FreeCAD for Beginners

Version: 2022-04-22

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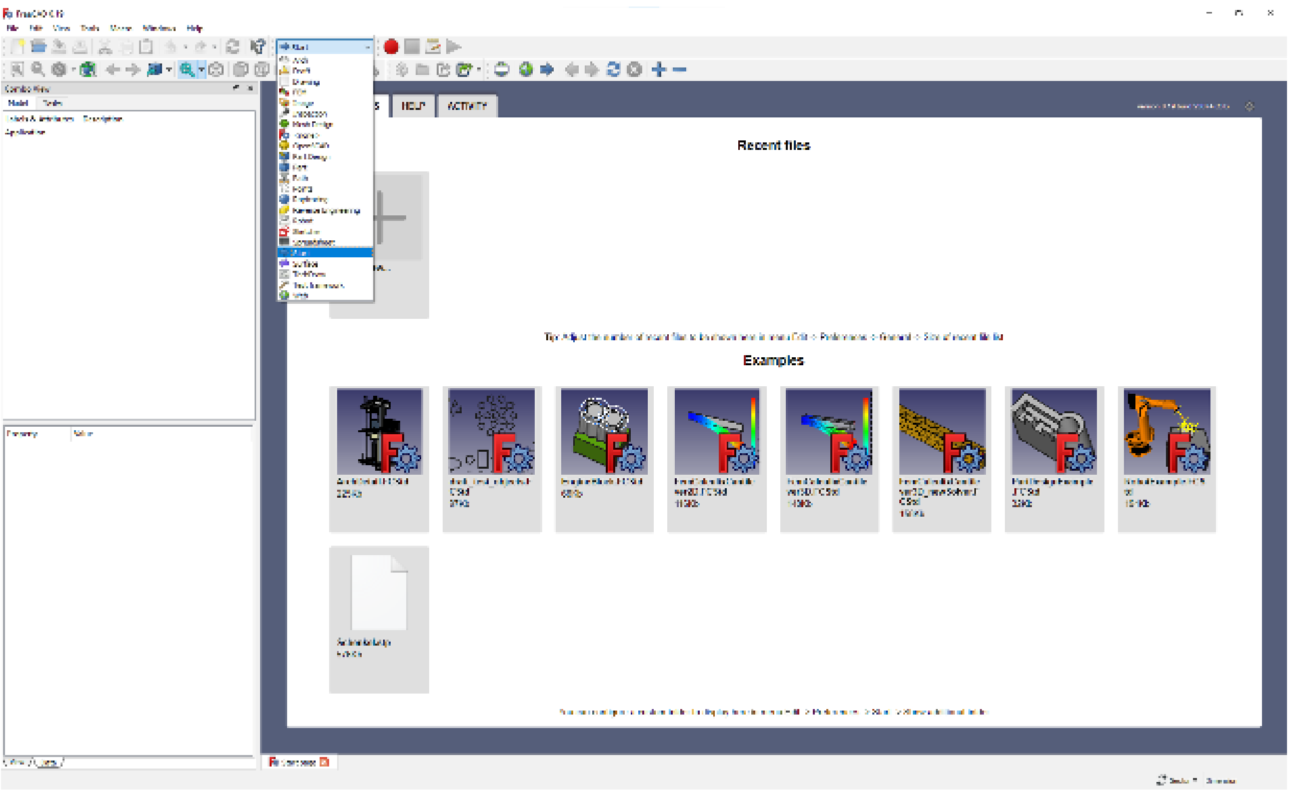
Introduction

I am using FreeCAD Version: 0.19 for Windows 64bit. 0.19 is recommended by most FreeCAD contributors and developers recommend. Download it from here https://github.com/FreeCAD/FreeCAD/releases/tag/0.19_pre and follow the installation as mentioned on the same page.

Impotrant Note
Since you started this, FreeCAD has changed so some things might no longer be in the same place or should be done a bit differently. So if you do not find the icon or menu item it as stated in this document, please do a little search. I'm am 100% sure you' ll find it back. Among other things here is just one example: FreeCAD has changed how you move a part on the Part workbench: you now right-click the part in the file tree and then select Transform.

When you open FreeCAD, it defaults to a 'start' workbench which has limited tools but shows recent files and example files in the viewer window on the right-hand side.

FreeCAD uses 'workbenches' which you can switch between. If you click the drop-down menu where it says 'Start', you will see all the installed workbenches.



The idea of workbenches is simple: imagine a large workshop with multiple benches, and each bench has a different collection of tools focused around a theme. FreeCAD has workbenches with tools to make parts, create architecture, assemble collections of parts into assemblies, create toolpaths for CNC, create technical drawings, and much more.

The beauty of these workbenches is that FreeCAD automatically carries all your work between them when you switch benches. As you progress in FreeCAD, you will use more workbenches, and you might even install extra workbenches you discover in the FreeCAD community.

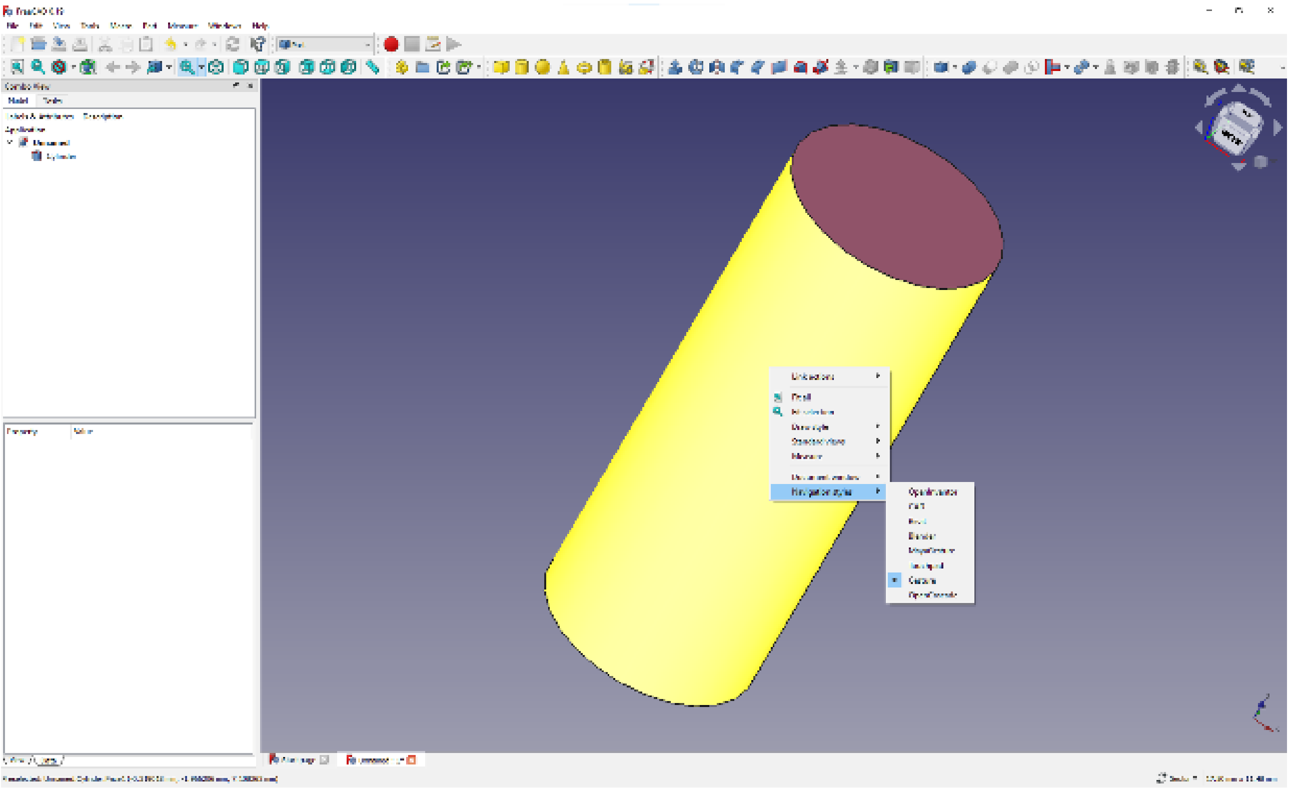
Going from Fusion360 to FreeCAD?

Watch this 10-minute video https://www.youtube.com/watch?v=_GxJkB23ZHM

First Design using Part workbench

To begin, click 'Create new' and the document viewer window will create an empty project in a new tab. Next, you are going to switch from the 'start' workbench to the 'part' workbench using the workbench drop-down menu. The part workbench is a good place to start to get oriented and to explore one of the simplest ways to make parts. Before you go too far, though, let's explore how you navigate, zooming and moving objects, in the preview window.
You should be able to see a collection of tool icons that contain yellow shapes of a cube, cylinder, sphere etc. Left-click on the cylinder icon and you should see a cylinder appear in the viewer window. It probably will have defaulted to a top view of the cylinder, so you may only see a circle as you are looking straight down from above. There are numerous ways to switch the viewpoint in FreeCAD. First, you can use the cube in the upper right-hand side of the preview window to move to view different faces. You can also use the blue 'view' icons that appear as a cube with a single face marked as solid in the toolbars. If you hover over the view icons, you get a description of the view type they relate to and also a number. This indicates that you can also swap views by just using the number buttons 1, 2, 3, and 4 on your keyboard.

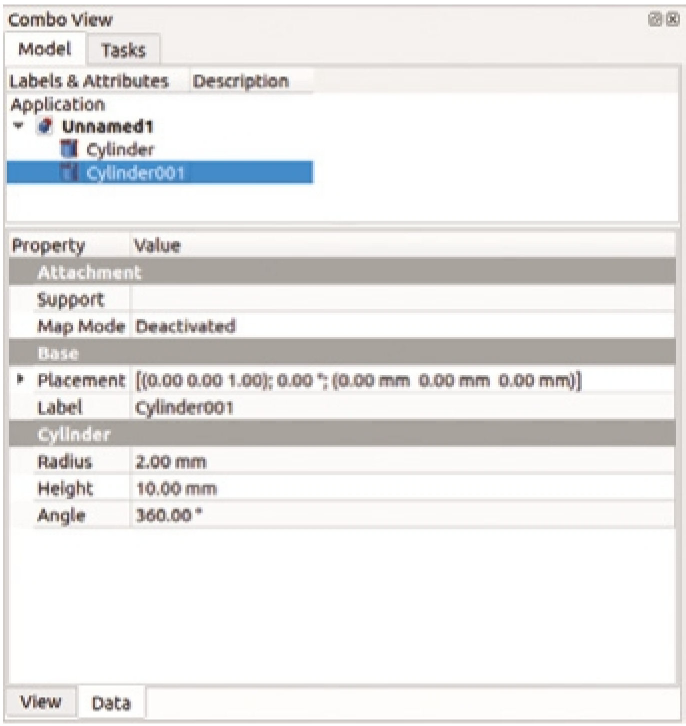
Navigation
However, a common way to navigate and select items and parts in FreeCAD is in the viewer window itself, and there are numerous different navigational styles available for you to choose from. If you hover over the viewer window and right-click, you can scroll to 'navigational styles' and a drop-down list appears.



Some of the navigational styles are based on other CAD environments, so if you are used to using, for example, Blender, then you might prefer that style. You use the 'gesture' style option that gives left-click for rotation in the document window, right-click for moving, and the centre button/wheel is for zooming. Of course, you need an object like the cylinder you created to allow you to judge which navigational style suits you best.

Set Up Your Workbench

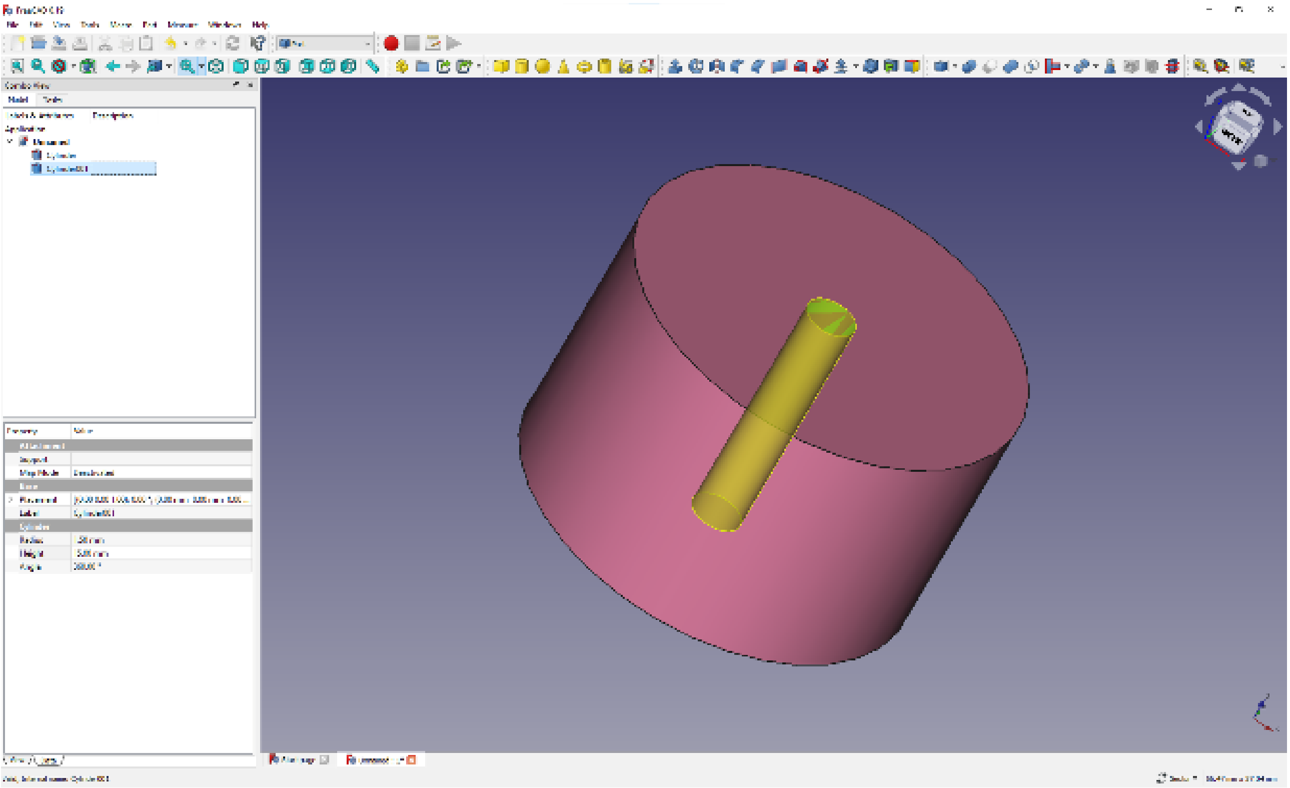
The part workbench is used largely to create parts by combining solid primitive objects like the cylinder you created. It’s definitely a good place to start, but you may find as you learn more/other ways to create parts on different workbenches, you use this workbench less. Returning to the cylinder you created, you can see that on the left-hand side of the screen there is a file-tree-type view, and under the heading ‘unnamed project’, you can now see that ‘cylinder’ is listed.



You can single-left-click on this cylinder label in the file view to select the object, or you can click on the various faces and edges of the cylinder in the document window to select them. Clicking on the label in the drop-down menu, you should see that underneath this a new dialog box ‘Data’ appears with details about the dimensions and position of the cylinder. Initially, the first things you can change here are the radius and the height of the cylinder. Let’s imagine you are making a simple wheel for a toy car, and make the wheel radius 12 mm. Next, let’s set the height of the cylinder to 15 mm.

Whole Holes

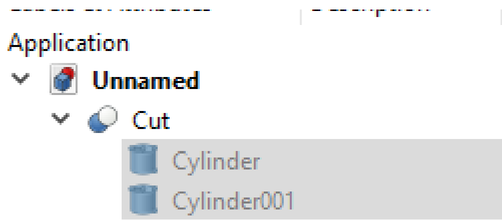
Click the cylinder tool once again and you'll see another cylinder appear in the file tree view. You may not see the cylinder appear in the document viewer window, however, as it's smaller than the other cylinder and is currently positioned inside it. If you left-click the new cylinder in the file tree view, it should appear highlighted in the document viewer window. Change the radius of the new cylinder to 1.5 mm and adjust the height to match the first cylinder, which was 15 mm. If you click anywhere off the image of the cylinders in the viewer window, you should now be able to see the ends of the cylinder you created in the top and bottom of the original cylinder.



Quick Tip: Hover your mouse over any tool icon in FreeCAD for a couple of seconds to get a text description of what the tool does.

Slicing And Dicing
Similar to most drawing and CAD packages, you can combine objects in multiple ways to create new objects. You should be able to see a selection of tool icons, some of which feature blue circles overlapping in different ways. Hovering over these icons, you should find one described 'make a cut of two shapes'. You are going to use this tool to cut the 1.5 mm radius cylinder you just made out of the larger one. Select the larger cylinder first in the file tree and then hold either the SHIFT or the CTRL key down whilst you select the smaller cylinder. Once they are both selected, click the 'make a cut of two shapes' tool icon and a hole should appear through your first cylinder. If for any reason both cylinders disappear, it probably means you have selected them in the wrong order and have cut the larger cylinder out of the smaller one, which leaves no object behind!
Press Ctrl + Z to undo and try again. The shortcuts may be different on your OS.
NB: You can see, and change them in the Tools > Customize menu.

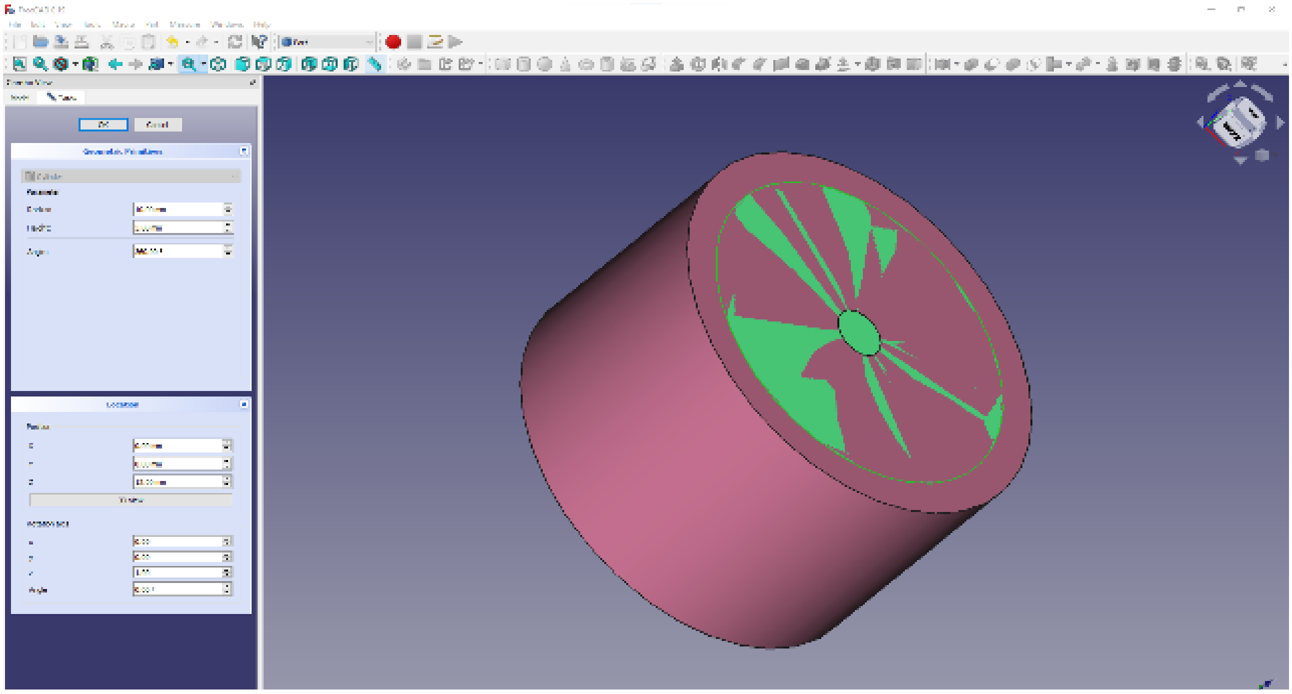
You should now see that in the file tree your cut object has been renamed to 'Cut'. If you click the small arrow to the left of 'Cut', you should see a drop-down which contains both your original cylinders but greyed out.



