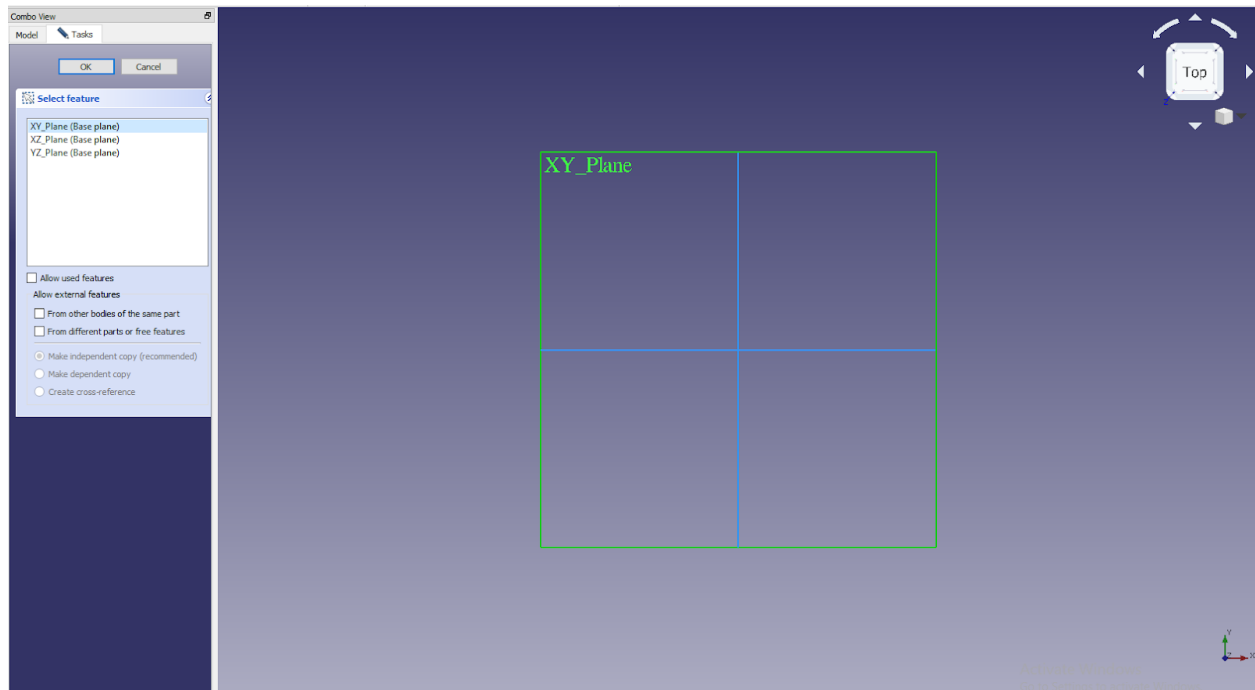


To create a 3D object you first have to create a 2D shape to represent it, then you can manipulate the 3D version to the desired shape.

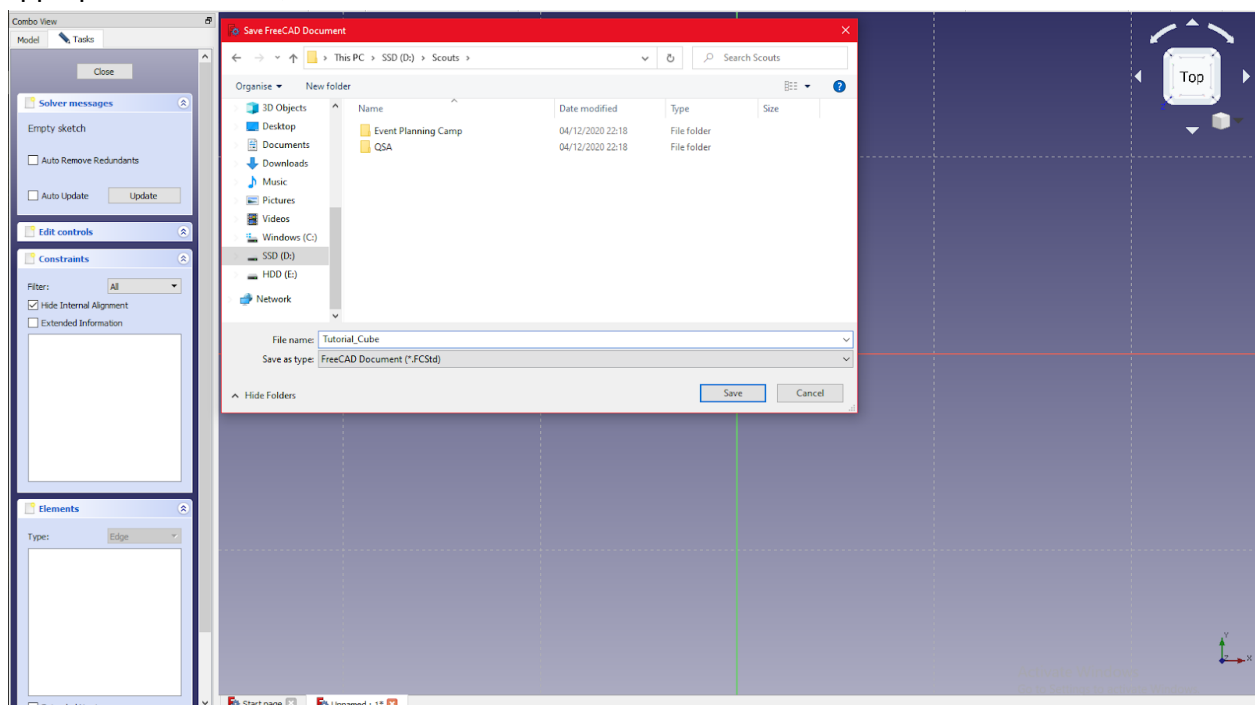
To create a 2D “sketch” you may go from Combo View -> Tasks -> Create body -> Create sketch. If the combo view is not present it is likely that you did not select “Part Design”, the buttons are also on the top toolbar with corresponding icons to the ones in the menu below.



To draw your shape you will have to select which plane you intend to draw it on, this tells the program which way is down relative to your design. This is not particularly important in your design and any plane may be selected. However, I would recommend using the XY\_Plane as this is usually the easiest orientation to use when your design is turned into a physical object.

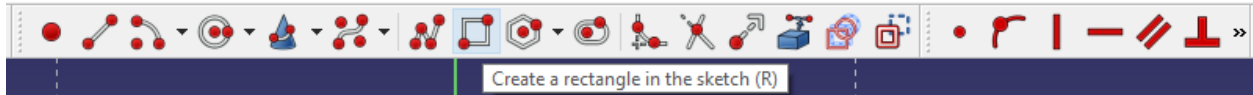


This is a good time to save the file. If you don't have one already you may wish to create a folder for work done with Scouts (or even just for this camp!) It is advised to save the file with an appropriate file name, such as "cube" or "tutorial".

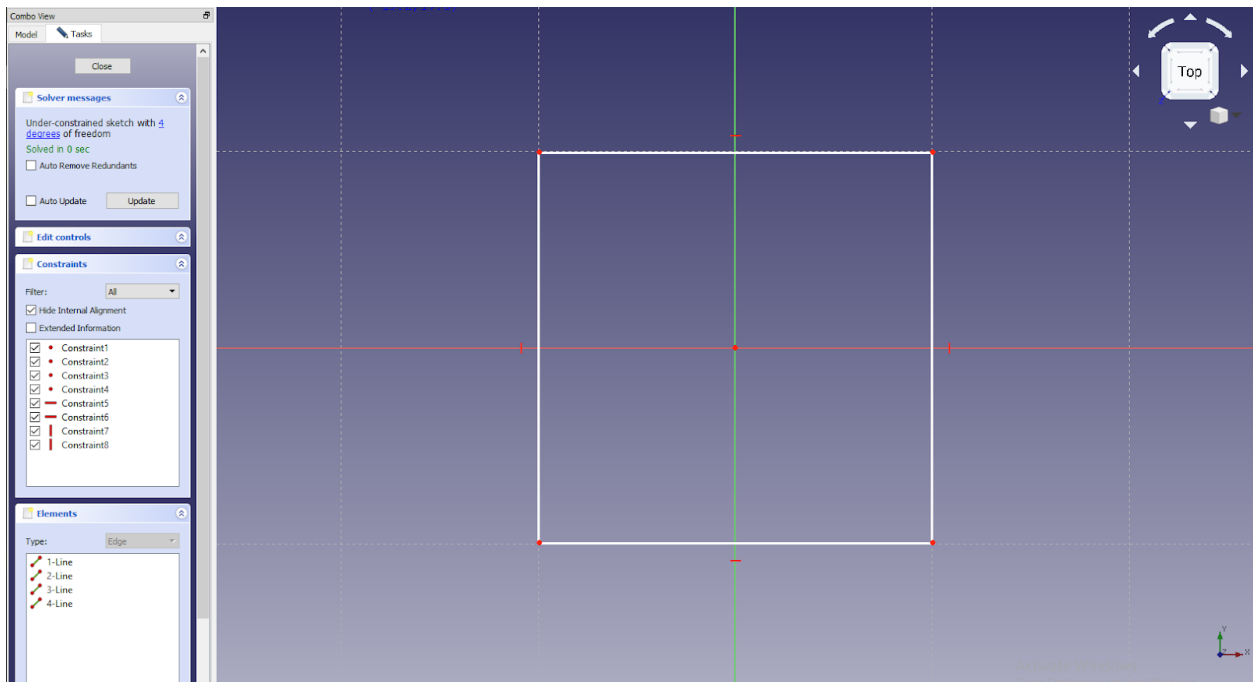


The first step in designing the cube is to create the 2D shape that will later be converted to 3D. For more complex shapes it isn't always obvious what this 2D shape should be, for a cube a square would be an obvious choice.

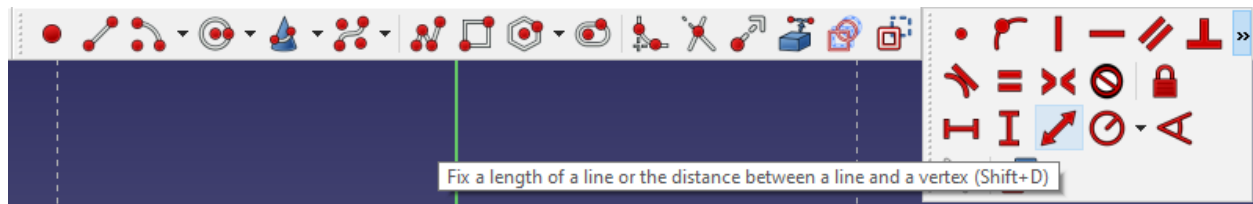
To create a square we will use the “Create a rectangle button”.



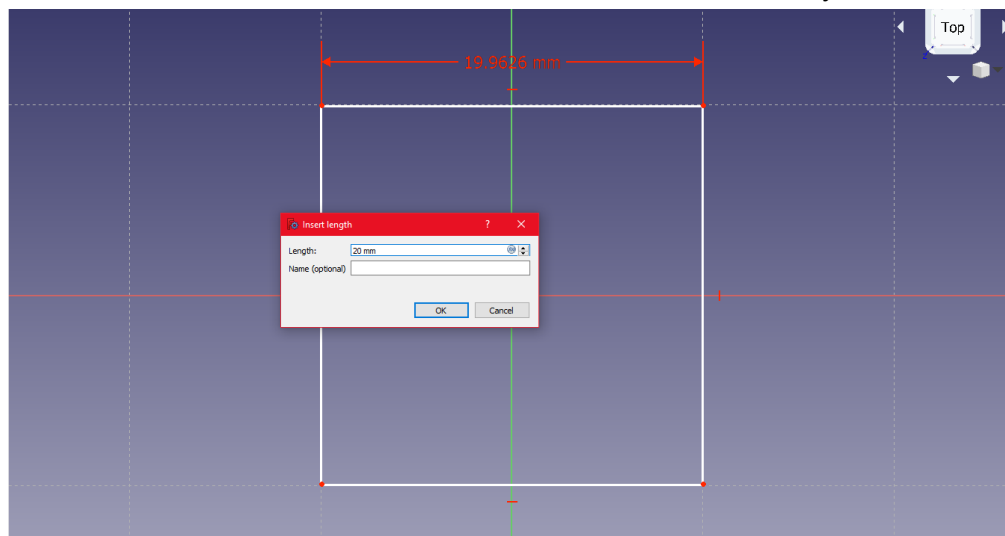
Once the button click twice on the plane to create the diagonal corners of your square. It may be useful to try and centre your square around the centre of the cross (the origin of your plane).



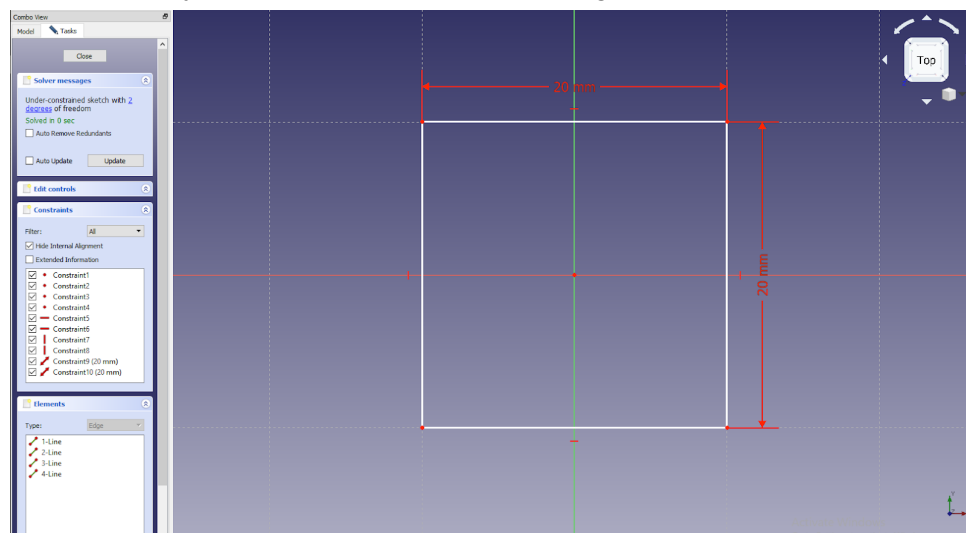
Your shape may look square, but it is unlikely to have perfect dimensions. It is necessary to add some “constraints” to tell it what size it should be. To do this we will be using the “fix length” button, it can be found by pressing the double arrows to show more constraint options.



The fix length feature will tell a line how long it is (or as you will see later will tell a line how far away it should be from another point). By clicking on a side of the square it will tell you what the length of that side is. We can edit that size by changing the value in the dialog box. It is important to note that all dimensions in FreeCAD are measured in mm by default.

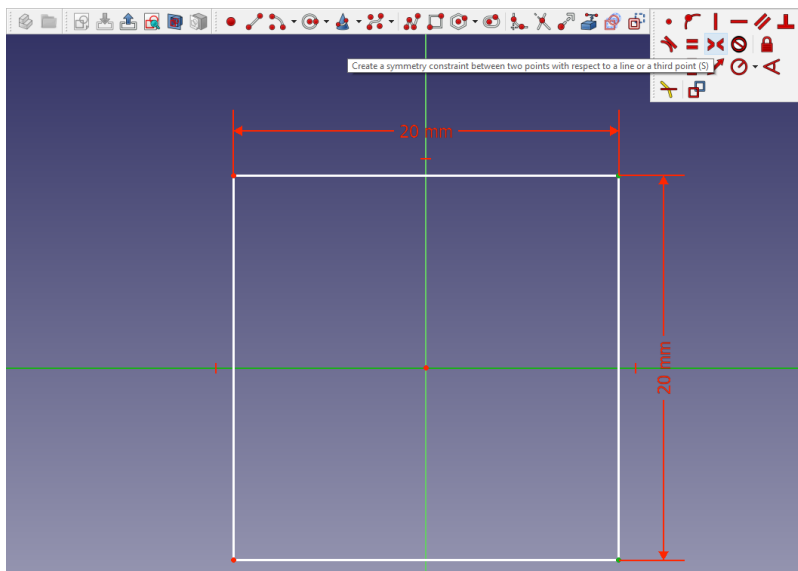


To complete the square you will need to set both the length and the width to the same size.