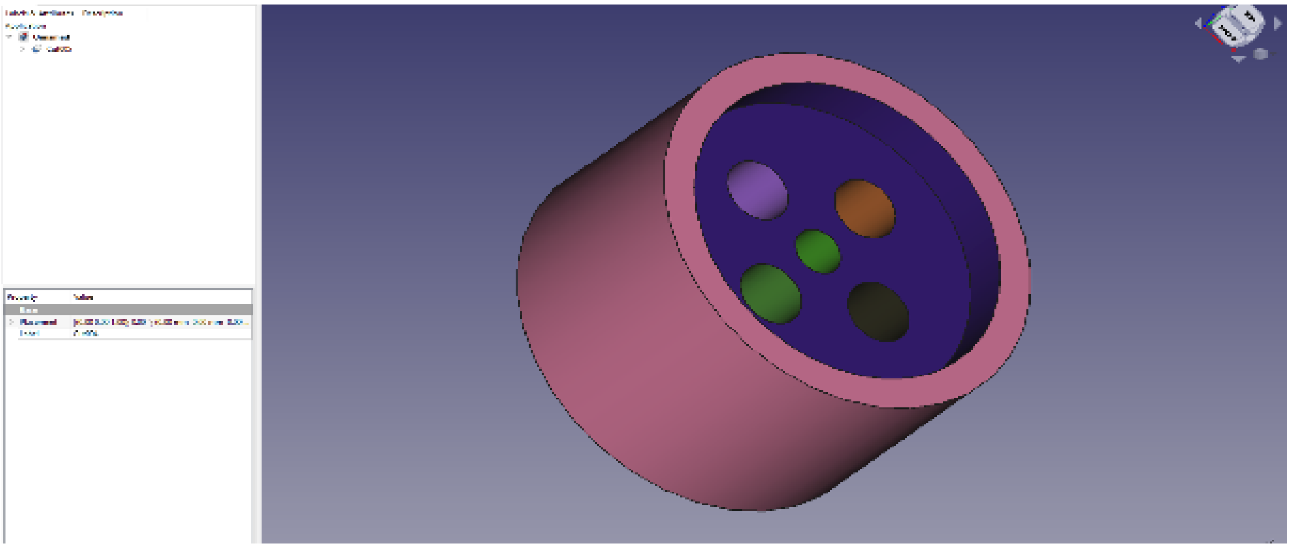
These objects, despite being components of a completed cut, can still be altered.
For example, if you wanted to change the radius of the hole to 2 mm from its current 1.5 mm, you can do this without redoing the cut operation. Select the file tree name for the smaller cylinder, then press the SPACE bar. This should make the cylinder that made the hole visible in the viewer window. You can then use the dialog box to change the radius of the cylinder and then press the SPACE bar again to turn off the visibility of the object. This should reperform the cut with the new-sized cylinder. You can also change the position of the hole if needed by moving the object.

Wheely Good
The wheel design looks a little unrefined, so let’s add some features to make it look a little cooler. Let’s create another cylinder and change the radius of this cylinder to 10 mm, and set its height to 3 mm. You are going to perform another cut with this object, but you want to move it to the top of the wheel design. To do this, double-click on the new cylinder object in the file tree. You should now see a Tab box called ‘Tasks’. In the dialogbox ‘Location’, you can now change x, y and z-values to move the cylinder around to the right position.

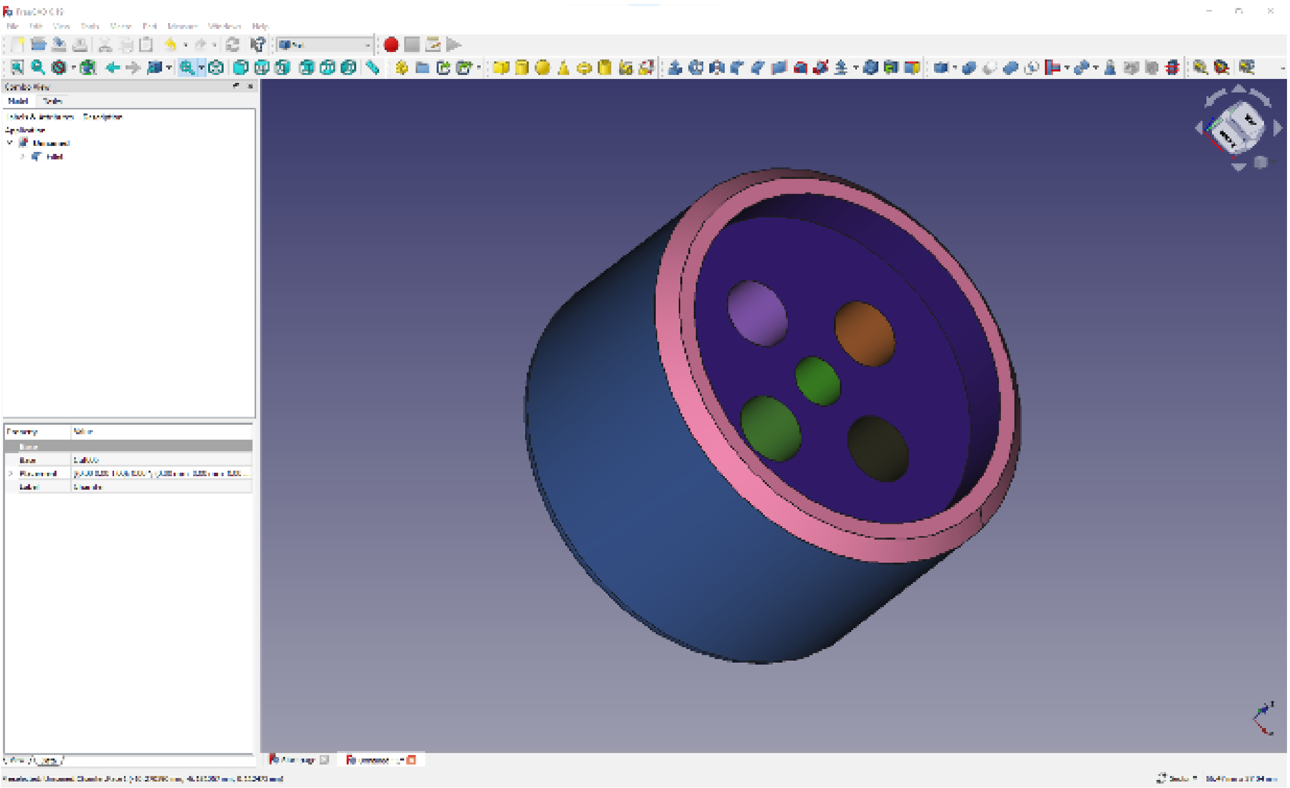
This usually defaults to one millimetre, but you can set it to any step size you prefer. For example, in the case, if you set the Z-position to 12 mm, bringing the 3 mm-high cylinder flush with the top of the design. Whilst you don’t need it for the wheel, it’s worth having a play with it. When done, click ‘OK’



Next, perform a cut to remove the new cylinder from the design.
You then continued to make the wheel look a little fancier by creating four more cylinders of 2 by 12mm. You moved the four cylinders equally outwards by 5 mm from the centre of the wheel using the moving techniques you learnt earlier. You should end up with a simple wheel.

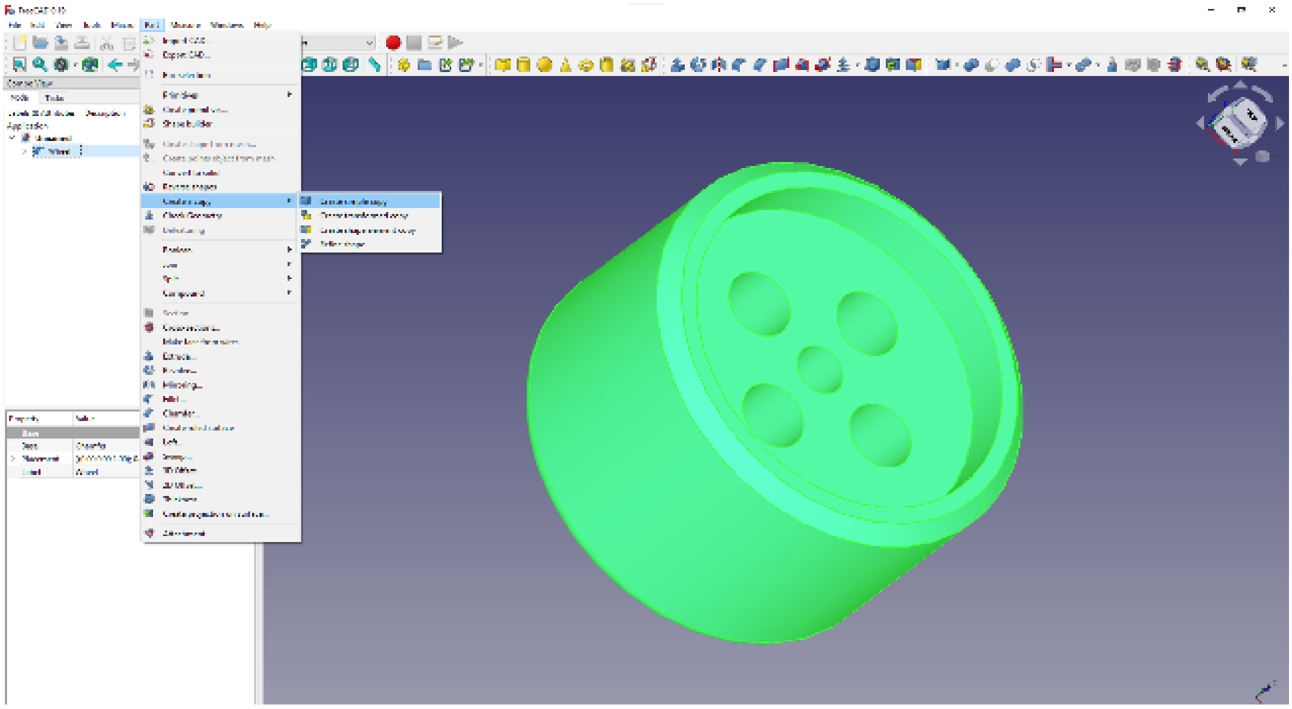


Finishing Touches
the wheel is looking wheel-like, but the part workbench has some simple tools you can use to embellish it a little. You may have noticed in the viewer window that you can highlight/select either the faces or the edges of objects. Often you can also select corner points of objects, but as everything is circular in the wheel design, that doesn't apply here. The part workbench has two nice tools for adding either fillets or chamfers to edges of objects. They appear as two tool icons: the fillet tool as a curved area between two flat surfaces, and the chamfer tool as an angled flat surface. In the viewer window, left-click to select the outside edge of the top of the wheel, and then click the fillet tool. In the dialog box, you can see a list of edges, including the selected edge. This is useful if you want to apply multiple fillets to multiple edges in one operation. You can also adjust the radius of the fillet in the dialog. You'll leave ours at 1 mm and click OK to apply the fillet. Both the fillet and the chamfer tools can be used on either external or internal edges. As an example, let's select the edge inside at the bottom of the large cut-out at the top of the wheel. Click the chamfer tool this time, and set the chamfer length to 0.5 mm and OK it. You should see that a nice internal chamfer has been created. Finally, you added a matching fillet to the bottom outside edge of the wheel.

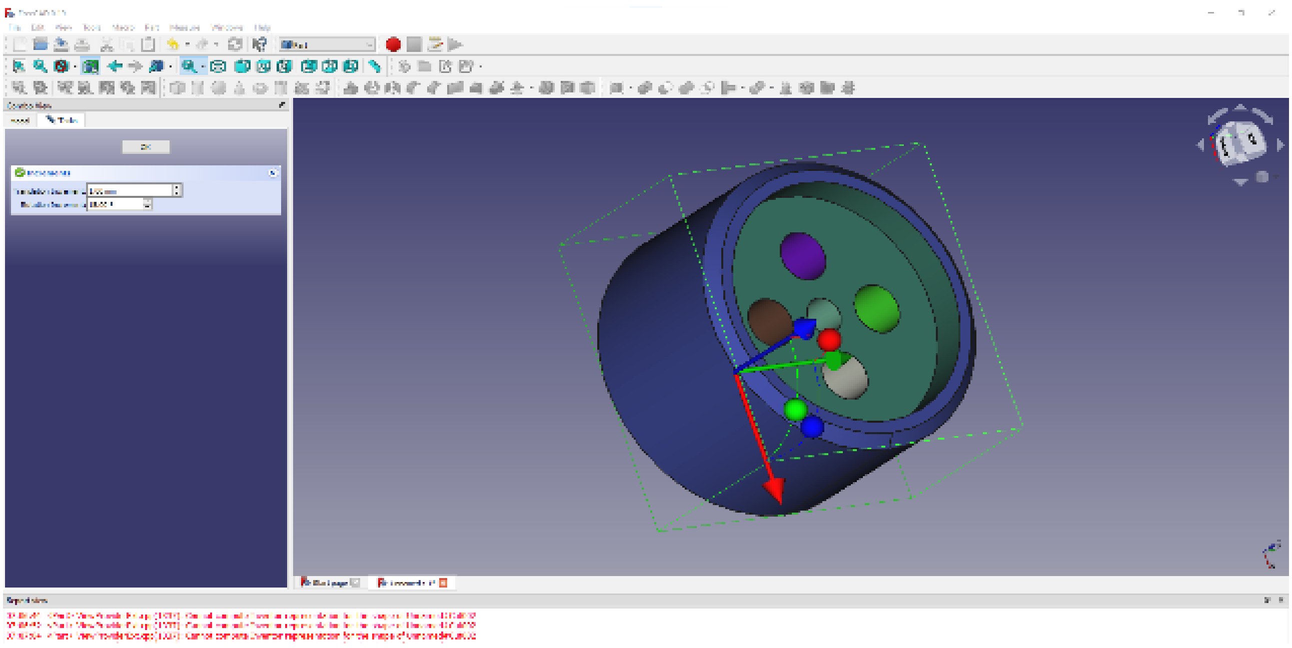


Rename
Now you have the basic wheel design (it's called 'fillet' in the file hierarchy). Whilst you can click it to select the entire wheel, if you double-click it, this reopens the dialog for the fillet operation. This means that this part can be tricky to move. A common way to deal with this, which is also useful for us as you want to have four of the wheels, is to create simple copies of a part.
First, to help keep things orderly, let's right-click on 'fillet', select 'rename', and call it 'wheel'. Renaming this doesn't change the fact that if you double-click it, it won't open up the fillet dialog, but will help us track when you make multiple items.

Create Copies
To create the copies, single-left-click 'wheel' in the file tree view to select the wheel and then go to the 'part' drop-down menu; select 'create a copy', then 'create a simple copy'. Clicking this option will create a new 'wheel001'.

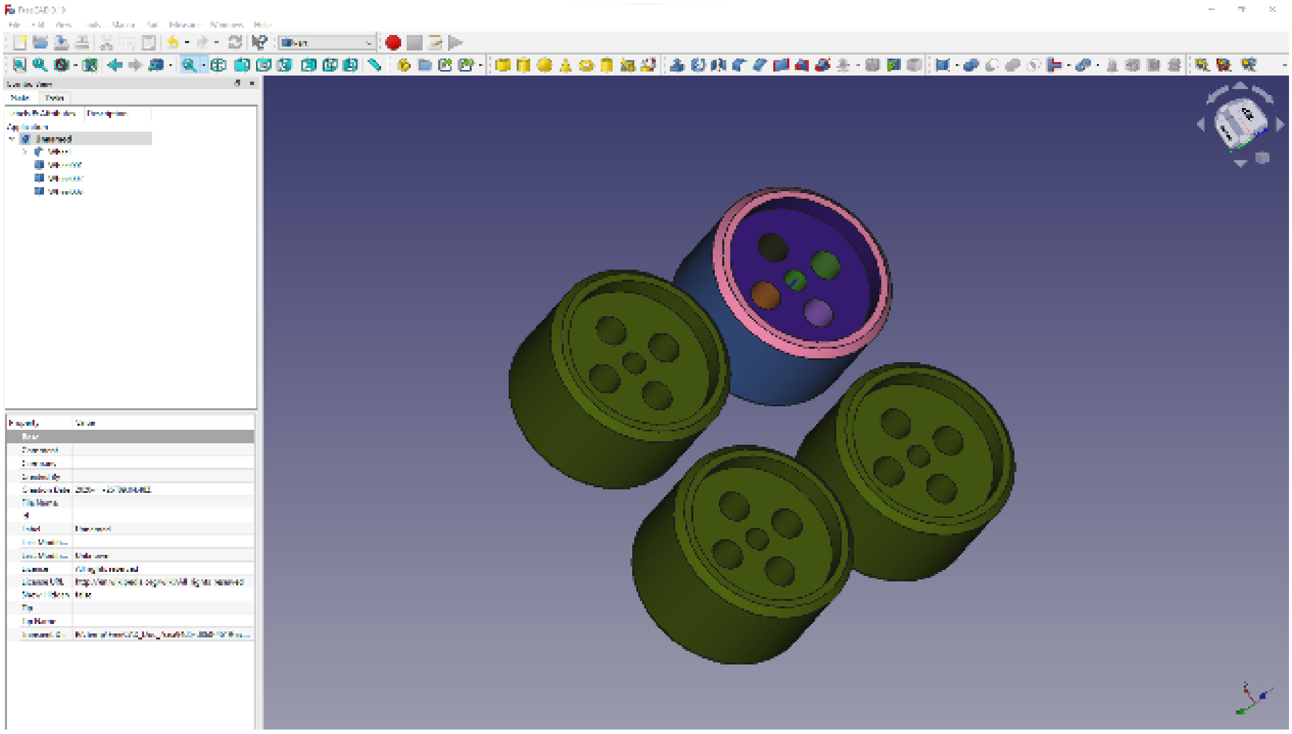


Clicking this option will create a new 'wheel001'. If you double-click on wheel001, you can see that it can now be moved in the 3 axises as well as rotate as a separate item. The behaviour of the change you can set in the 'Tasks' tab.

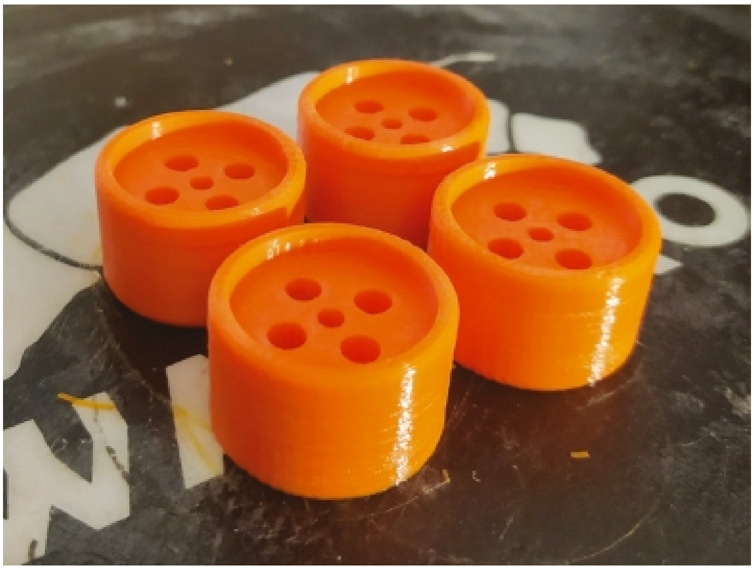


As you made a 'simple copy', your new wheel doesn't contain the hierarchy of operations and objects your original has and also is not dynamically linked to the original object. This means that if you make further changes to your original 'wheel', these aren't pushed through to the simple copies. However, of course, you can delete the copies, make a change to the design, and recopy if needed.

Let’s make copies and lay them out as a set of four.



The Next Step
We’re going to 3D-print these wheels, and you could just export one instance of the wheel and then duplicate it in the slicer software – but, again, it’s just as easy to do this in FreeCAD. Having made the four copies of the wheel, let’s save the work and then, select all 4 wheel. Next click ‘file – export’. You can export a mesh file of the wheels ready for slicing for 3D printing. Select ‘STL Mesh (*.stl, *.ast)’ as file type to save as and give it a name eg. ‘4Wheels.stl’.
Having exported the STL file, you could open it in the favoured slicer and 3D-print them.



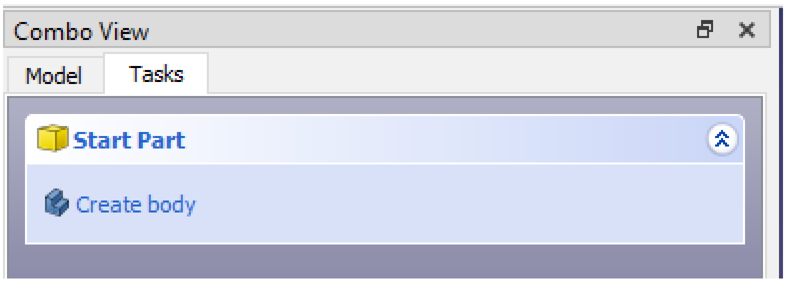
Transparency
Selecting items in the file tree makes them become highlighted in the document viewer window and, whilst this is useful sometimes, it’s easier to set some component items to be different colours and adjust their transparency. This can help us see the internal geometry of parts, and can allow us to check internal positions. To do this, highlight a part in the file tree view and right-click. Scroll to ‘appearance’. In the menu that appears, you will see that you can set the material type and also the shape and line colour, and also set the transparency.

Using Part Design and Sketcher workbenches

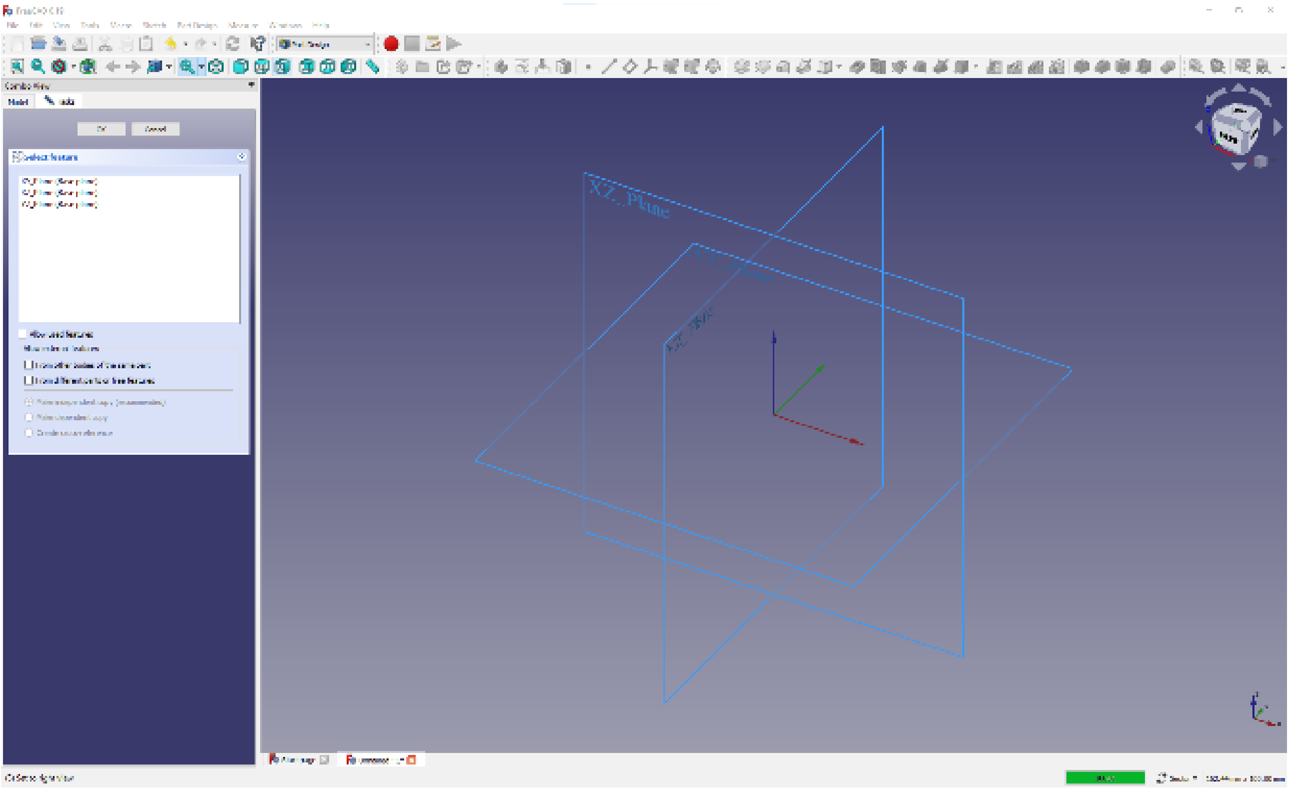
You are going to design a small bird feeder using the 'Part design' and 'Sketcher' workbenches using different methods where you will create a 'body' using the 'Part design' and 'Sketcher' workbenches. The techniques are a little more complex, but give a much more control over designs.

What is a body?
A body is best described as a component part that is one continuous solid object. To visualise this, an example could be a nut and a bolt. If you wanted to model a nut and bolt, they would both be separate bodies that could then be fastened together as an assembly. The 3D form of a nut and a bolt can be modelled as a continuous single piece, and so each can be considered a body. In this tutorial, you are using sketches and performing operations on the sketches such as 'padding' where you extrude a sketch, and 'pocketing' where you cut the shape of a sketch into an existing object. However, you can also add primitive objects on the Part Design workbench similar to the use of the part workbench.

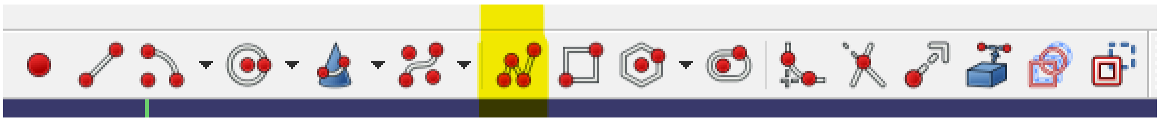
Pick your workbench
Opening FreeCAD, start a new document and then select the Part Design workbench from the drop-down menu. You should see in the combo view, on the left-hand side, a single option which is 'create body' – click this.



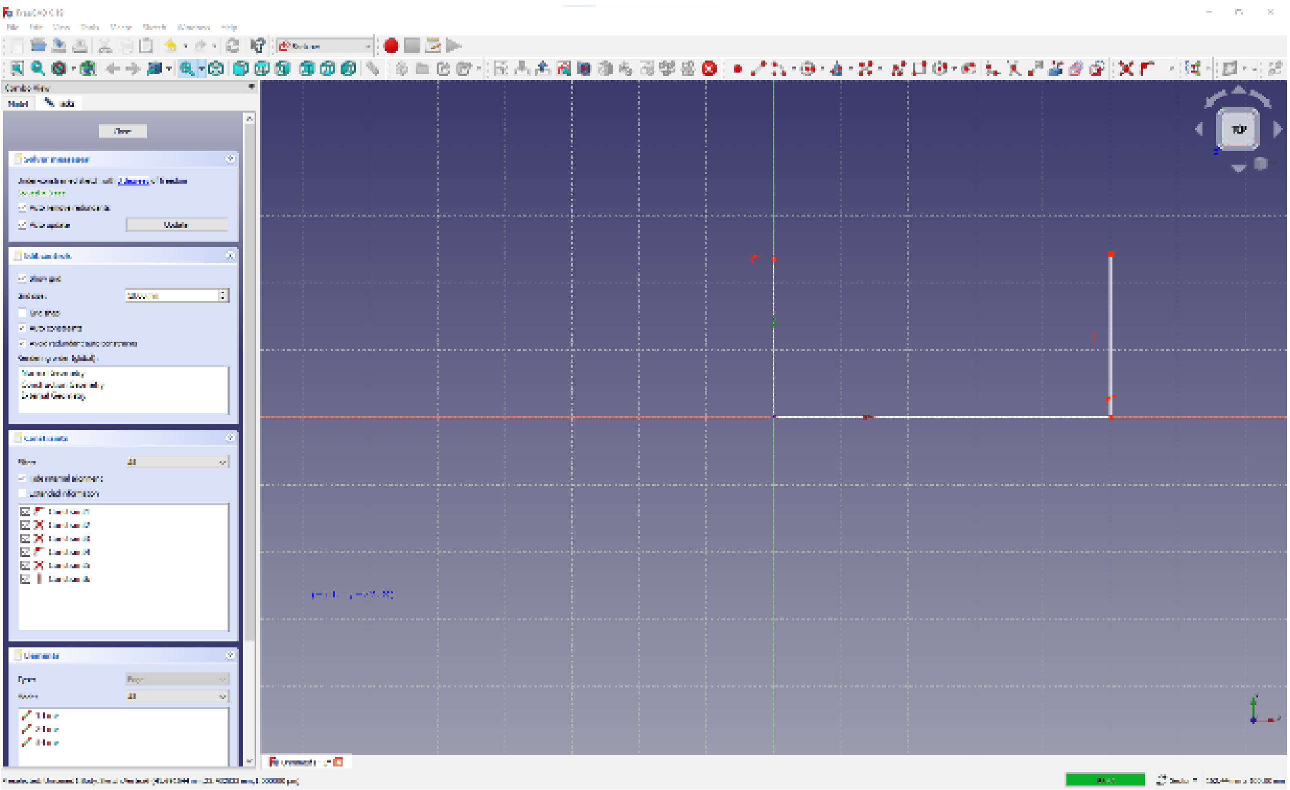
You will then be presented with two options: 'Create sketch' and 'Create boolean'. Click 'Create sketch'. You are then presented with a choice of which plane to base the sketch in: XY, XZ, or YZ.
An easy way to visualise the XY plane is by imagining a 3D printer and looking down on it from above onto the bed. If sketched onto the printer bed, then working you are working on the XY plane. Select the XY plane and click OK. At this point, FreeCAD should automatically switch to the Sketcher workbench.



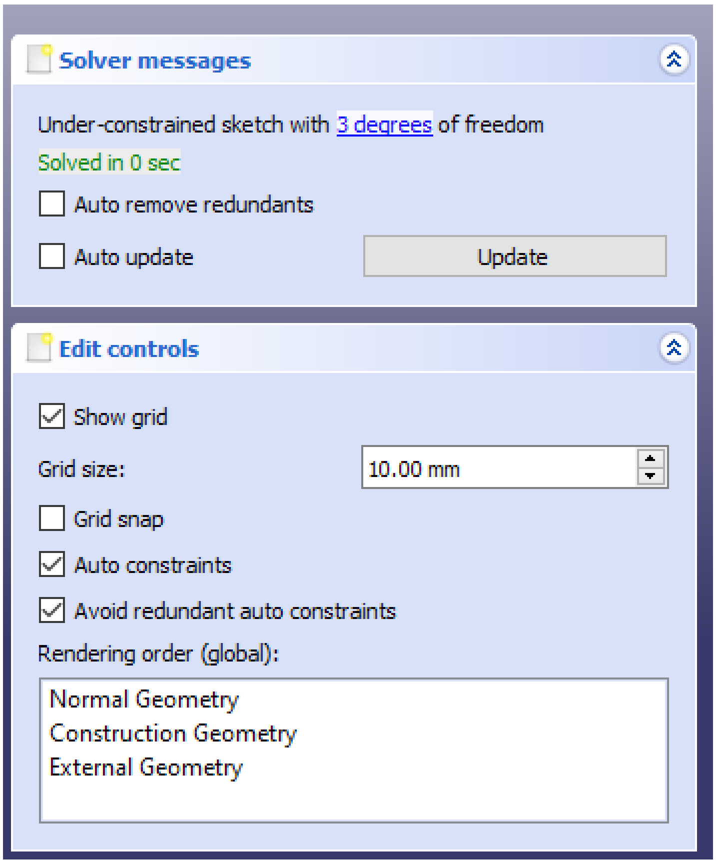
Next, select the 'Create a polyline' tool icon?



Draw three connected lines in a U-shape roughly laid out. Starting at the top left-hand side of the U, click on the vertical zero-line on the Y-axis at any height and then drag the line down to the origin point of the XY plane.
Conveniently, when hover over either zero-line or any point they will turn yellow. Clicking when yellow, will place the drawn object on the highlighted point or line. Next, pull the line out horizontally and left-click before pulling it up vertically. Right-click to end the three lines. If you have done this correctly, the horizontal line and vertical line will be white, but will have a small red horizontal or respectively vertical line.



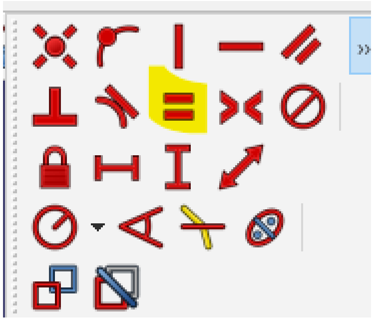
In the combo view panel, you should see a 'Solver messages' tab with a message about numerous 'degrees of freedom'.



The aim is to reduce these degrees of freedom to zero by constraining all the items within a sketch. Constraining a sketch is the process of adding constraints that complete all the information about a sketch item's positions and dimensions. This is an incredibly useful

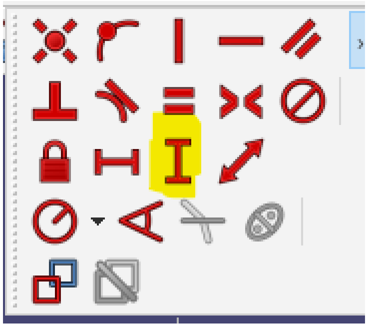
approach, meaning you can return to sketches at any point in the design and make changes that will be automatically recalculated into the whole model.
The sketcher solver automatically adds some constraints when it guesses, for example, you want a line to be vertical or horizontal, as you have just done.
Starting on a line and including the origin point on the XY plane also means that you don't need to constrain these lines positional co-ordinates, but you do need to constrain the lengths of the line. Later you will constrain items that aren't on these lines, so you'll see how to set positions of items relative to other parts of the design.

Constraining choices
As you get used to applying constraints, you will realise that there are lots of different ways to achieve the same results. To constrain the lines, go to select the two vertical lines. To do this, right-click to deactivate the line drawing tool and then left-click on both the vertical lines in turn. Next, press the 'Create an equality constraint' tool.



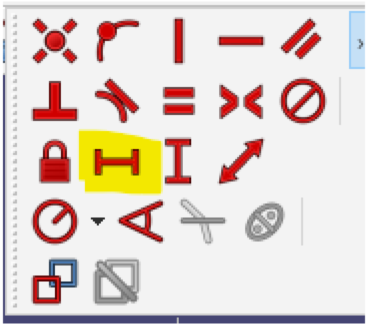
Note: you might need to expand the toolbar by clicking the double right-pointing arrow.