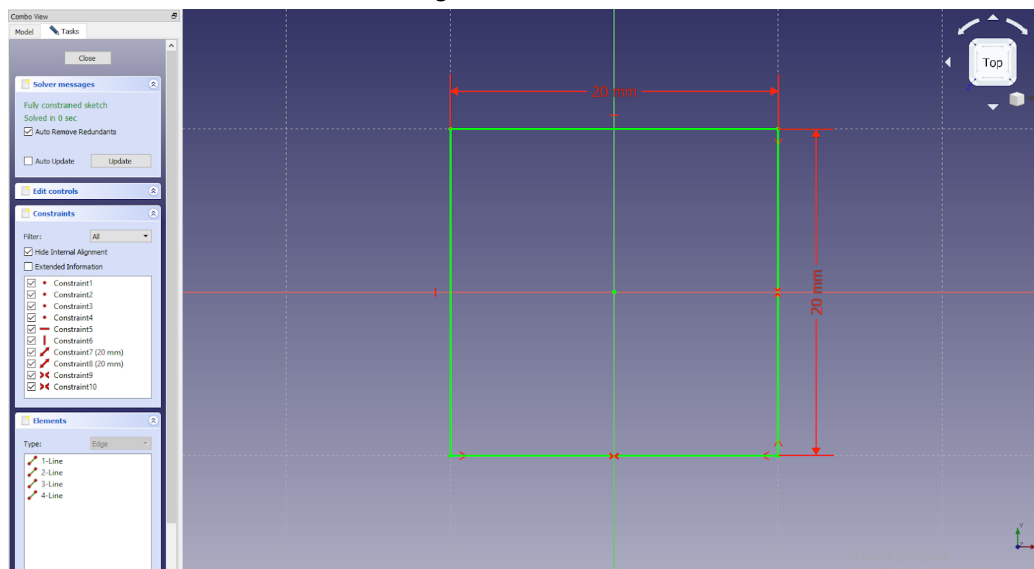


Although you have by now completed making a square, you cannot yet convert this to a cube. This is because your design still has two “degrees of freedom”, which means it can still be manipulated in two ways. To constrain these degrees of freedom we will have to use more constraints.

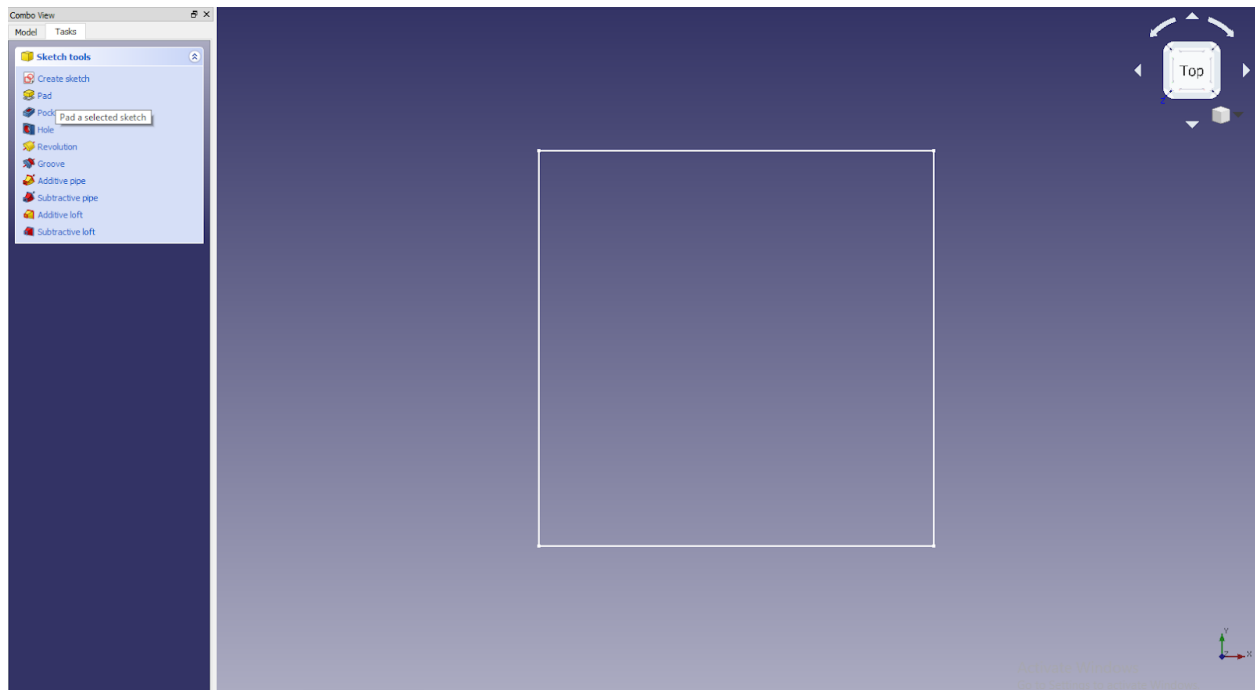
In this case the sketch can still be moved in relation to the plane (in either direction). So to constrain it is necessary to relate it to the cross (the axis) or the dot (the origin). This can be done by using the “symmetry” constraint, which is found above the “fix length” constraint. The symmetry constraint will set two points (the corners of your square) an equal distance apart from a line of symmetry (one of the axis).



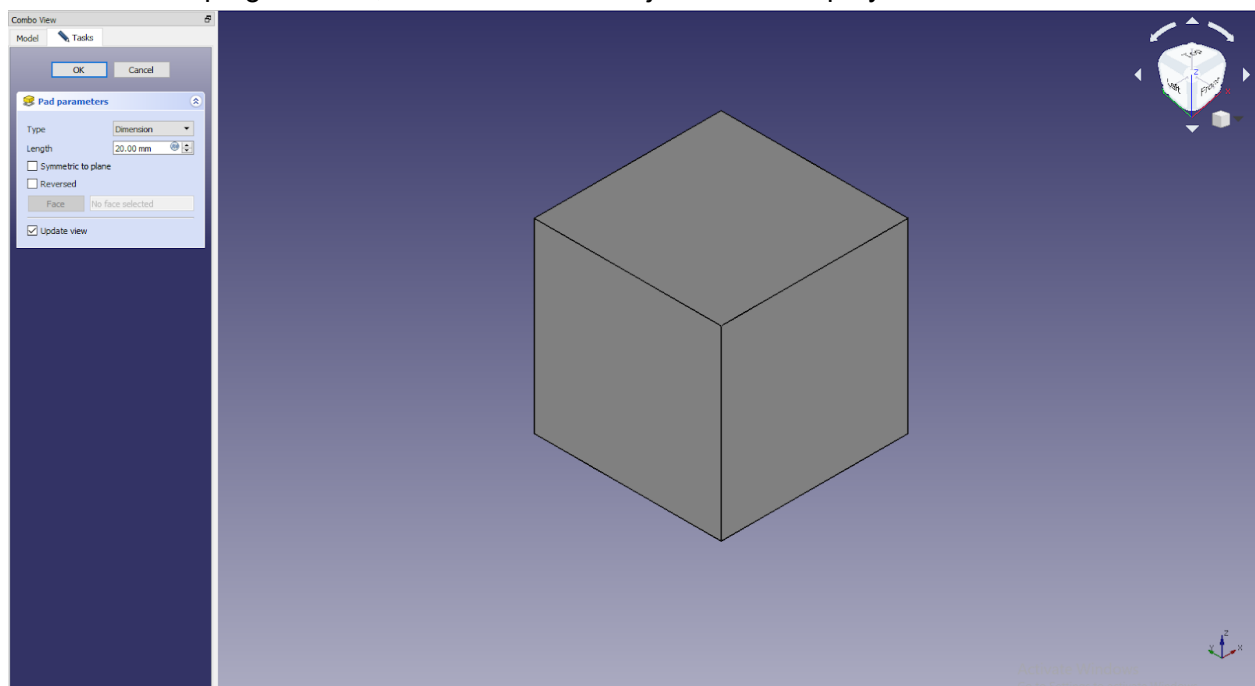
Once this is done your design is fully constrained and can be turned into a 3D object now. In Combo view now click close to exit editing the sketch.