The two lines should now appear as an equal but undefined length. If you left-click on the red point at the top of either line and drag it, you should now find that both lines extend together as they are constrained as equal. This ability to extend the line is a degree of freedom, and sometimes, clicking and dragging items in the sketch can help identify what you need to constrain. Left-click to select 'Fix a vertical distance' tool icon

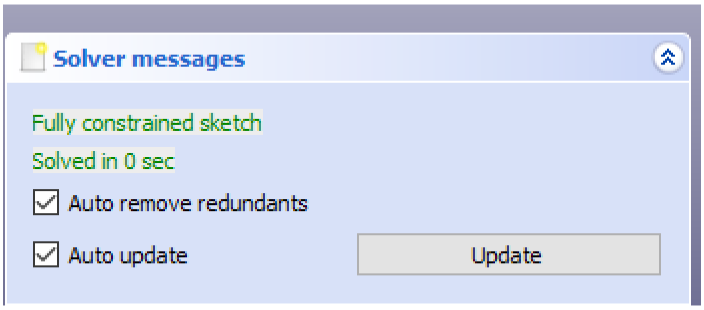


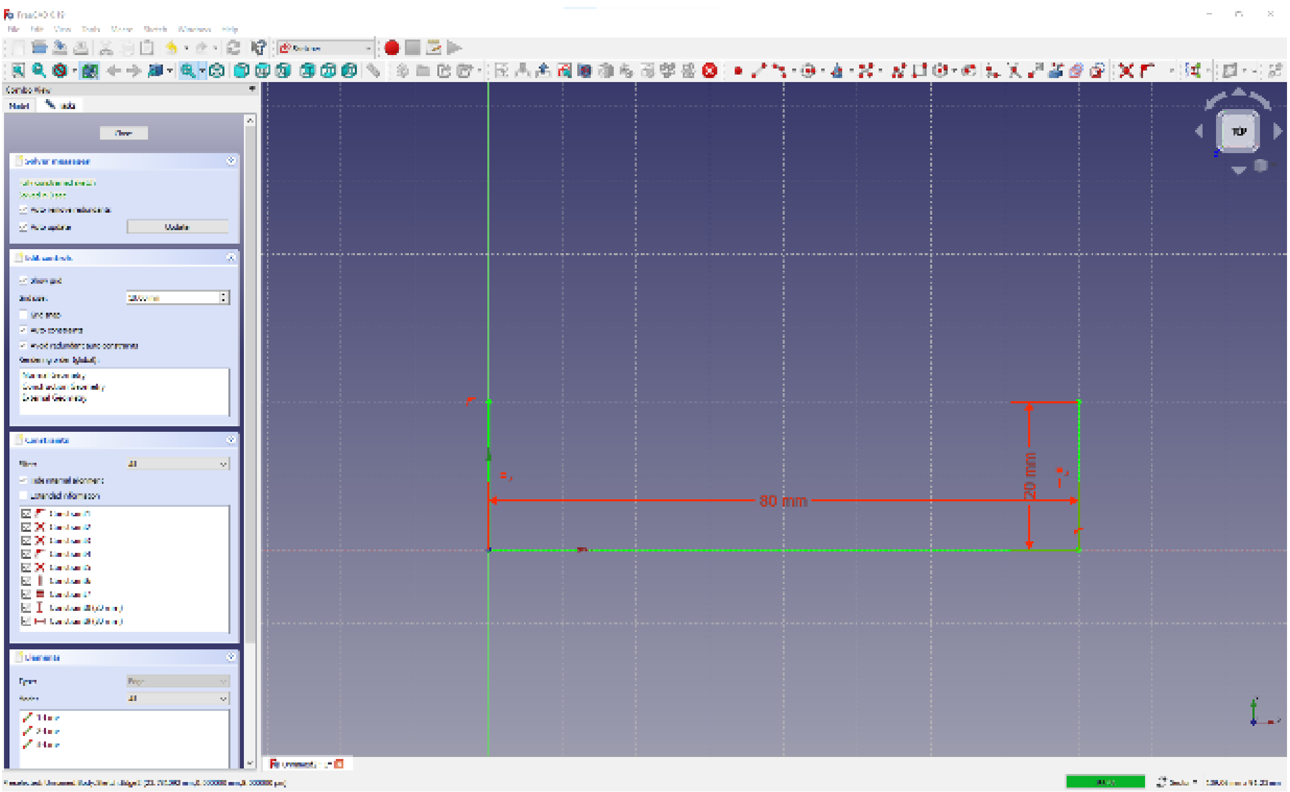
In the dialogue box that appears, type '20 mm', and then click OK. You should now see a red constraint label with '20 mm' in the viewer window – both vertical lines are now set to 20 mm. If you ever want to change this value, double-click on the constraint in the viewer window and the dialog box will reopen. You can click and drag the constraint around in the viewer window, and as you add more detail and constraints, you might do this to keep things clear.

Next, select the horizontal line and click the 'Fix a horizontal distance' tool

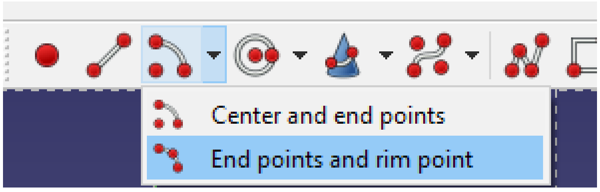


Set this length to 80 mm. You should now see that the sketch as it stands is fully constrained, the solver says there are no degrees of freedom, and that all lines are green.

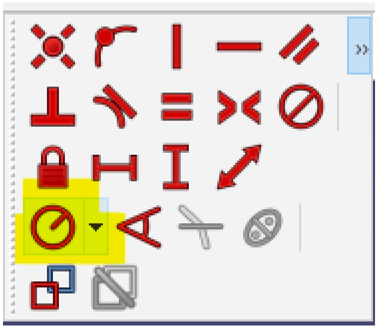




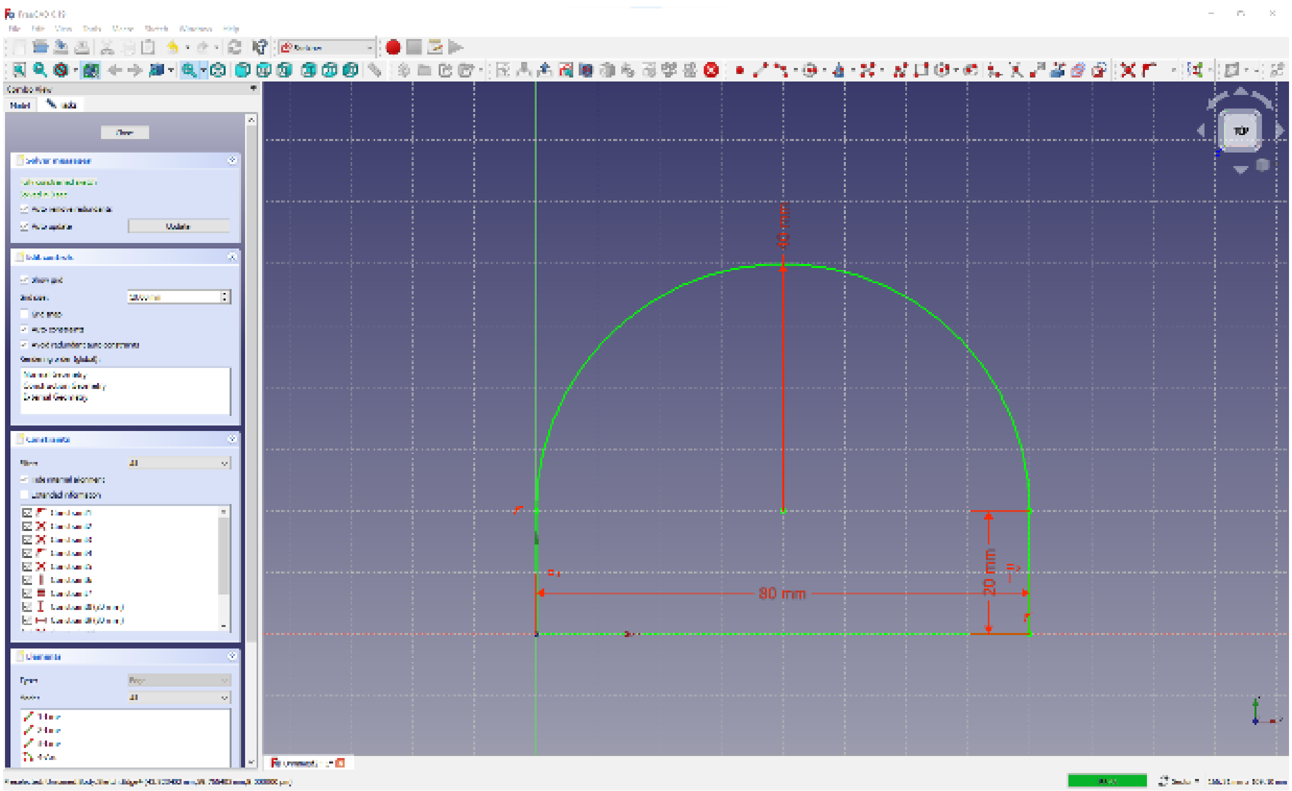
A bit sketchy
Click the 'Create an arc' tool icon, but click its drop-down menu and scroll to the 'End points and rim point' option.



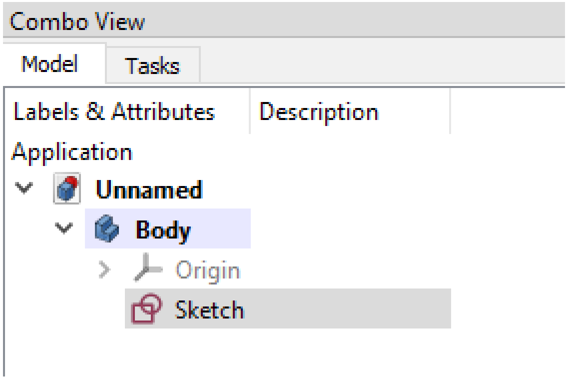
Hover over the uppermost point of the left-hand vertical line until the point becomes yellow, single left-click, and then move across to the top point of the right-hand vertical line and left-click again – this creates the end points of the arc. Next, move up and towards the centre of the two vertical lines to finish the arc roughly in the correct place. To constrain the arc – as the end points are already constrained – you just need to set the radius of the arc. Highlight the arc line and then click the 'Constrain an arc or a circle' tool icon



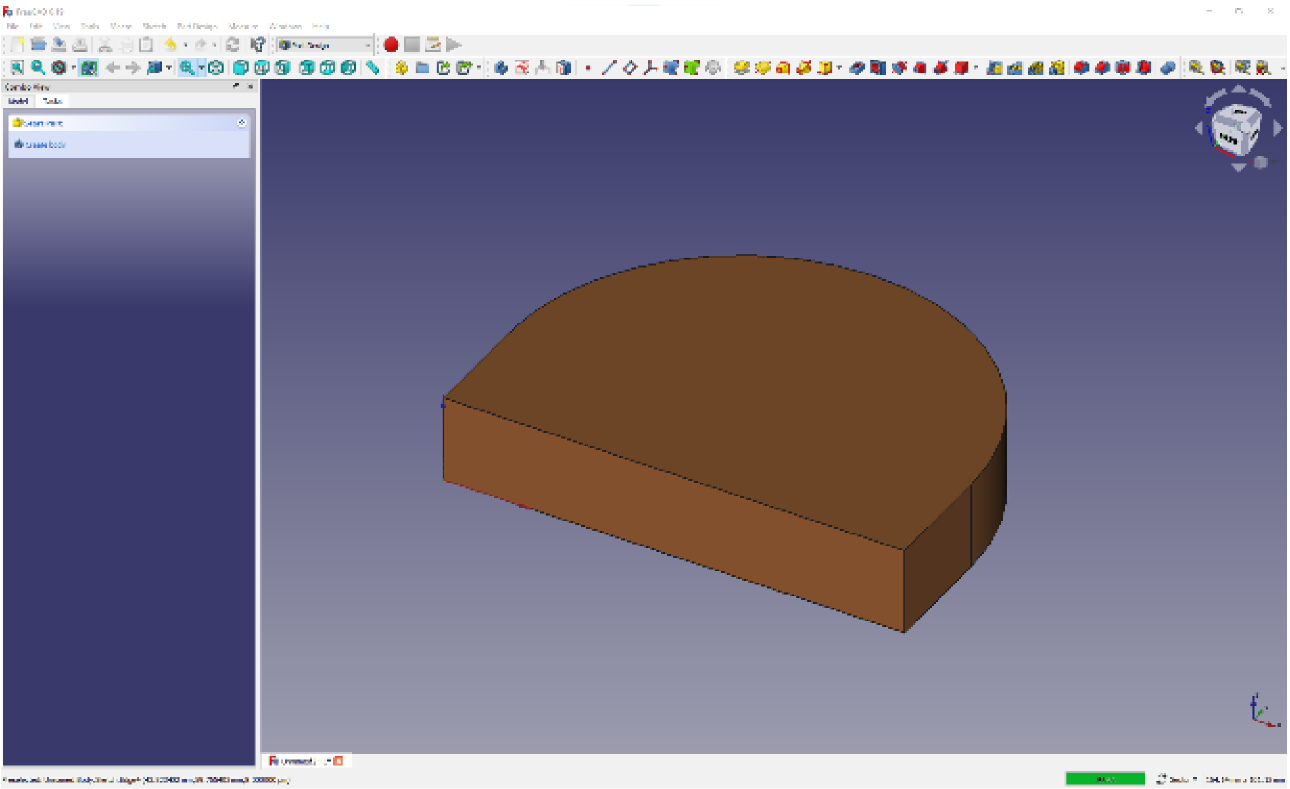
and in the dialog box, set the radius to 40 mm. You should now have a fully constrained sketch, with all lines turned green.



This sketch is the main outline of the bird feeder, so click 'close' on the sketch solver panel in the combo view window, and return to the Part design workbench. Make sure that the sketch you drew is highlighted in the file tree, then click the 'Tasks' tab just above the file tree view.

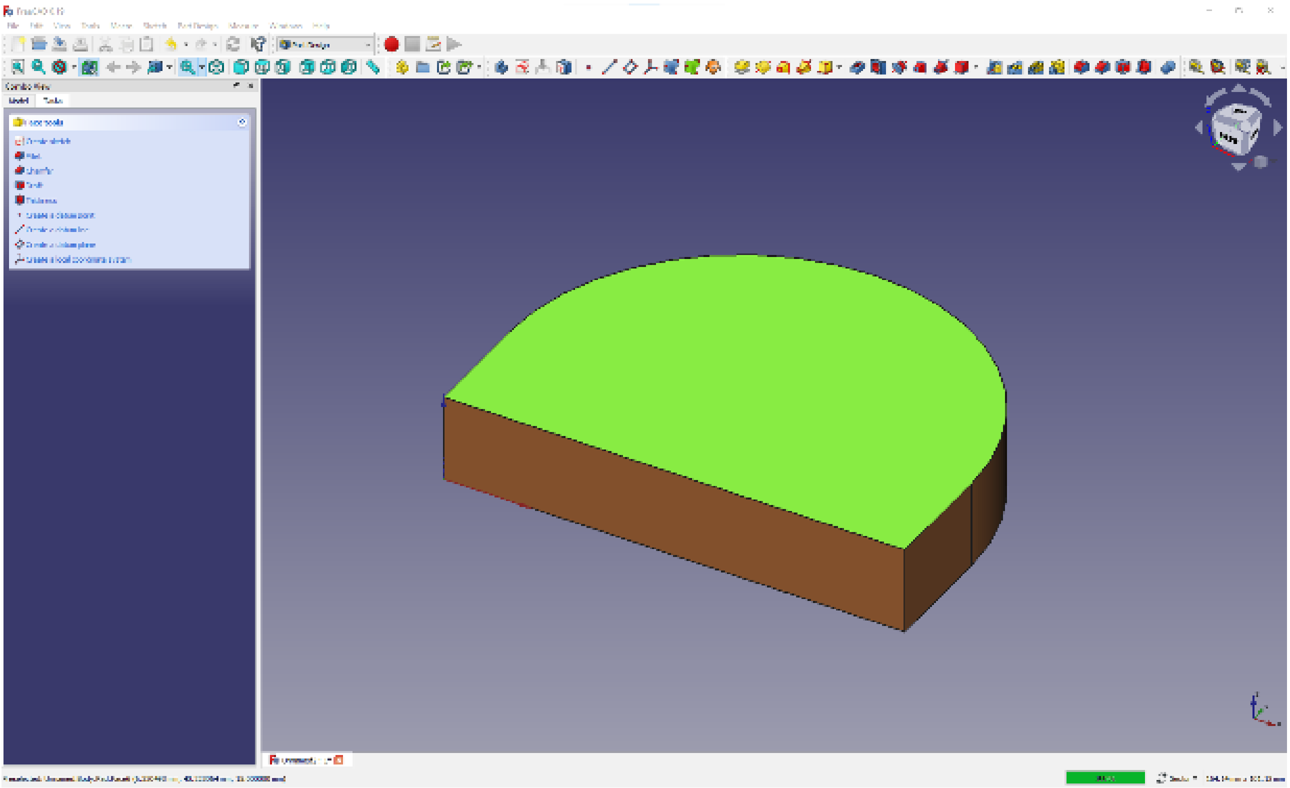


Click 'Pad', and then type '15 mm' in the dialog box to set the thickness of the pad/extrusion.



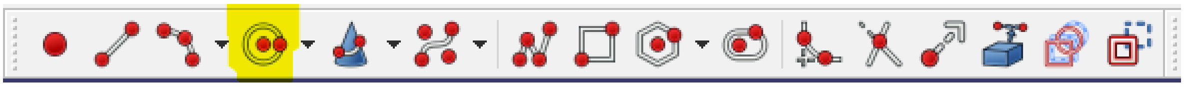
For the rest of this project, you are going to create more sketches, but these sketches are going to be drawn onto faces of the pad object you have just made. The first job is to make a pocket for the bird feed to be contained in. So, left-click in the viewer to select the upper face of the object.

Note: to fit the content on the screen you can zoom in or out holding the Crtl-key and rolling the mouse-wheel or click the ‘Fit the whole content on the screen’ button



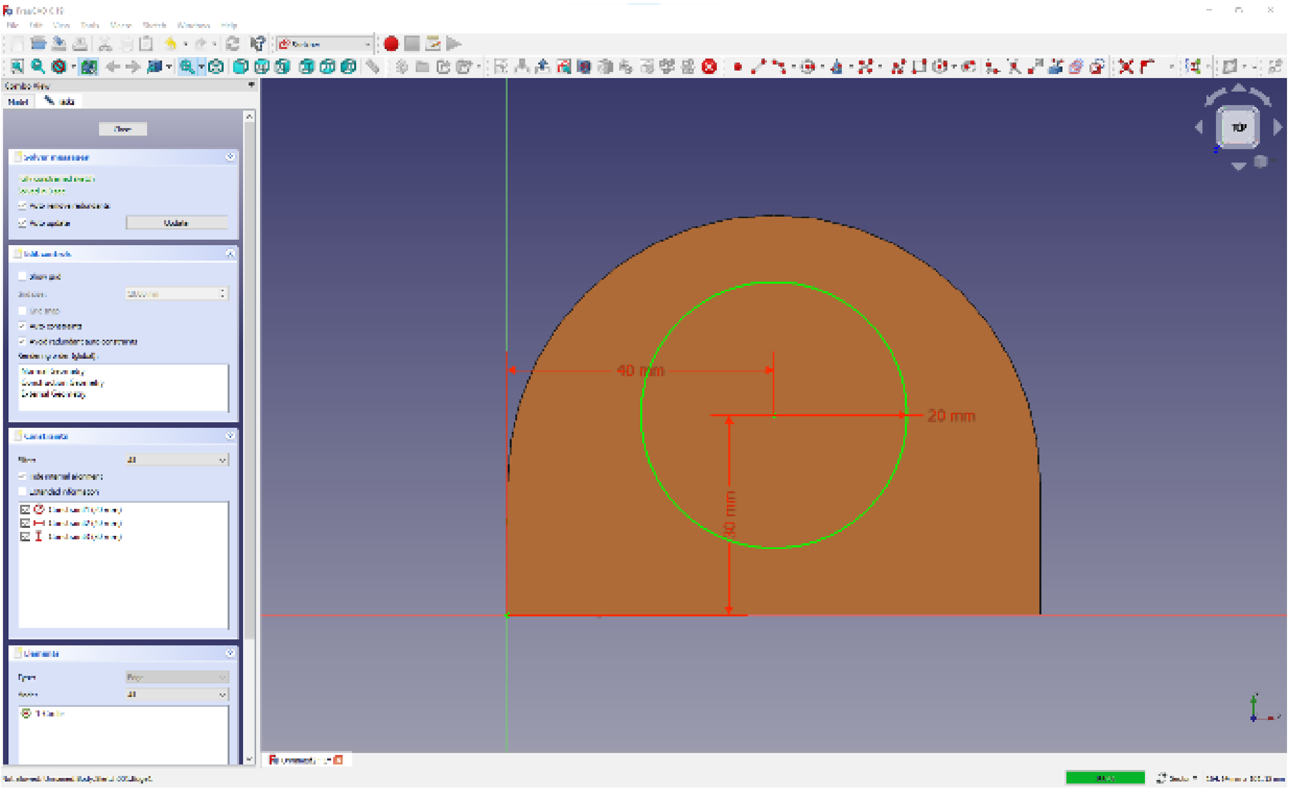
Then click the 'Create sketch' tool icon which should now bring you back into the Sketcher workbench with the face of the pad you selected in view.

Click the 'Create a circle' tool icon.

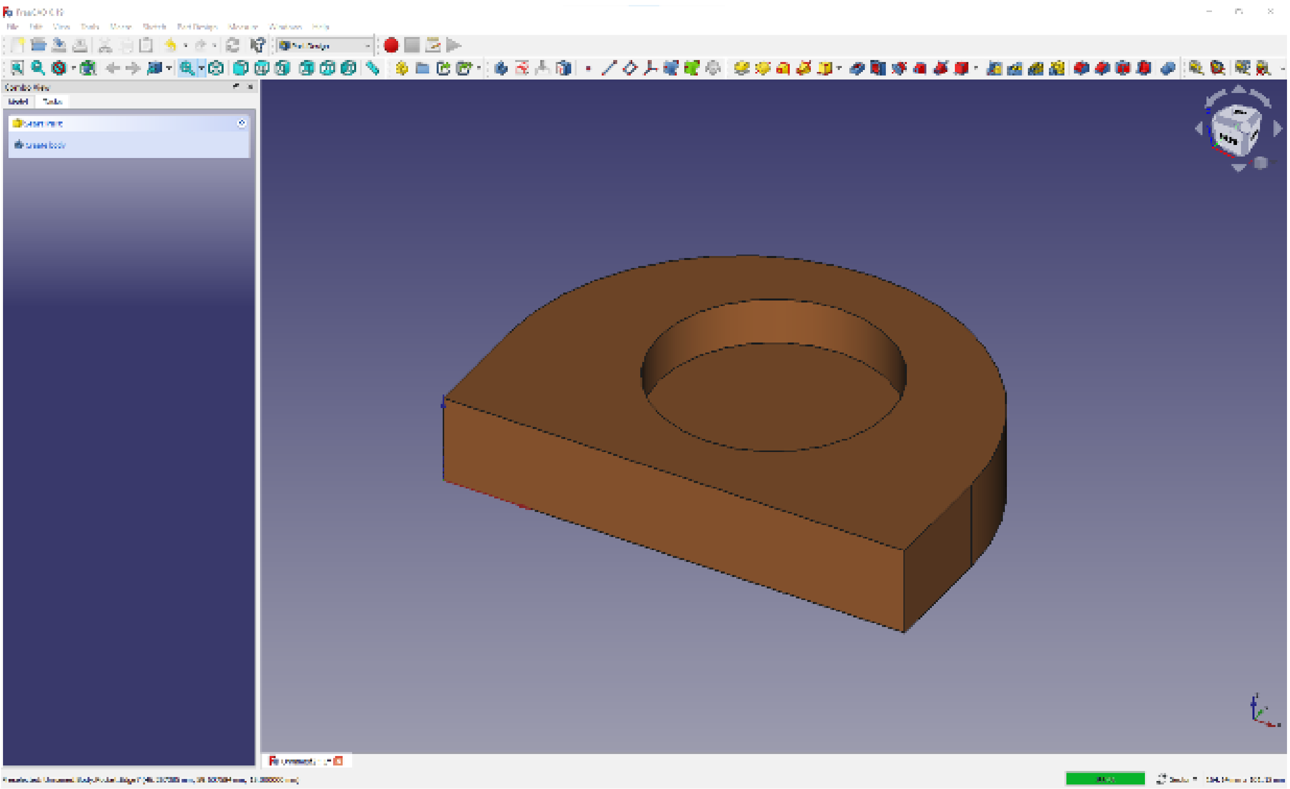


Left-click and drag and left-click again to draw a circle of any size anywhere in the Sketcher viewer. Use the 'Constrain an arc or circle' tool you used earlier to set the raduis to 20 mm. Exit the tool, pressing the Esc-key.

Next, click to select the centre point of the circle. To position the 20 mm circle, click the 'Horizontal distance constraint' tool used earlier and set the distance to the midpoint of the feeder, which is 40 mm. Next, set the vertical height of the centre point of the circle relative to the zero point at 30 mm. This should place it nicely centralised in the feeder design and should be fully constrained.



Close the sketch, and with this new sketch highlighted in the file tree, click 'Pocket' from the list on the 'Tasks' tab. You should now see that it’s cutting the circle as a pocket into the bird feeder design. Eet the depth of the pocket to 8 mm.

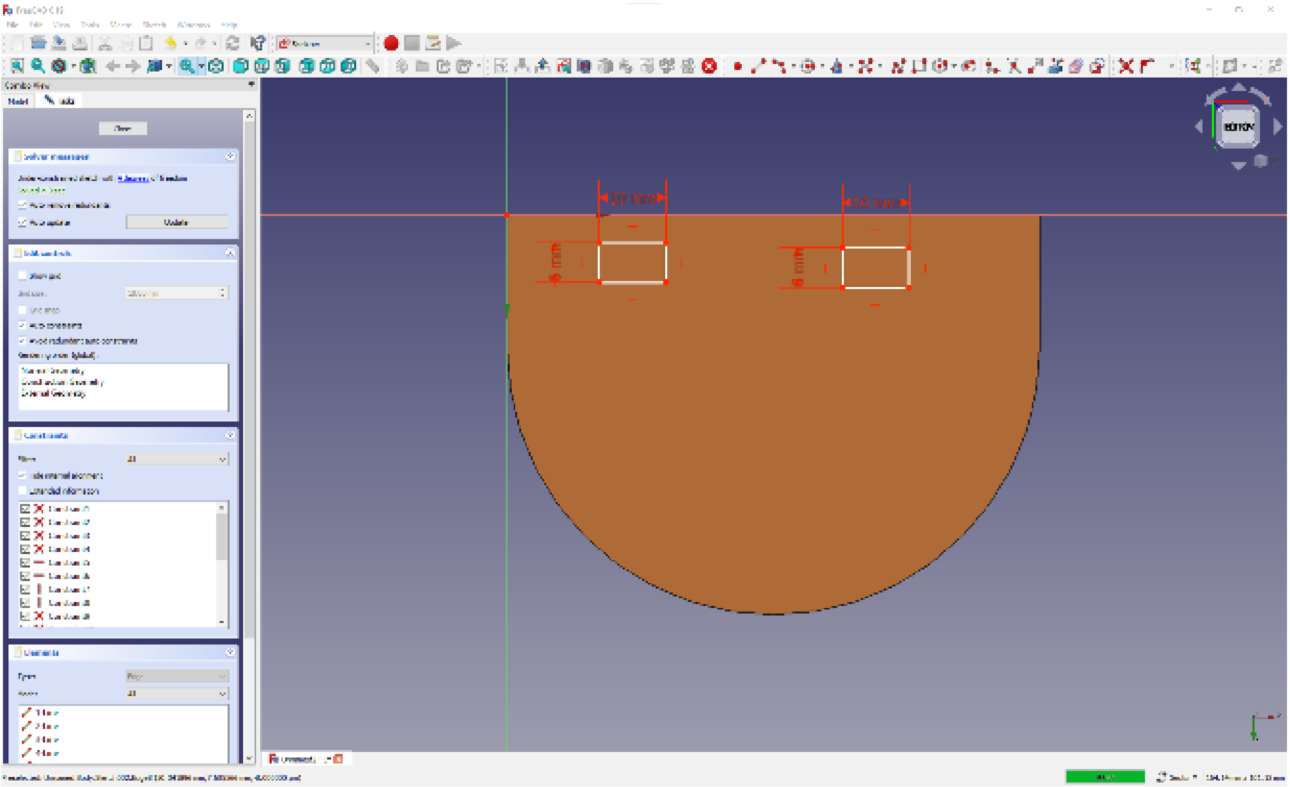


Quick Tip
You can use the constraint tools in different ways – you can select the sketch item and then click the constraint tool, or you can select the tool first and then click the item or items.

A sticking point

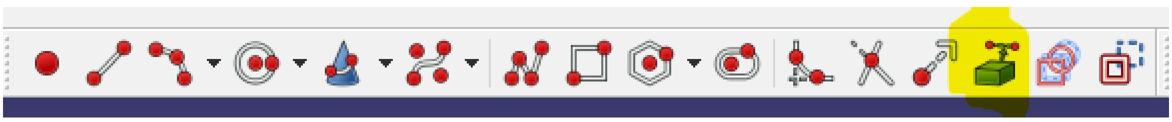
To attach the bird feeder, create a pocket and a slot which is used to fit some rubber window suckers. The arrangement of the slotted pocket could also allow the bird feeder to slot onto a couple of small screws.

In the Part design viewer, rotate the bird feeder and selected the bottom of the design opposite to the pocket just created. As before with this selected, click the 'Create a sketch' tool. Next, select the 'Create a rectangle' tool icon. Draw two rectangles of any size anywhere in the sketcher window. Conveniently the rectangle drawing tool constrains the vertical and horizontal lines, so all you have to add is the line lengths and the positions. Make the rectangles 10 mm by 6 mm.

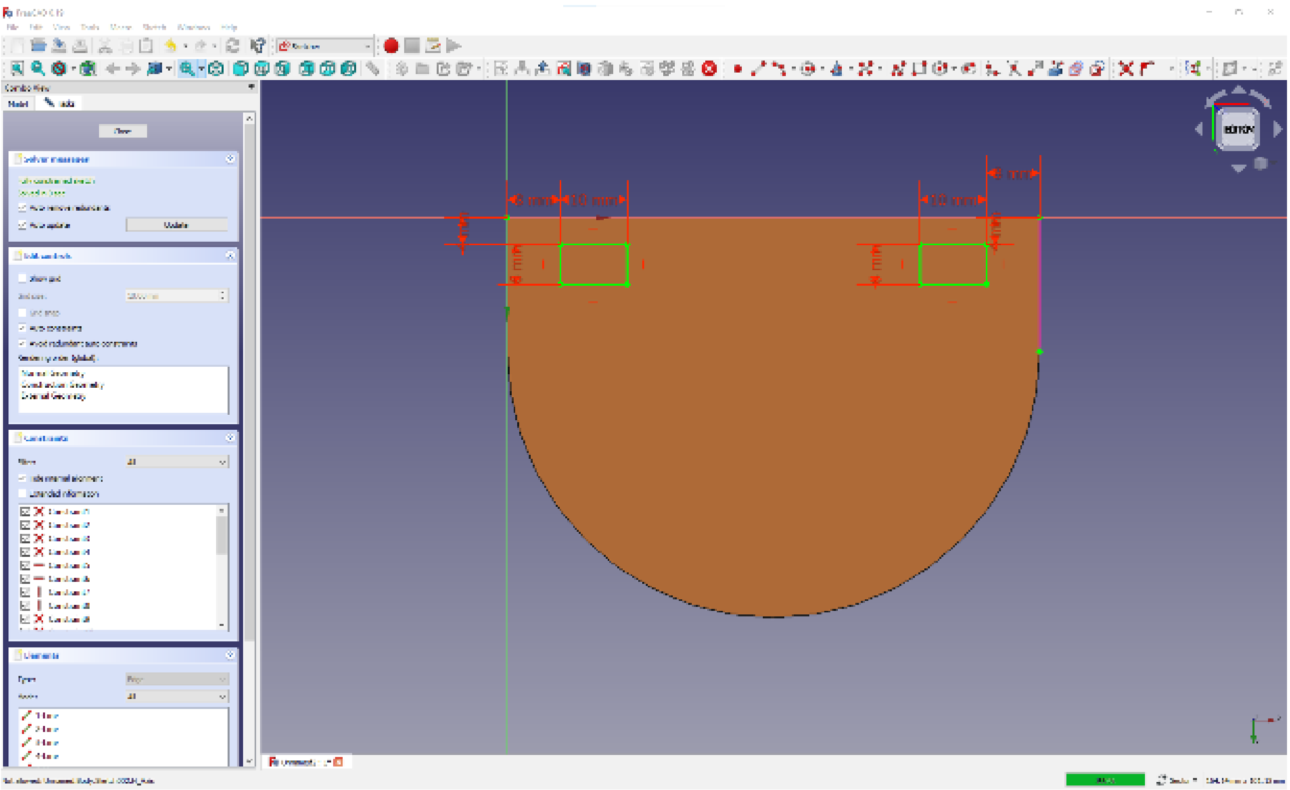


Then use the zero axis point to locate the closest rectangle to it, 8 mm across and 4 mm down from the straight edges of the design using the horizontal and vertical distance constraint tools and a corner point of the rectangle, as you have done with other items.

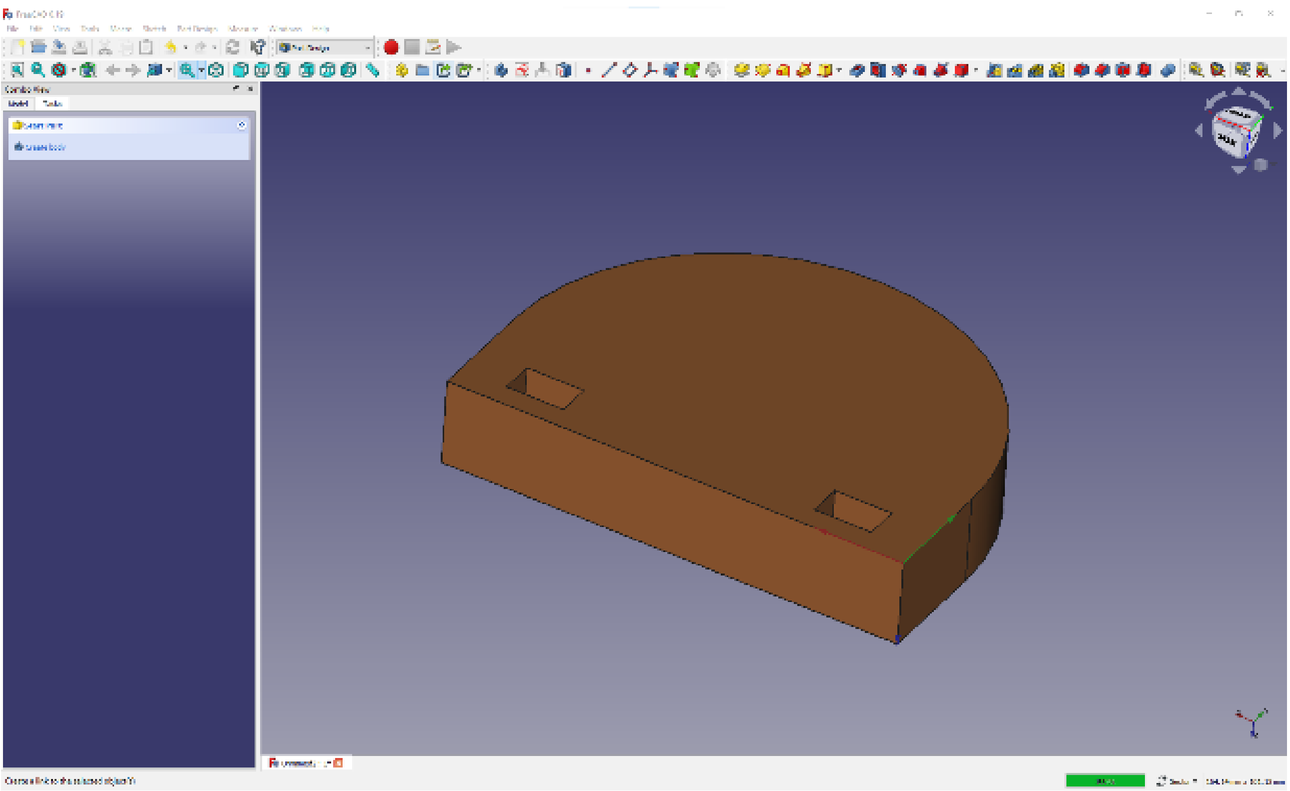
However, to position the other rectangle, use a clever little function. By clicking the 'Create an edge linked to an external geometry' tool, you can select and import lines from the underlying object which aren't in this sketch.



Hover over the 20 mm vertical line at the far edge of the bird feeder and select it, once highlighted. This now creates an edge and points at either end to which you can attach objects or constraints. This means that instead of calculating the position of the other rectangle from the XY zero origin point, you can now constrain this by saying it is 8 mm and 4 mm from the opposite corner of the object.

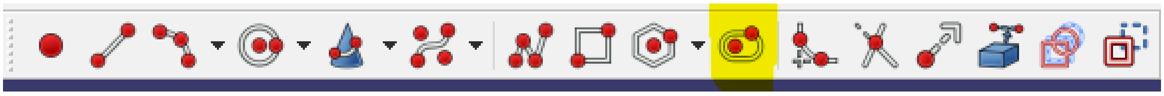


Having constrained the rectangles sketch, close the sketch and performed another pocket operation to a depth of 10 mm.