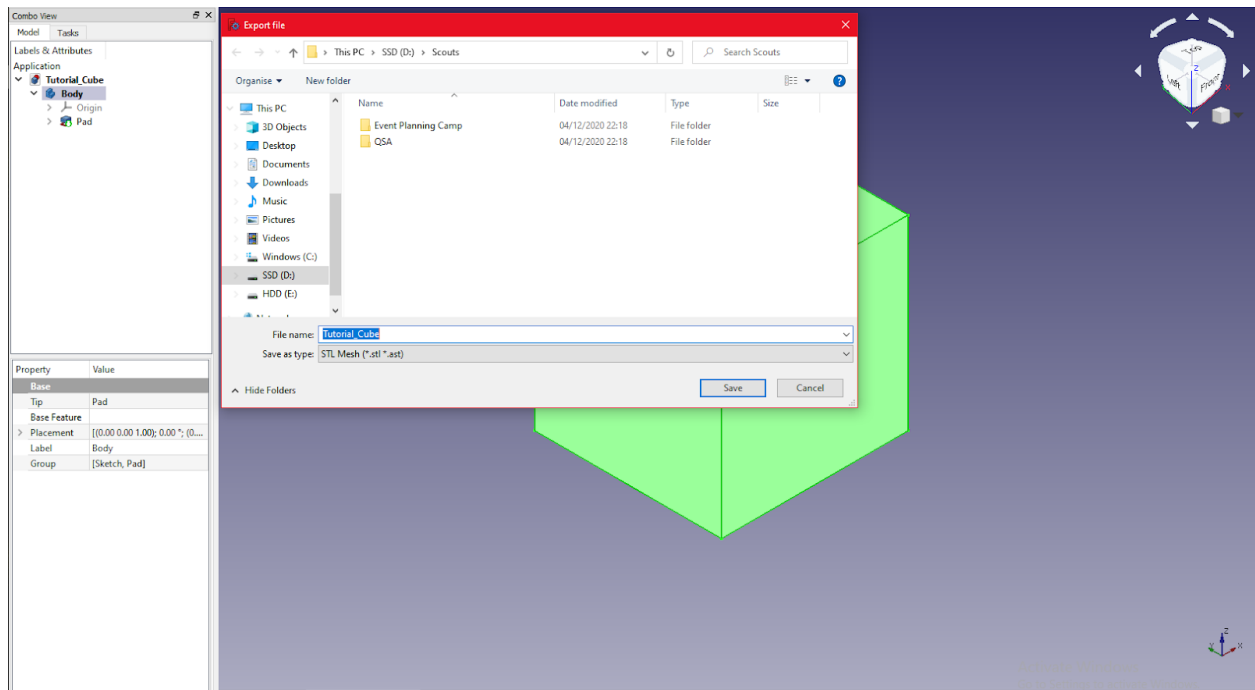
You will now be greeted to a similar screen to this. There are many ways that this shape can be converted to a 3D object. The simplest method of conversion is to “extrude” the shape out. This can be achieved by using the “Pad” feature.



The “Pad” feature will simply extend the object upwards the amount that you specify. To complete the cube you will have to extend the object by the same size as the length (and width) of your square. Before confirming you may wish to change your viewing angle around, using the arrows in the top right corner, to check that the object is the shape you desired.

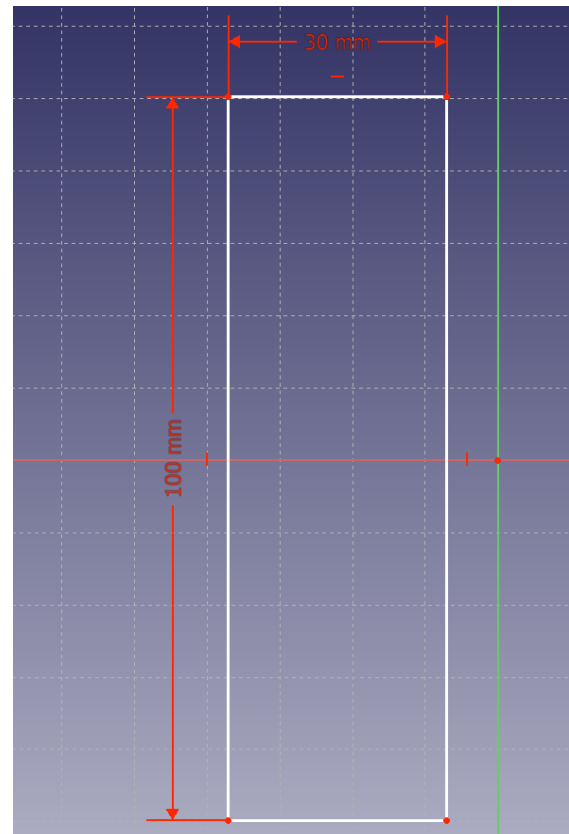


If the object you have created is intended to be 3D printed then it will have to be exported as an .STL file. To do this click on your object then go file -> export. You should save this as “your name -name of object.STL”.

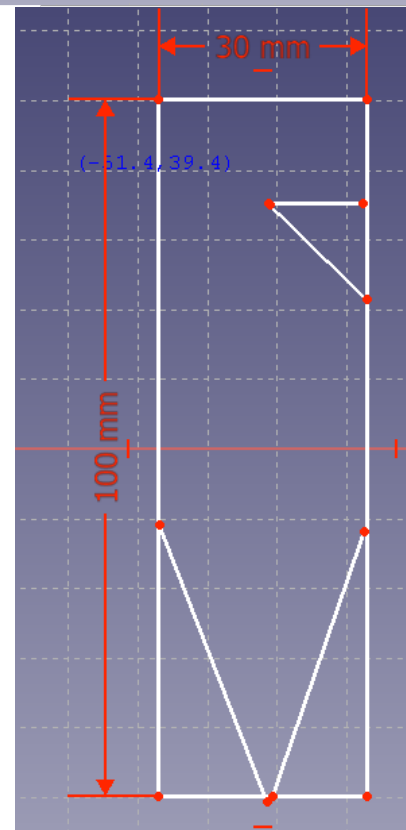


## Tent Peg

To create the tent peg you can make a new file in the same way as was described in the cube tutorial. Once you have created your file you need to create a rectangle of the general size of your tent peg. I have used dimensions to 100x30mm, but you may wish to create yours thinner.

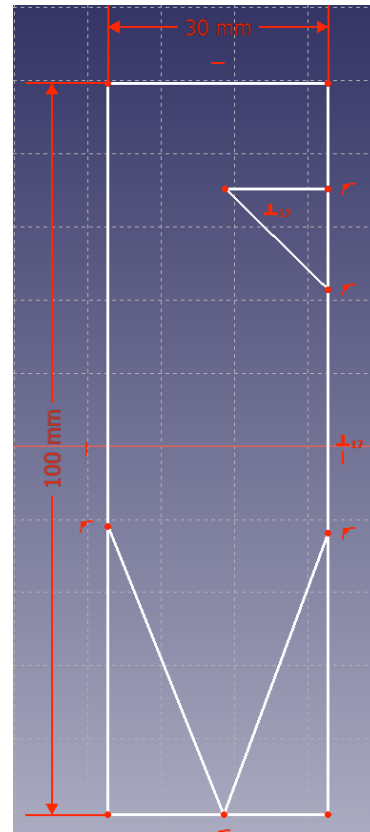
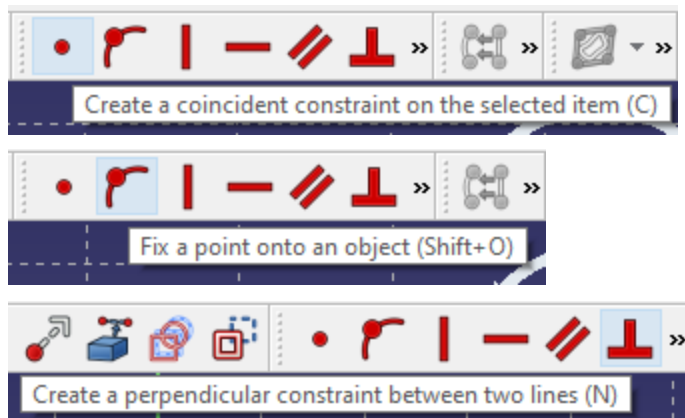


Now is a good time to block out the general of your tent peg. To do this create the shapes that you want the tent peg to take. I created a stake at the bottom and a notch at the top for the guide rope. To draw these I used the draw a line tool, it is not important to be precise as we shall align the lines using constraints.