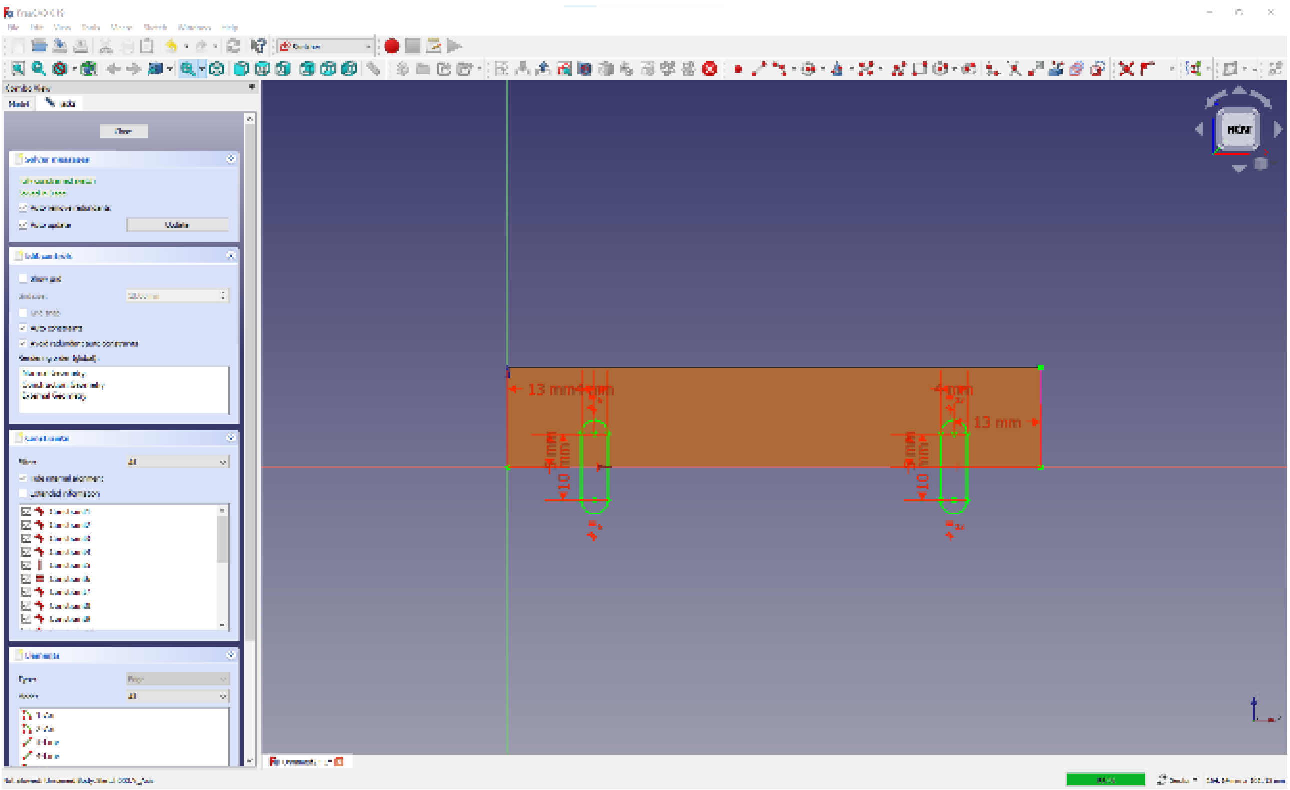


Next, cut slots that go into these pockets, so select the face of the back of the bird feeder object that need to be sketched on and create a new sketch.

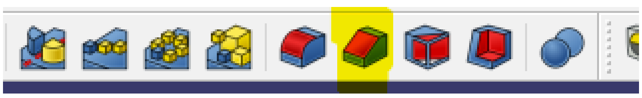
Finishing off
In this sketch use the 'create a slot in the sketch' too. This tool works similarly to the rectangle tool but creates a pill-shaped slot item.

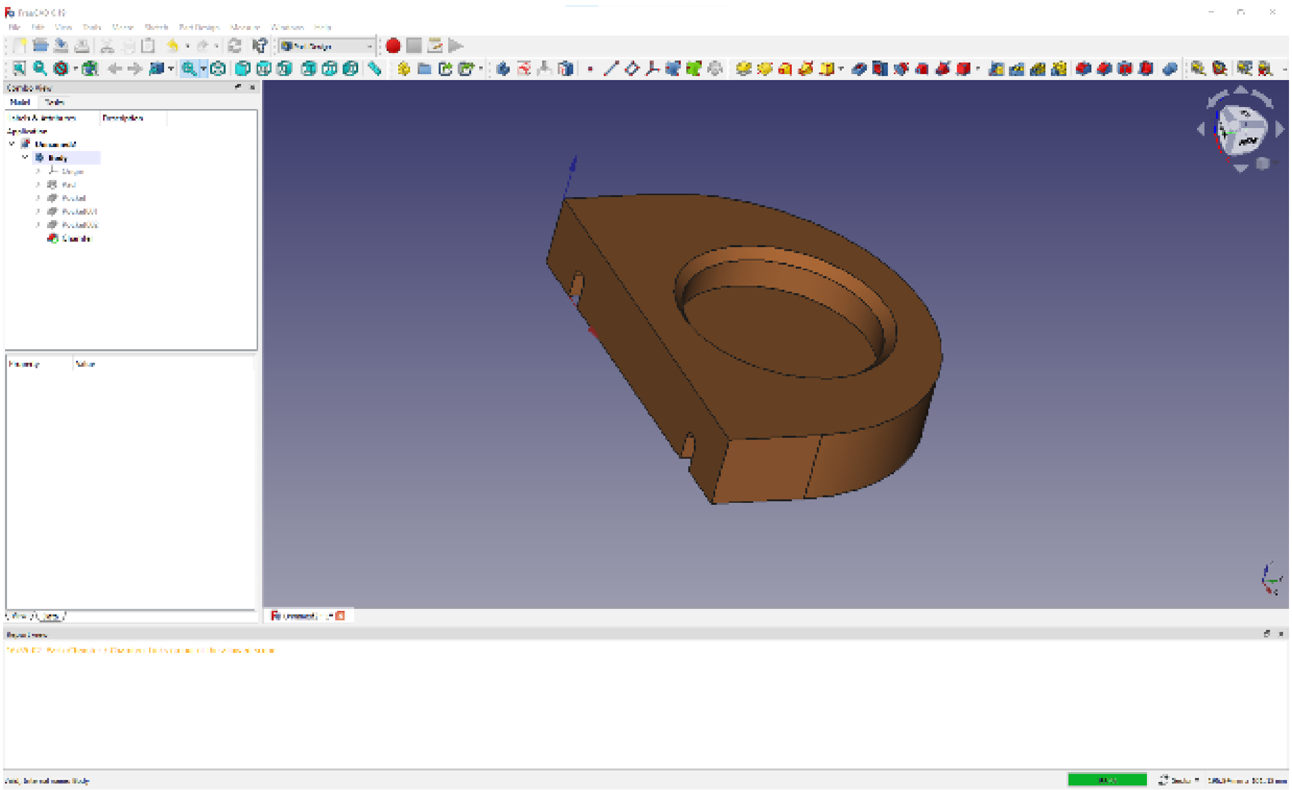


Similarly again, some of the edges and arcs are partially constrained. You can constrain these in multiple ways, but opt to select two opposite points on either side of the slot and constrain the horizontal distance between them to 4 mm. Then constrain just one of the vertical lines to 10 mm as both sides are considered an equality. With a fixed width and height, there is no need to constrain the arc, and so position the slot sketches in the correct places to coincide with the rectangular pocket. Use the point at the centre of the top end of each slot (the centre point of the radial arcs) to position the slots to coincide with the rectangle pockets. This should be 13mm from the edge. Remember to use the 'Create an edge linked to an external geometry' tool.
Position top centre point of the slots at 5 mm from the bottom.



Close the sketch and again performed a pocket, setting the depth to 5 mm, to cut into the rectangular pocket, but not deeper into the object.
Finally, Added a fillet to the edge of the top of the feeder and the feeder bowl by selecting the top face of the feeder and clicking the 'fillet tool' icon and setting the fillet radius to 2 mm.





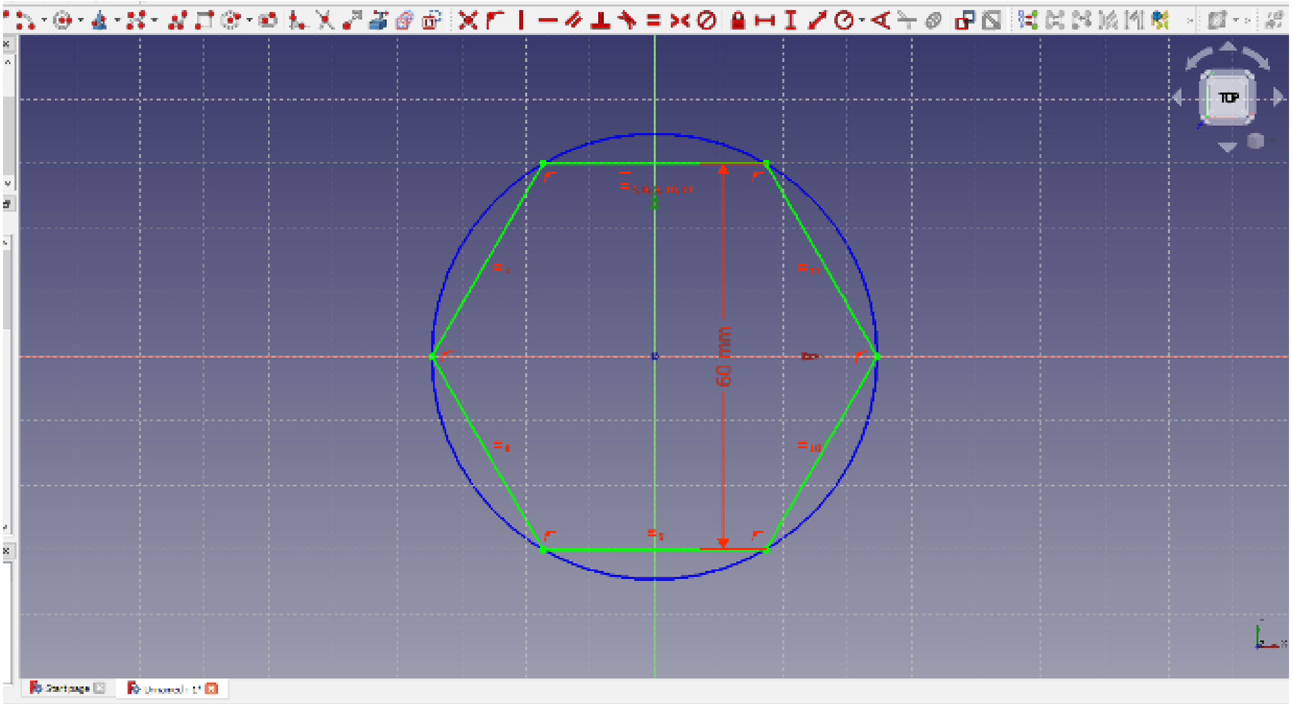
Hopefully, you now have a feel for how sketching and constraints work – the great advantage being that, if you want to change a dimension or position at a later date, you can click through to find the underlying sketch constraint, edit it, and the whole design relating to that change will be updated. It’s a powerful and useful way to work.

Quick Tip
You have mainly created sketches on the object and pocketed them, but, of course, you can create sketches and extrude them just as easily using the 'Pad' tool

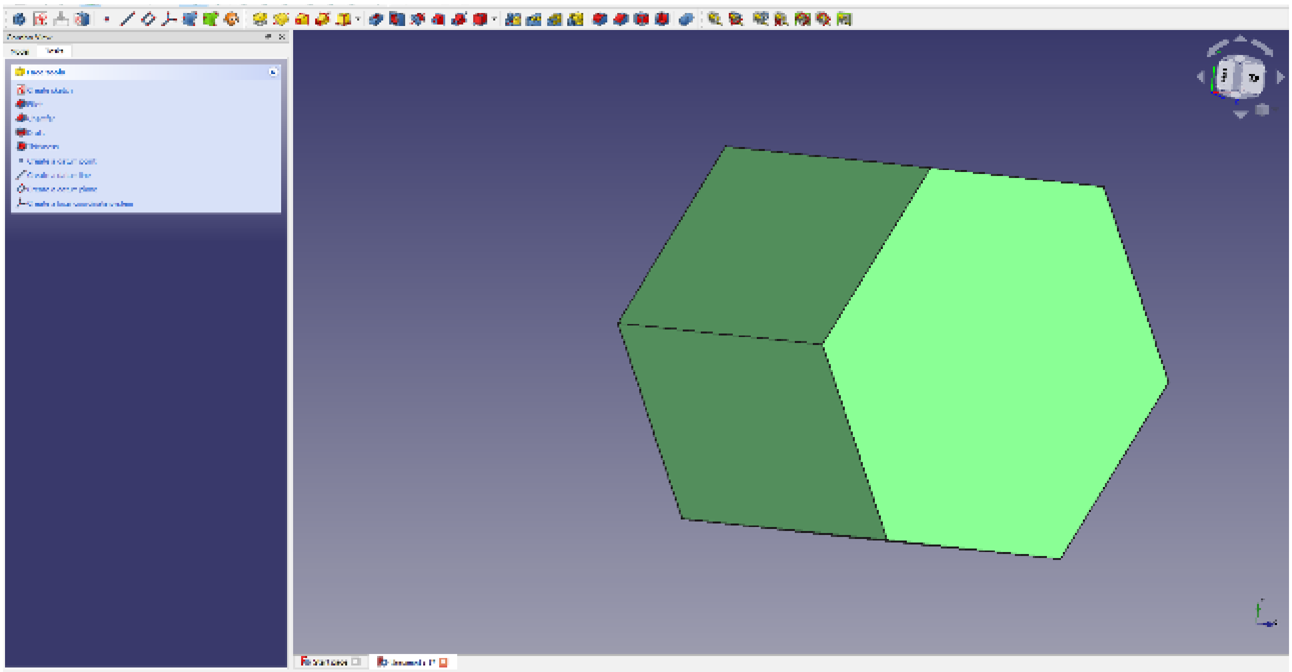
Thickness

For a quick example of using the Thickness' tool, begin in the Part Design workbench and create a new body. Create a new sketch in the body on the XY plane, then select the polygon tool to draw a hexagon. Start your hexagon on the 0,0 point of the axis so that it is positionally constrained. You should have a hexagon with two degrees of freedom.

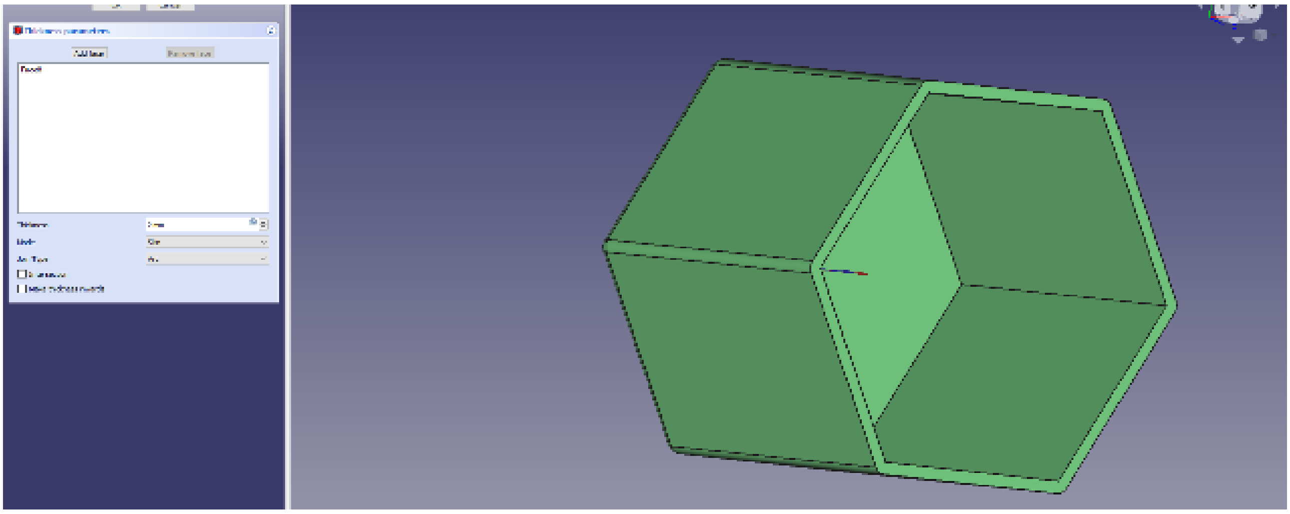
A simple way to constrain a hexagon is to make the uppermost line of the hexagon horizontal by selecting the line and clicking the 'create a horizontal constraint' tool. Next, you can select two nodes vertically above each other, and set a vertical distance to fully constrain the sketch



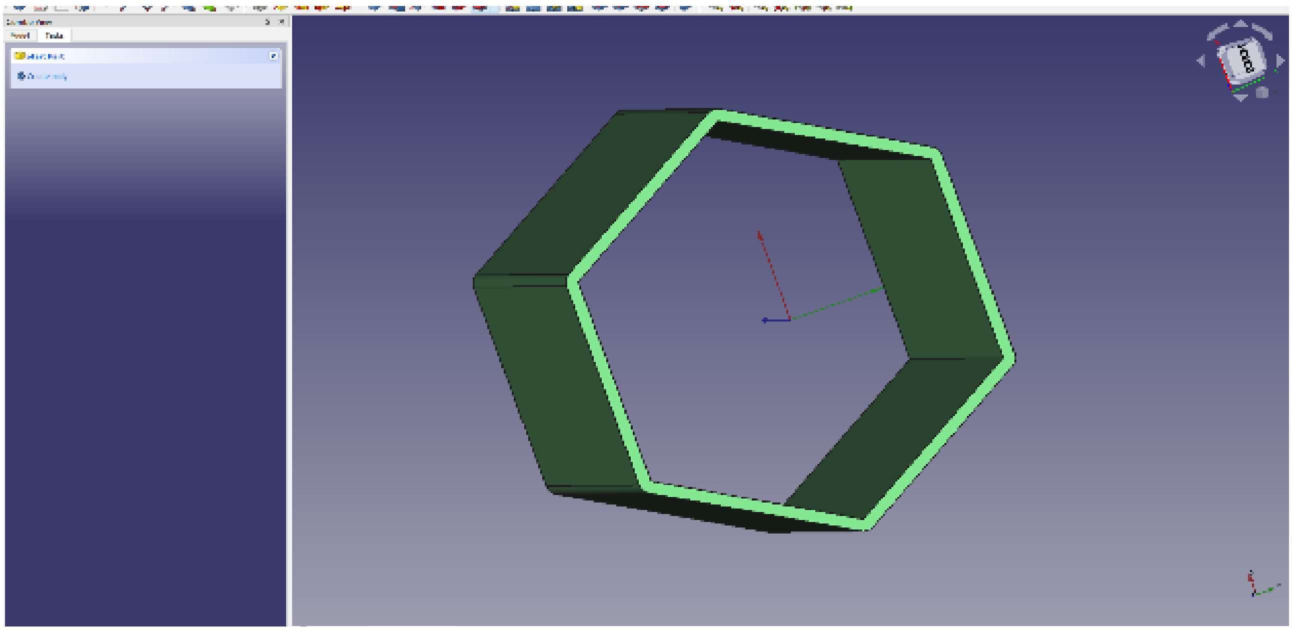
Close the sketch and then pad the sketch with 50 mm using the pad dialogue found on the tasks tab in the combo view window.