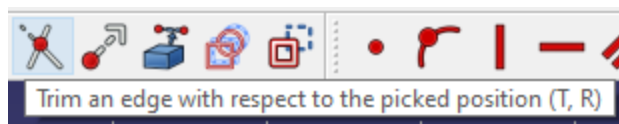




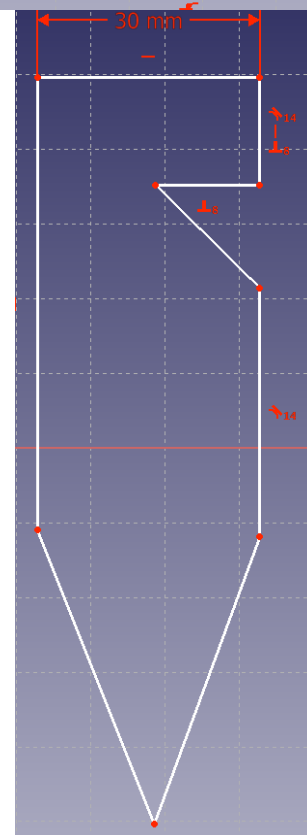
Then using the constraints connect your lines and points so that they are incident to the rectangle; I used point, incident, and perpendicular constraints.



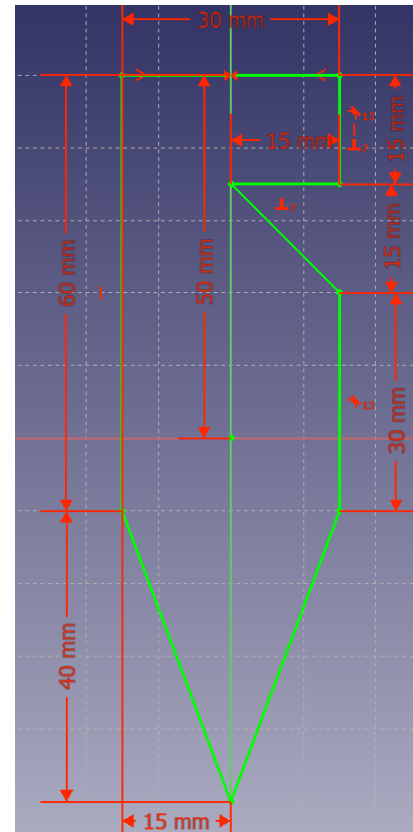
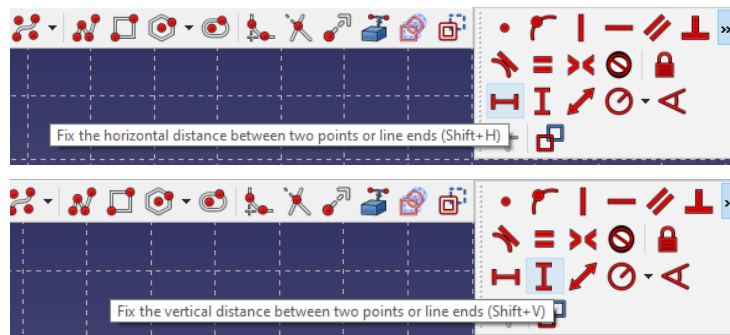
Using the “trim edges” tool remove all the lines that do not make up the outline of your tent peg. You may also delete lines by selecting the lines and pressing delete. If you are unable to remove the lines by either method it means too many of the constraints are in relation to it and you will have to do that step again.



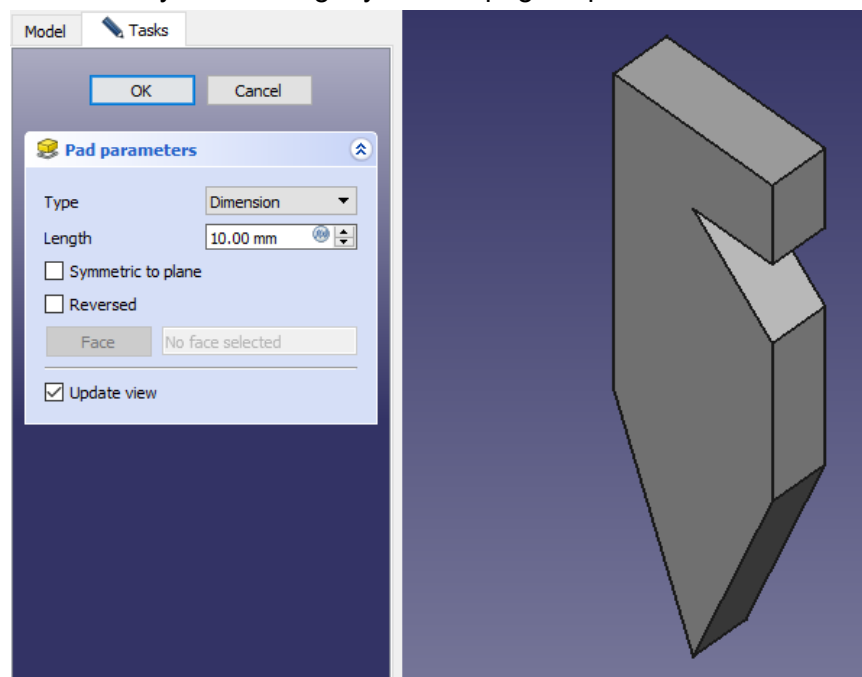
It is useful to note that the length dimension will be removed whilst you are doing this as you have modified the line that it refers to.



The final step in creating the sketch is to constrain your design. As with the square in the cube design you will have to define each length of the shape and define the position of the shape in relation to the plane. When defining the length of a diagonal you may wish to define it in terms of its horizontal and vertical height opposed to its absolute length. To do this you can use the horizontal and vertical constraints.

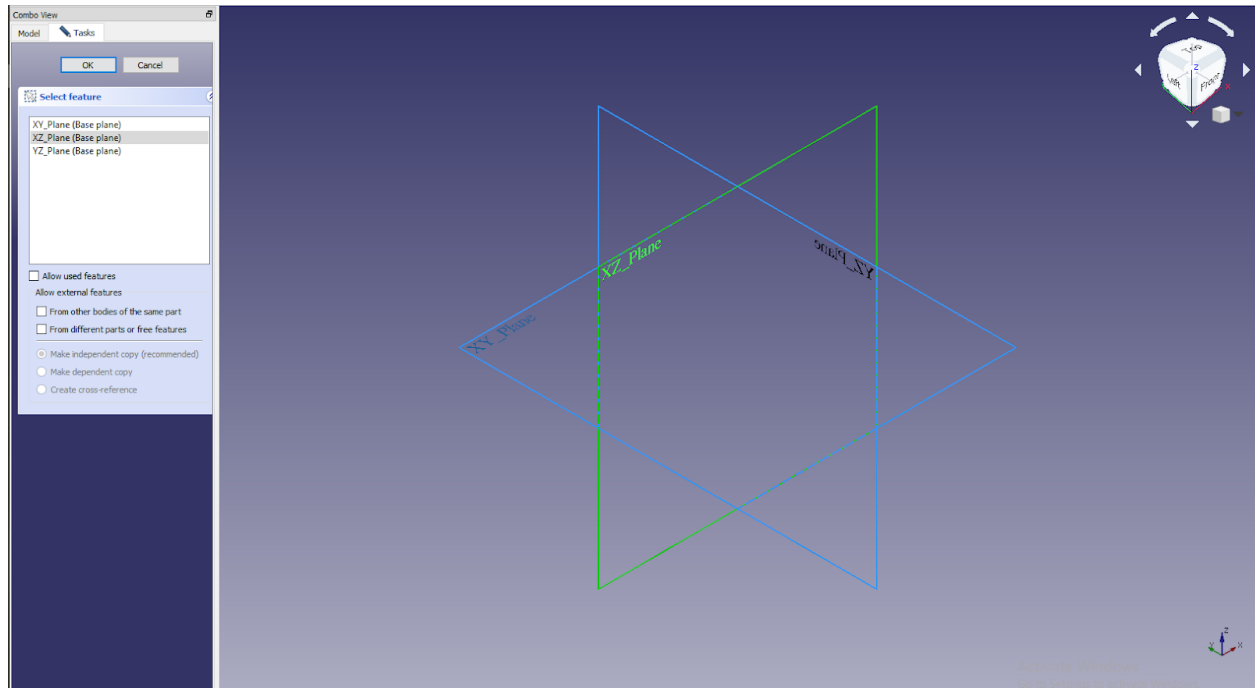


Finally you will have to extrude it into 3D using the “pad” tool in the same way that you did with the cube. If you wish to get your tent peg 3D printed it will also have to be exported as a .STL.

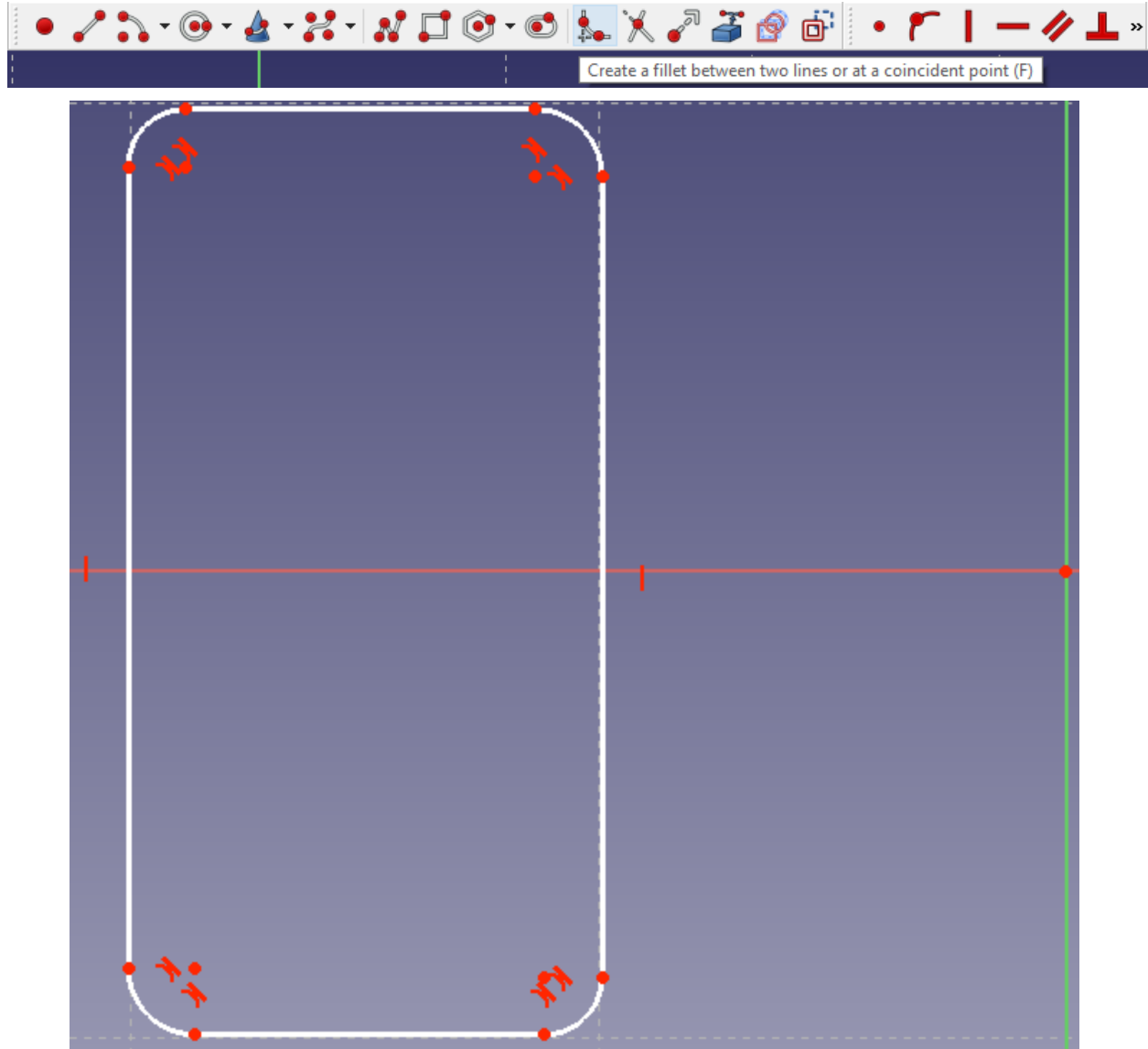


## Woggle

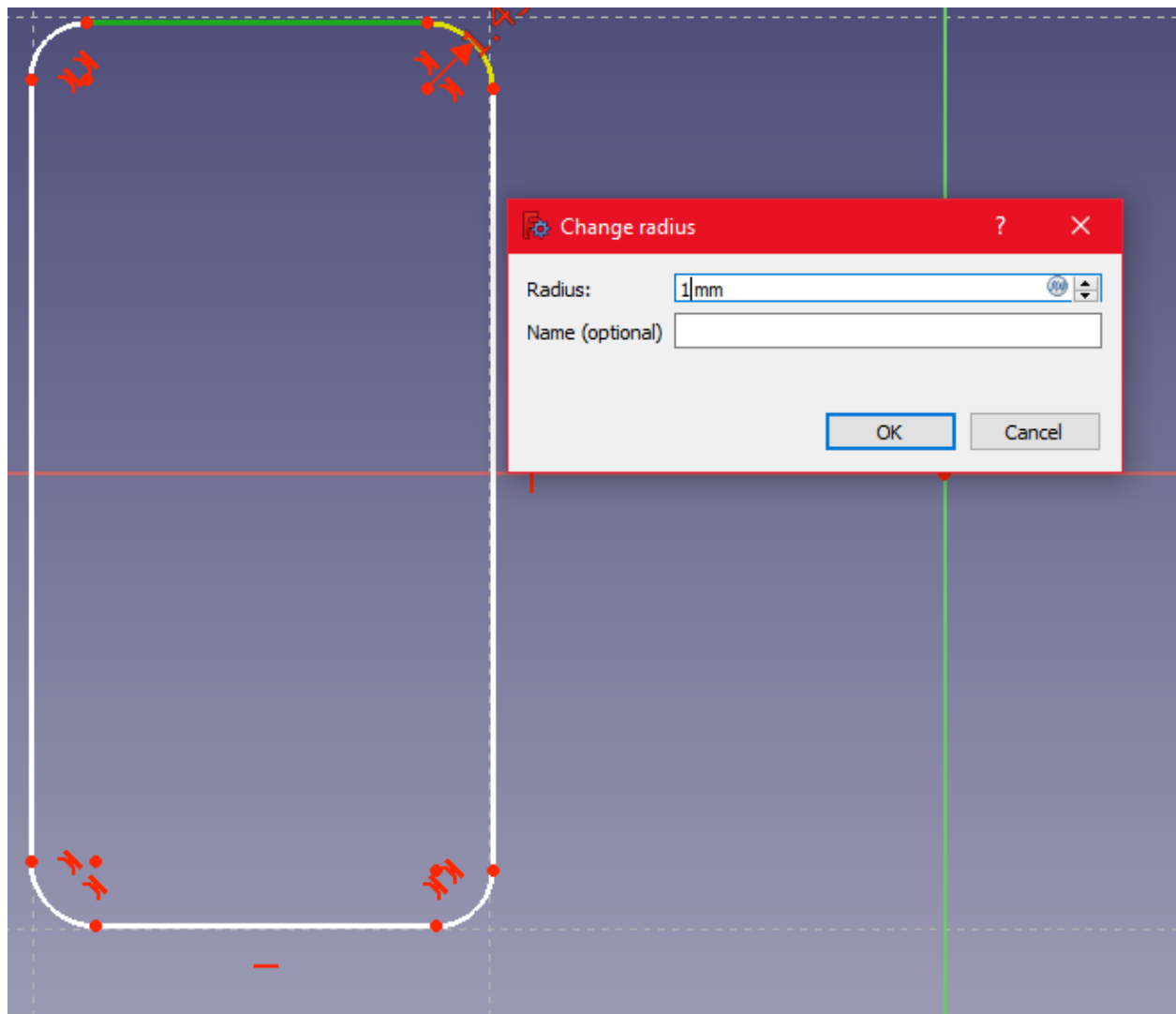
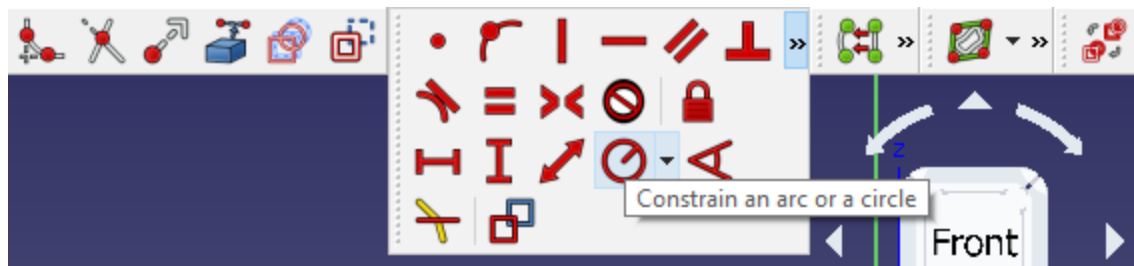
The final object that we will explain how to create in this tutorial is a woggle. To create the woggle we will be extending the 2D shape into a 3D object in a different way. As a woggle is a roughly cylindrical shape we will be rotating the shape instead of extending it upwards. For this reason I would recommend the XZ\_Plane for this design, but if you use another it won't particularly matter.