You are going to now use the thickness tool to hollow out your hexagonal extrusion. Select the upper face of the hexagonal extrusion and then click the 'Thickness' tool. You should see instantly that the part now appears as a hollowed version of itself. Make is 2 mm.
In the thickness tool dialog, you can see the selected face and change various parameters



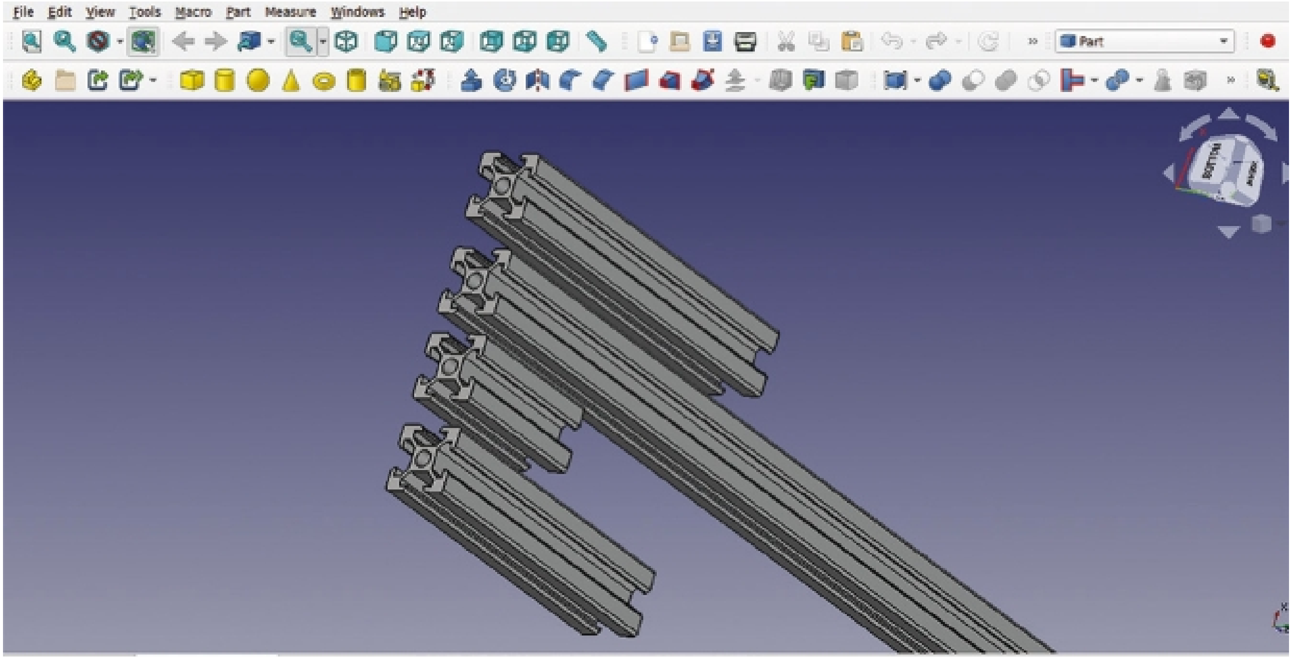
You can, of course, increase and decrease the thickness that is created. By default, the thickness dialog adds the thickness to the outside of the underlying geometry, so if you set the hexagon vertical constraint to 60 mm you now will have an object that is 62 mm (60 mm + 2 mm thicknesses). If you want to create an object that matches the external dimensions of the underlying geometry, click the 'Make thickness inwards' box. You can also swap between Arc and Intersection, which essentially toggles between creating filleted edges or sharp edges on the thickness.
Finally, if you want to apply a thickness but create an object which is more of a pipe than a bowl, you can click the 'Add Face' button. In the preview window, the original selected face reappears highlighted and you can select the opposite face. You should now have a hexagonal object with a wall thickness with both ends open.



Other options
The thickness tool is an excellent way to quickly create hollow geometries, but it does have some limitations. It can fail to work with more complex shapes. Things like curvy cones can be its nemesis. Later on we'll look at 'Lofting' tools, which enable us to create hollow parts with curved walls and more.

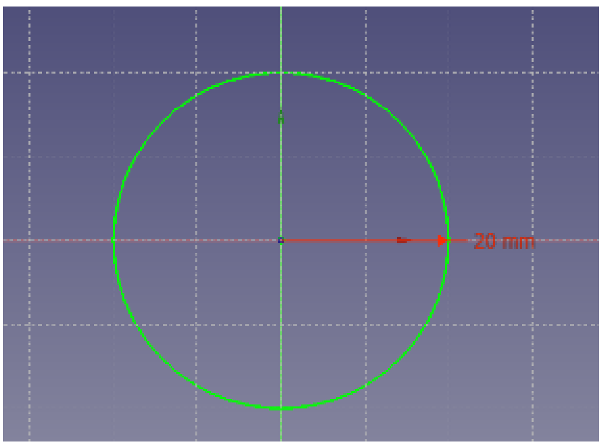
Let's get parametric!

You looked at making multiple 'simple copies' of parts using the 'part' menu on the part workbench. You can also use that approach to create simple copies of bodies that you have designed in Part Design. Sometimes you might want to make multiple similar parts, but with one or more dimensions changed – for example, if you were drawing a model of a 3D printer that used the common 2020 aluminium extrusion.

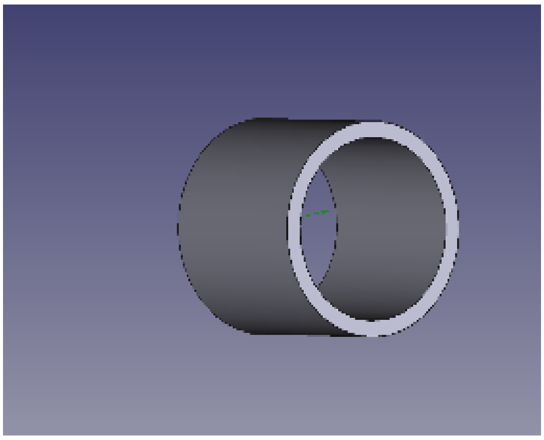


One way you could make this easier is to draw an aluminium extrusion profile sketch, and then pad or extrude it multiple times to multiple lengths. As a workflow, you might choose to use the Part Design workbench to create a body for your original extrusion and then create multiple simple copies on the part workbench.
This is a very simple example that introduces the idea of working parametrically. Parametric work in CAD is where you can change the model geometry by adjusting parameters such as dimensions. Whilst only being one parameter, you can use this mini project to learn about using a spreadsheet to store and alter parameters for objects and sketches.

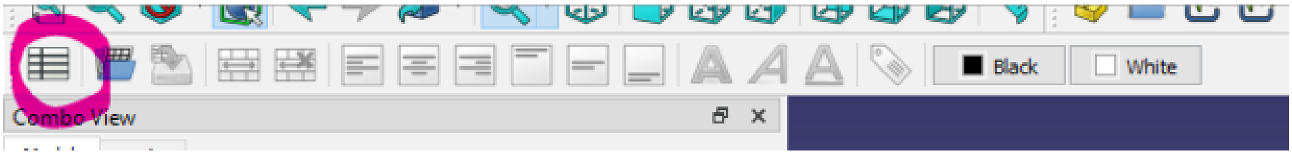
you are going to work with a simple design for a tube and created a new project. On the Part Design workbench you select to create a new body and then to create a sketch in the XY plane. you drew a shape of circle, put a constraint on it of 20 mm.



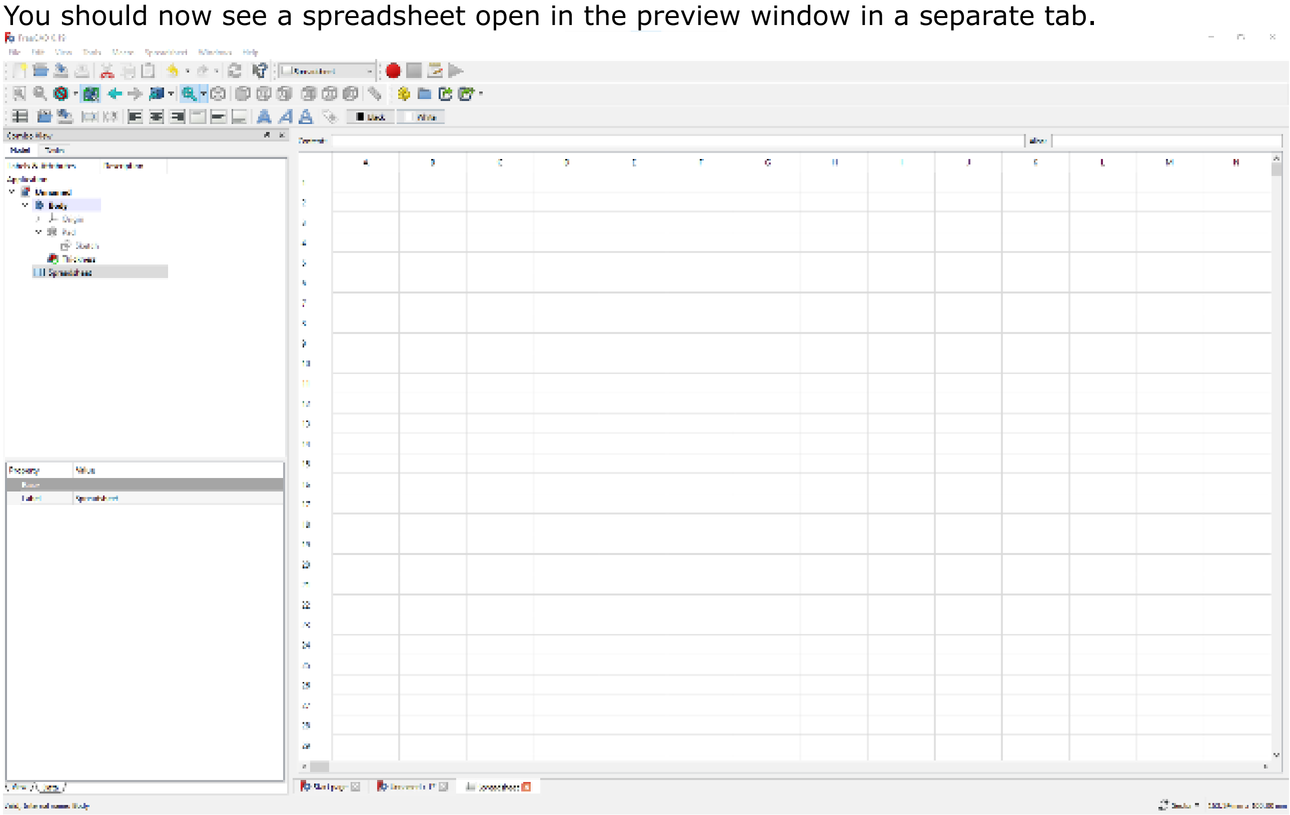
Having constrained the sketch, you now pad it to 50 mm, make an inner tichnes of 3 mm and remove the bottom.



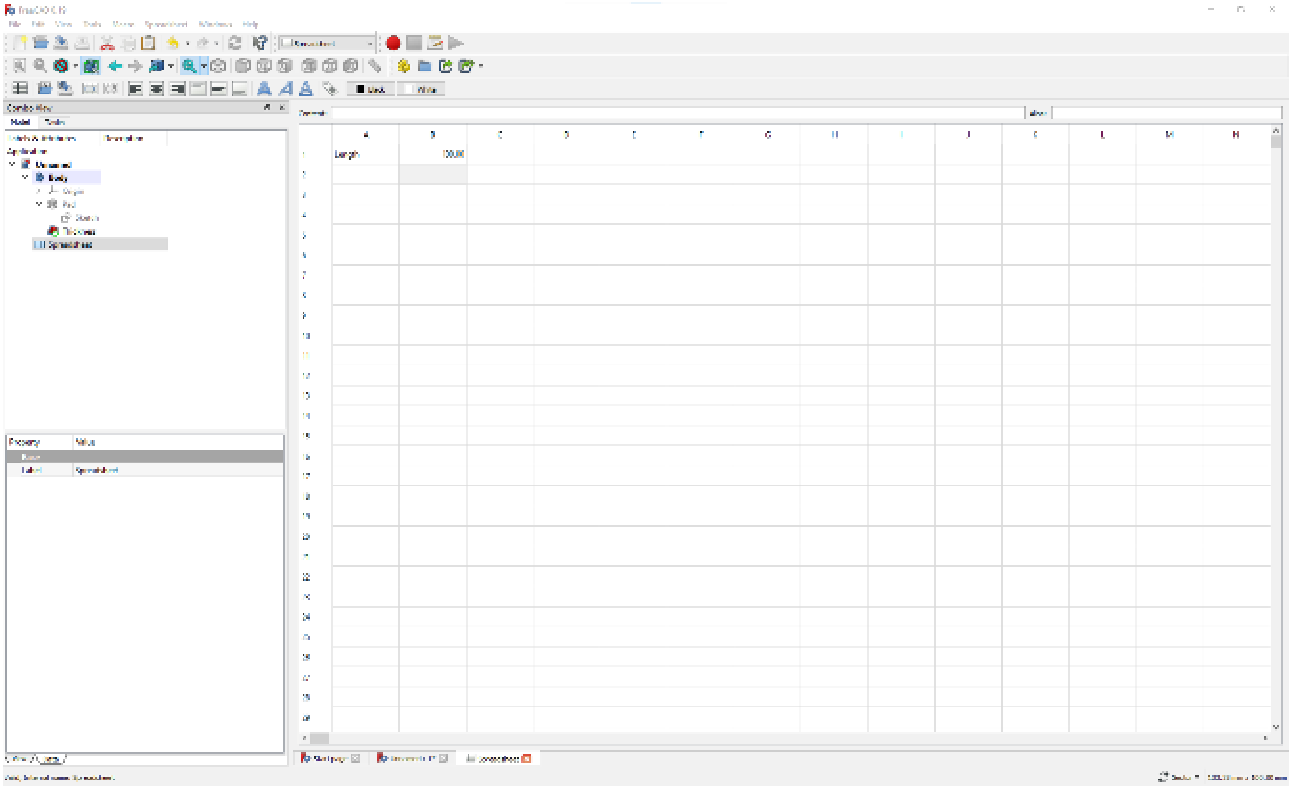
you could now, of course, just set a pad length of extrusion and then create a simple copy on the part workbench to create different length parts, but you want to learn how to use the spreadsheet function to further explore parametric work.
Let's move to the spreadsheet workbench using the workbench drop-down menu, and click the ‘create a new spreadsheet’ icon.



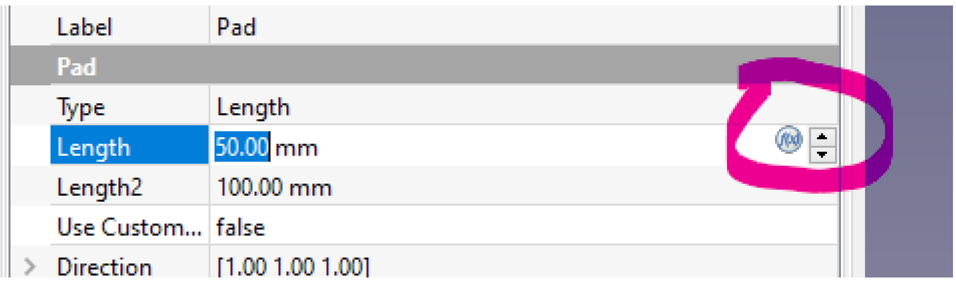
You should now see a spreadsheet open in the preview window in a separate tab.



This is very handy, as it allows quick switching between the model and the spreadsheet. If you close the spreadsheet tab, the spreadsheet is still an item in the file tree view and you can double-click to open it. The spreadsheet functions are very capable, and similar to standard spreadsheets you may have used in office software. For this simple task, you are going to write the label 'Length' in cell A1, and then in cell B1 you can type a length you want your extrusion to be. When you type a dimension into a constraint in the sketcher, it interprets that number as the units you use throughout FreeCAD, which can be set in the preferences menu. In your case, this is millimetres and will be the case for your spreadsheet values, so you don't need to define your input units in the spreadsheet. Set cell B1 to 100



Click back onto the preview tab with the model in it. Reopen the pad by double-clicking on the pad in the file tree. In the input box for 'length' you should see the current pad length value, but you should also see a blue circle icon on the right-hand side of the box . Click this to open the formula editor.

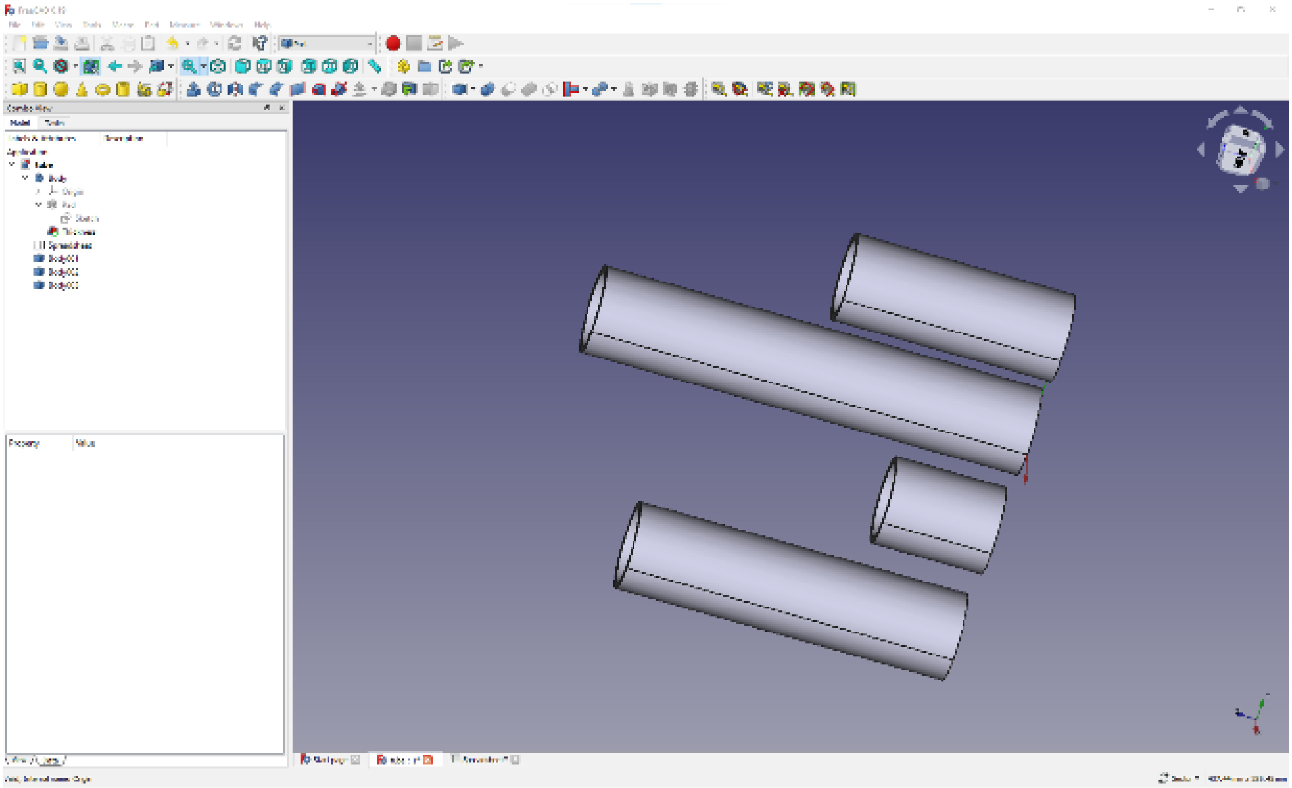


In this input box, now start to type 'Spreadsheet'. You should find that it automatically suggests the word spreadsheet as you start typing. Click the suggested 'Spreadsheet' from the drop-down and it should insert the word spreadsheet followed by a full stop. After the full stop, you need to input the cell location of the cell from which you want it to take its value. If you type 'B', it again should automatically suggest 'B1' as a value. This autosuggestion is useful when you come to projects with numerous cells of data in a spreadsheet, as it will only suggest cells that have data in them. Click the 'B1' so that the value in the formula editor reads 'Spreadsheet.B1'.