To start this sketch you should create a rectangle, using the rectangle tool as you have done in the previous tutorials, offset from the centre of the page. As this object is to be worn we will be rounding off the edges to make it comfortable to the user. To “round off the corners” we will use the “fillet” tool. After selecting the tool on the toolbar, either select two meeting lines or the point at which they meet. This will create an arc between the two.

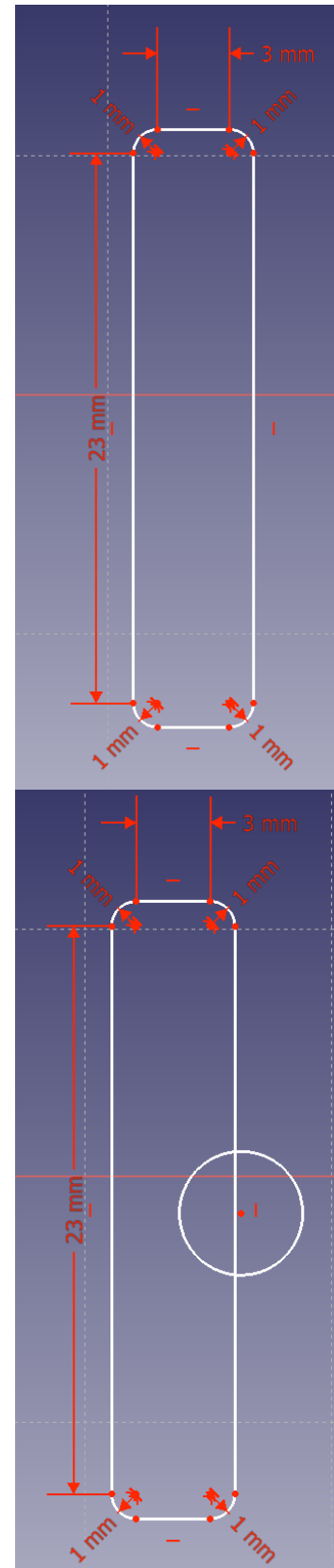


We can now define the size of which we wish these arcs to be. To do this we will have to define its radius. Use the “constrain an arc” tool and set its radius to 1mm. 1mm is smooth enough to take the edge off the project but if you want a more pronounced curve consider 2mm.

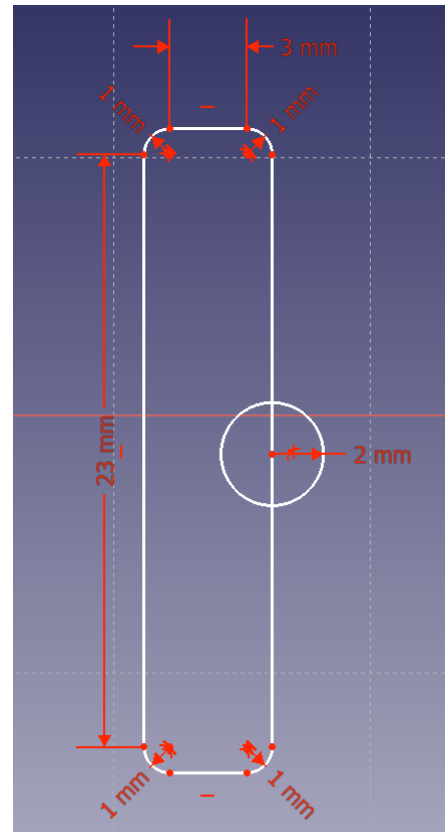
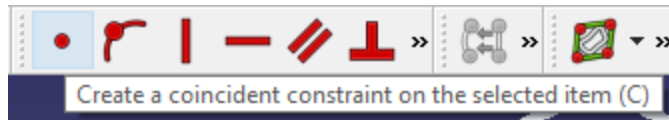


Once you have created the fillet you should define the height and width of your shape. Using the length tool set the height and width between arcs to 23mm and 3mm respectively (if you used 2mm arcs you should use 21mm and 1mm). When setting the height it is important to define the side furthest from the centre, this is because we will be modifying the side closest to the centre.

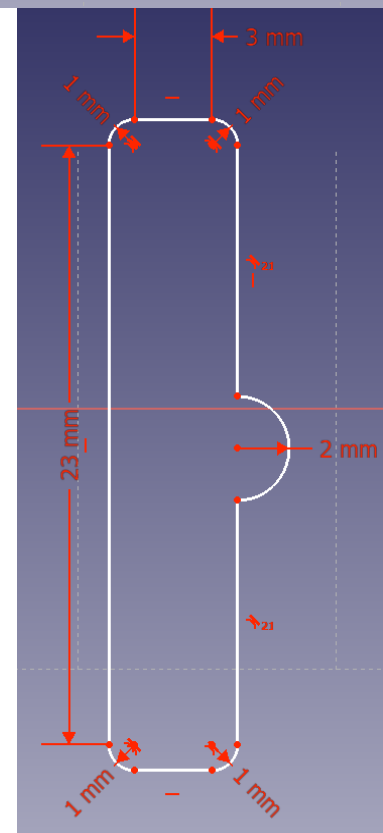
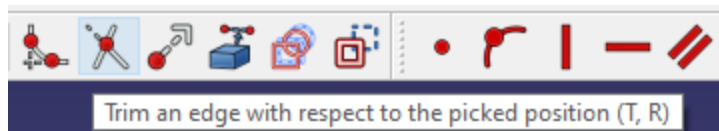
Now we will be creating a tab to grip to the fabric of your necker and prevent the woggle sliding off. To do this you will use the “create a circle” tool and place its centre close to the inside length.



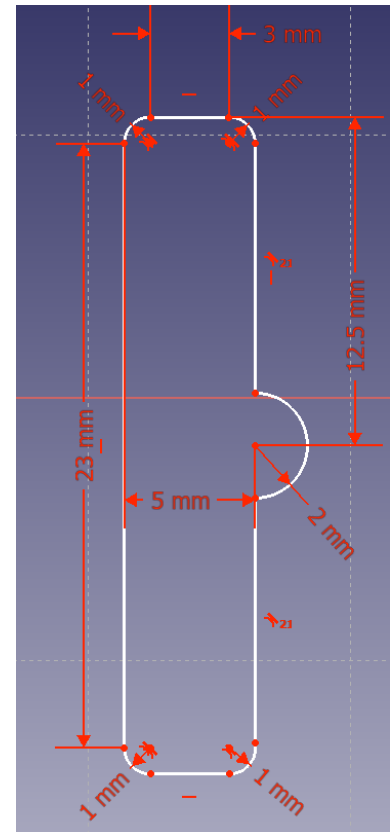
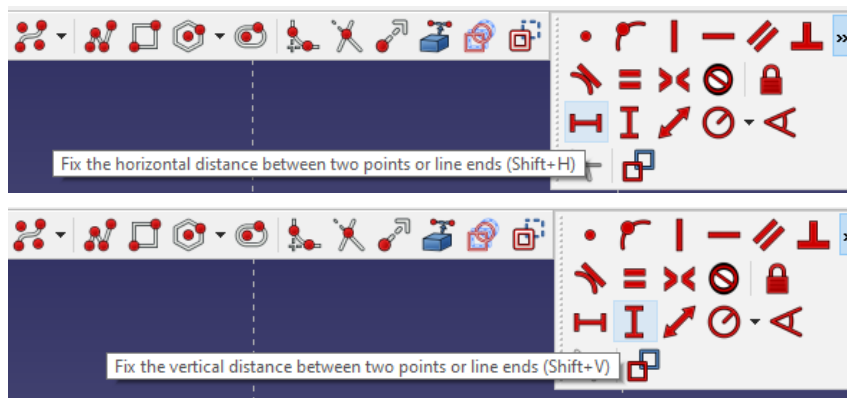
The next step is to constrain the circle that you have just drawn. To do this use the radius tool as before to set its size to 2mm. You should also constrain the centre of the circle to the inside length.



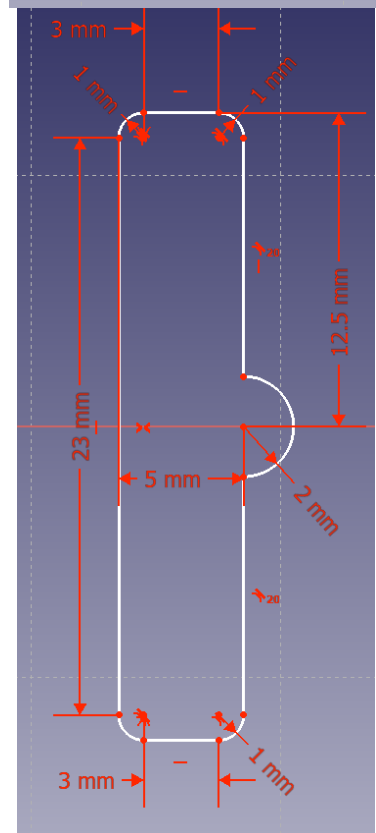
Then using the “trim edges” tool remove any excess internal details.



Having removed the detail the semicircle will no longer be defined in relation to the rest of the shape. Use the horizontal and vertical constraints to redefine the semi circle.

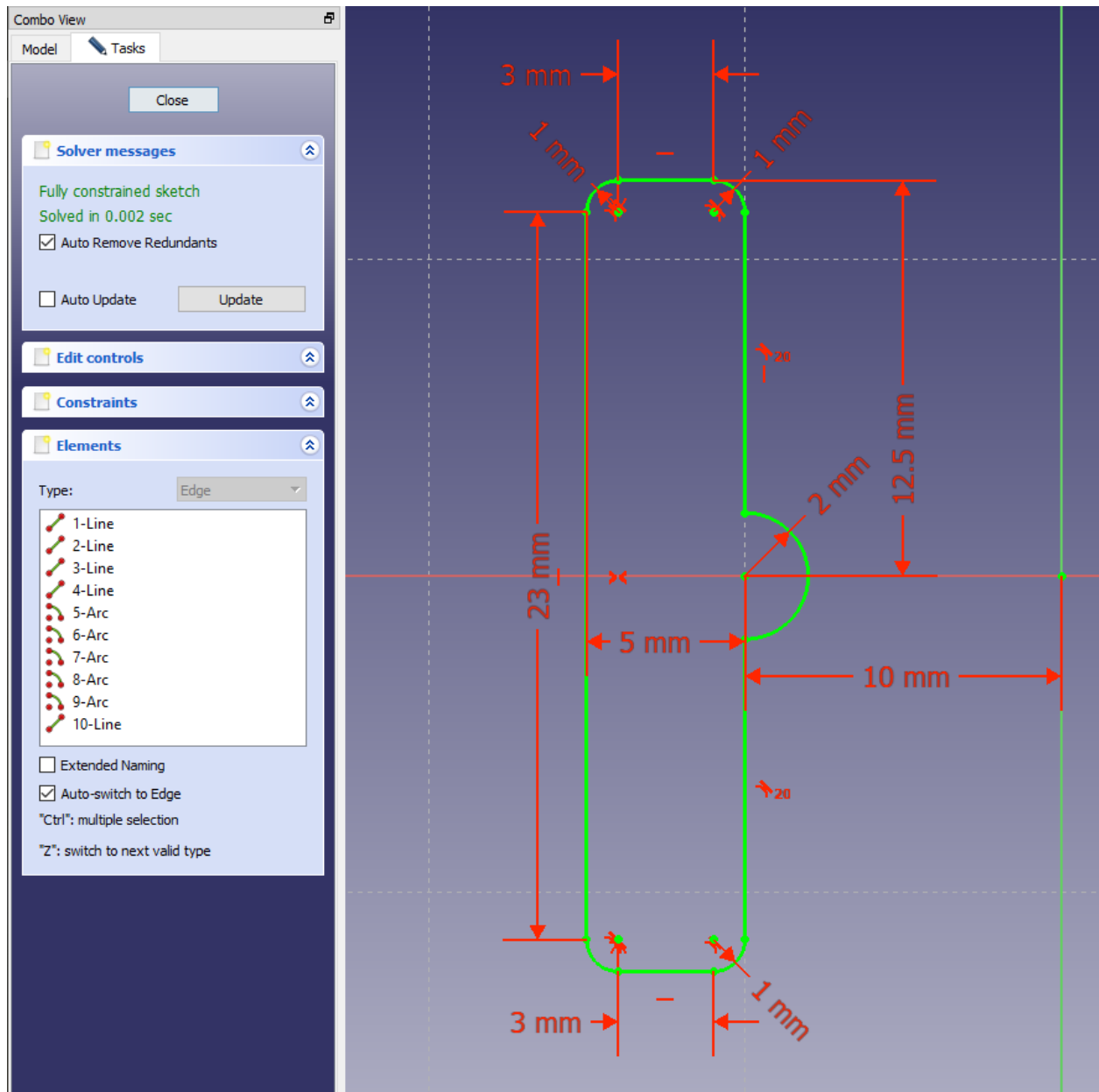


Using the symmetry tool define the vertical position of your sketch in relation to the plane. At this point you should also define any constraints left besides the horizontal position, for example I had not defined the distance between the bottom two fillets.





Finally, use the length tool (or the horizontal distance) to define the distance from the inside length and the vertical axis as 10mm. This distance has to be defined as we will be using the axis as the point of revolution when turning the object into 3D.



Once exiting the sketch use Combo view -> Sketch tools -> Revolution.. Revolve the sketch 360 degrees and you have completed your woggle. If you wish to print your woggle you must also export it as a .STL.