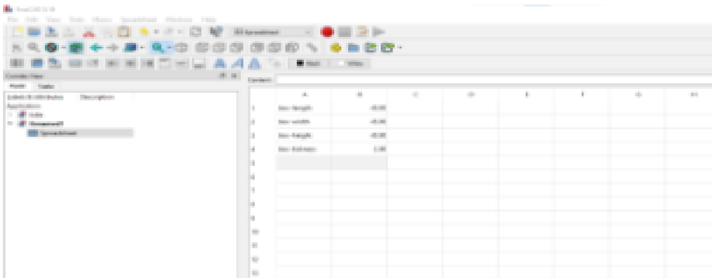
Pad	
Type	Length
Length	100.00 mm (Spreadsheet.B1)
Length2	100.00 mm
Use Custom...	false

Returning to the preview tab with the model in, you should see that the extrusion body is now padded or extruded to the 100 mm length. You can now create multiple simple copies from the body using the 'create a simple copy' tool in the 'part' drop-down on the Part Workbench, changing the length of the original body copy in the spreadsheet.

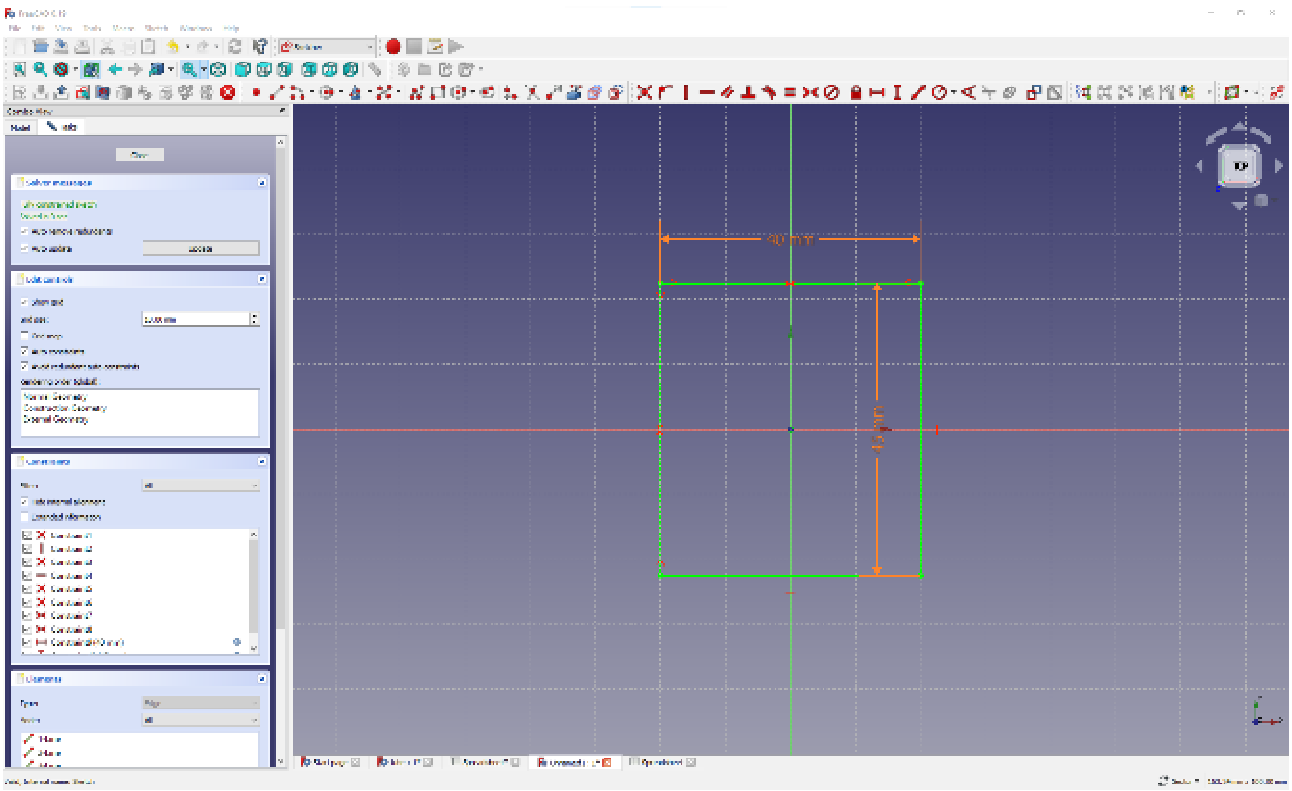
So now, you can change the length in the spreadsheet so the length of the (original) tube changes and then create a copy of it to have a new tube.



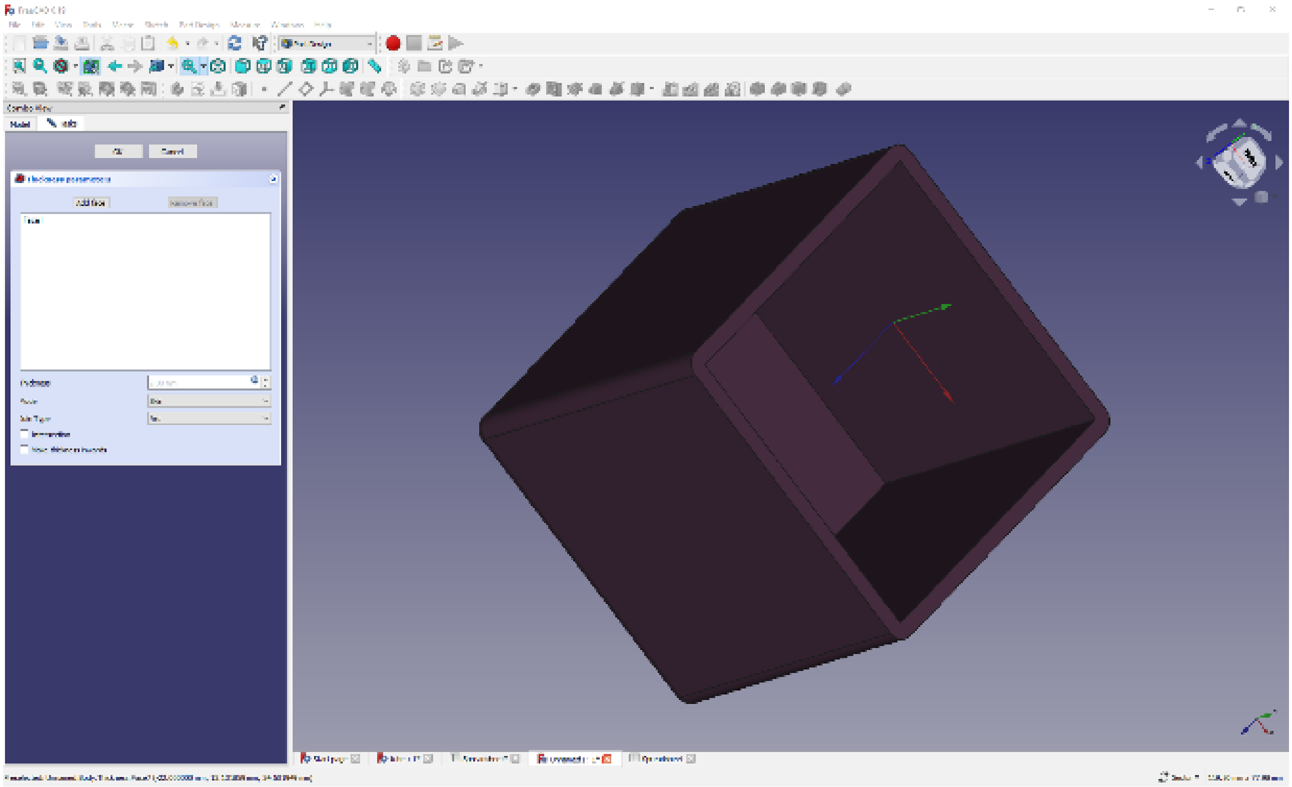
Let's combine your new skills!
Having explored using both spreadsheets and the thickness tool, let’s combine your new skills and create an automatic parametric box generator. This simple project makes it easy to create a box and lid of any size that you could 3D-print or CNC-machine.
Let’s begin by creating a new project. In the new project, go to the spreadsheet workbench and create a new spreadsheet. For the main body of your box, you are going to need four parameters: the box-length (40.00), box-width (45.00), box-height (50.00), and the box-thickness (2.00). Create labels for these parameters in cells, and in adjacent cells input some initial values: 40, 45, 50, 2



Next, jump to the Part Design workbench and create a body and a sketch in the XY plane. you are going to draw a rectangle around the zero point and then, at first, constrain it to always be centred around the origin point of the sketch. Draw the rectangle and then click the ‘create a symmetry’ constraint tool. Use the symmetry constraint tool to select the upper right-hand node, the vertical Y axis line and the upper left-hand node. Repeat this for the upper right, he X axis line and lower right nodes. Now the rectangle should always be centred around the 0,0 co-ordinates, regardless of its size.
Next, let’s add a constraint for the box length by selecting the top line and clicking the ‘Fix the horizontal distance’ tool. In the input box, click the blue circle formula editor button and insert ‘Spreadsheet’, followed by the cell location for the box length, which in your case was B1. Repeat this for the vertical line to set the box width, B2, and the sketch should now appear fully constrained.

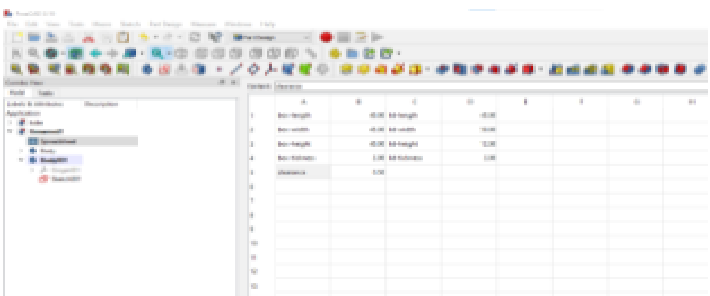


Close the sketch and perform a 'pad' task to extrude the box; set the pad dimension to receive the input from the box height cell, B3, in the spreadsheet. Next, you will click the upper surface of your box and use the thickness tool you explored earlier to hollow out your box. To set the thickness, you are going to link the thickness value in your spreadsheet, B4. you make it an outwards thickness. All being well, you should now have a hollow box in the preview window.

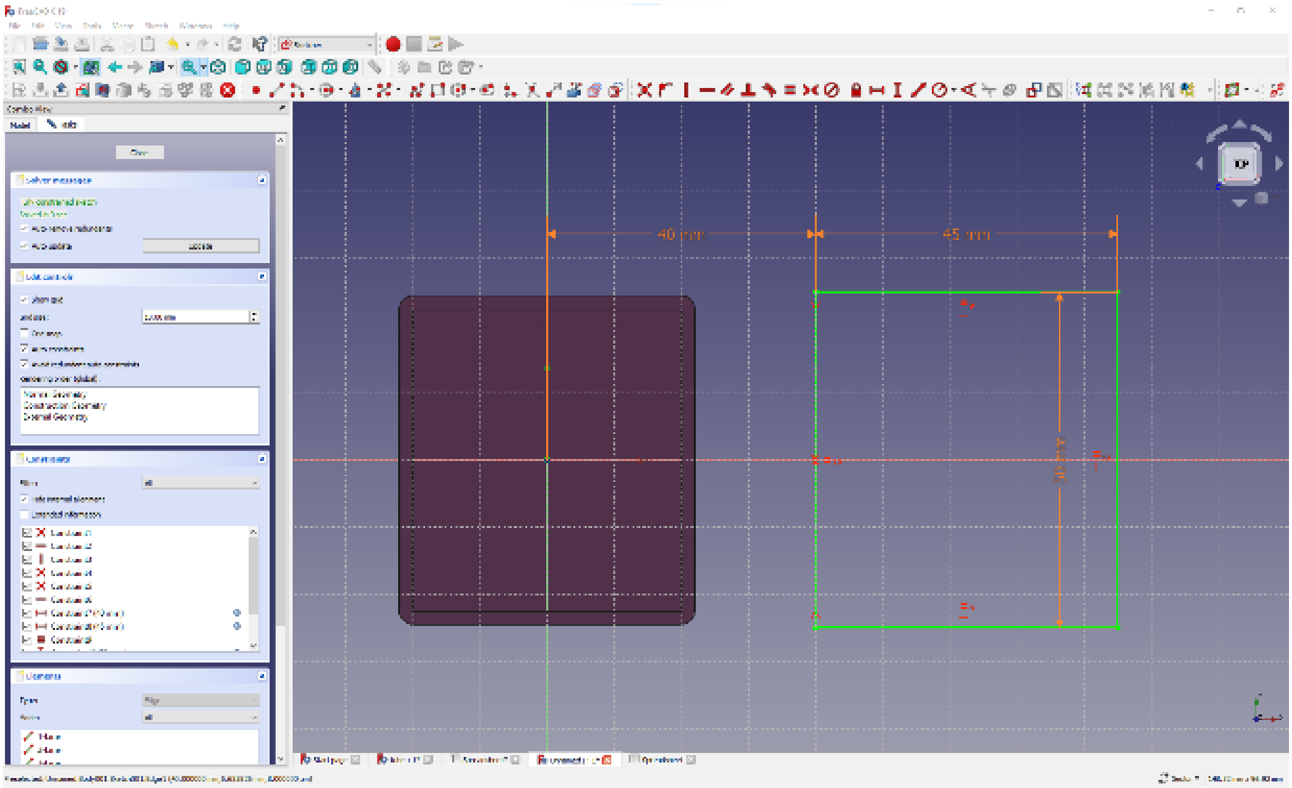


To make a lid, you are going to do the same process as you did to make the box section, but you are going to use some simple formulae so that the lid is generated to fit whatever dimensions are of the box created.

So go the the spreadsheet and add the lid length,width and height. But first all the clearance, being the space between the lid-edge and the box-edge to easy putting the lid on the box. Make that clearance 0.5 mm. So the length of the lid becomes now the length of the box + twice the thickness of the box + twice the clearance: =B1 + 2 * B4 + 2 * B5. Do identical for the width becoming = B2 + 2 * B4 + 2 * B5. The lid height can be any length, I choose 12. Thickness is the same as the bow: = B4

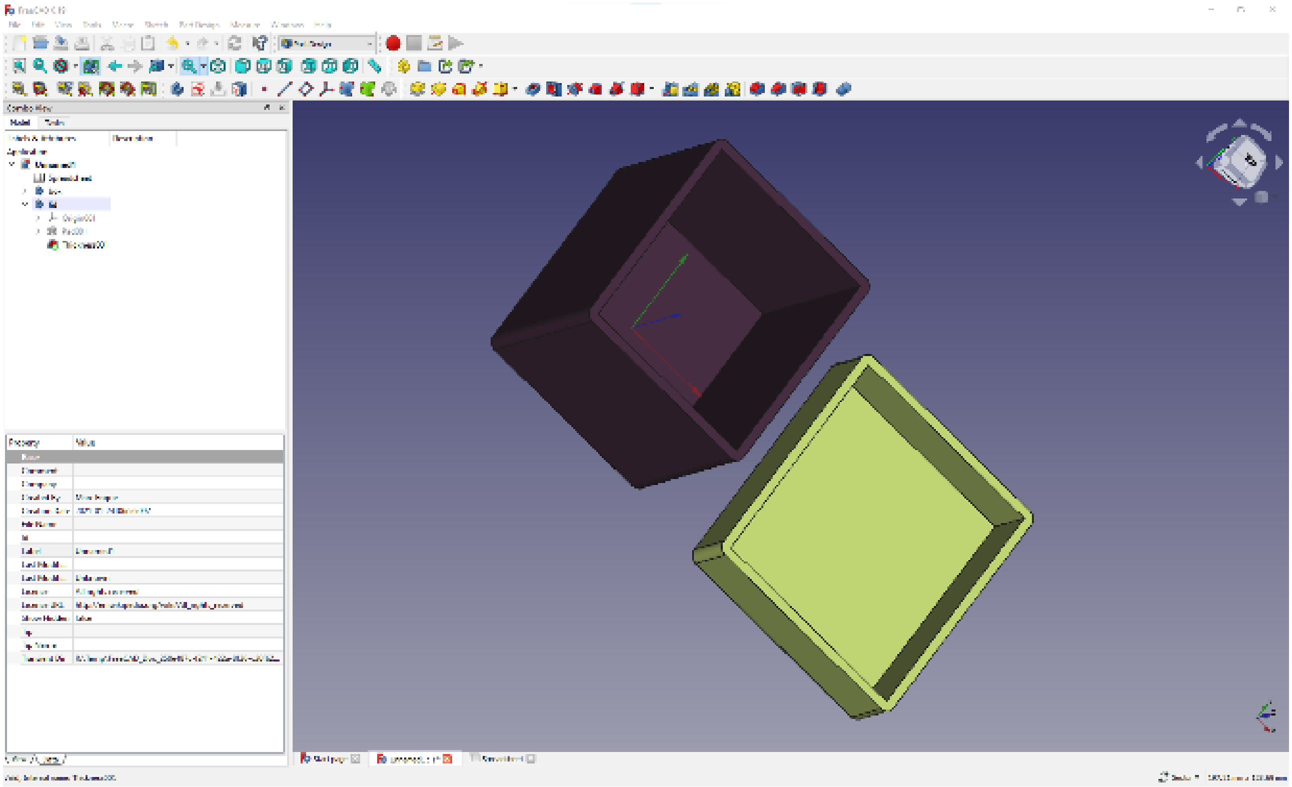


On the Part Design workbench, click to create a new sketch on the XY plane. You don’t need to import any geometry from your first sketch, but let’s draw another rectangle and place it along the x axis so it isn’t on top of your box base. To positionally constrain this lid rectangle, select the upper left corner node and the sketch origin point at 0,0 and set a horizontal distance constraint. For the value of this constraint, you have linked the spreadsheet cell that holds the box length value, B1. This means that the lid will always be away from the box with a gap in between them.
Next, create a symmetry constraint against the x axis line. Now add constraints for length and width based on the spreadsheet values for length D1 and width, D2.



Now pad the lid and add tickness outwards.

You constrained the length and width of the lid sketch by adding constraints linked to your calculated dimensions in the spreadsheet. You then performed a pad using the lid_depth value and, in turn, applied a thickness to the lid using the thickness value in the spreadsheet. You now have an automatic box generator at your disposal! Simply change the parameters in the spreadsheet and it will automatically generate your box with a fitting lid.



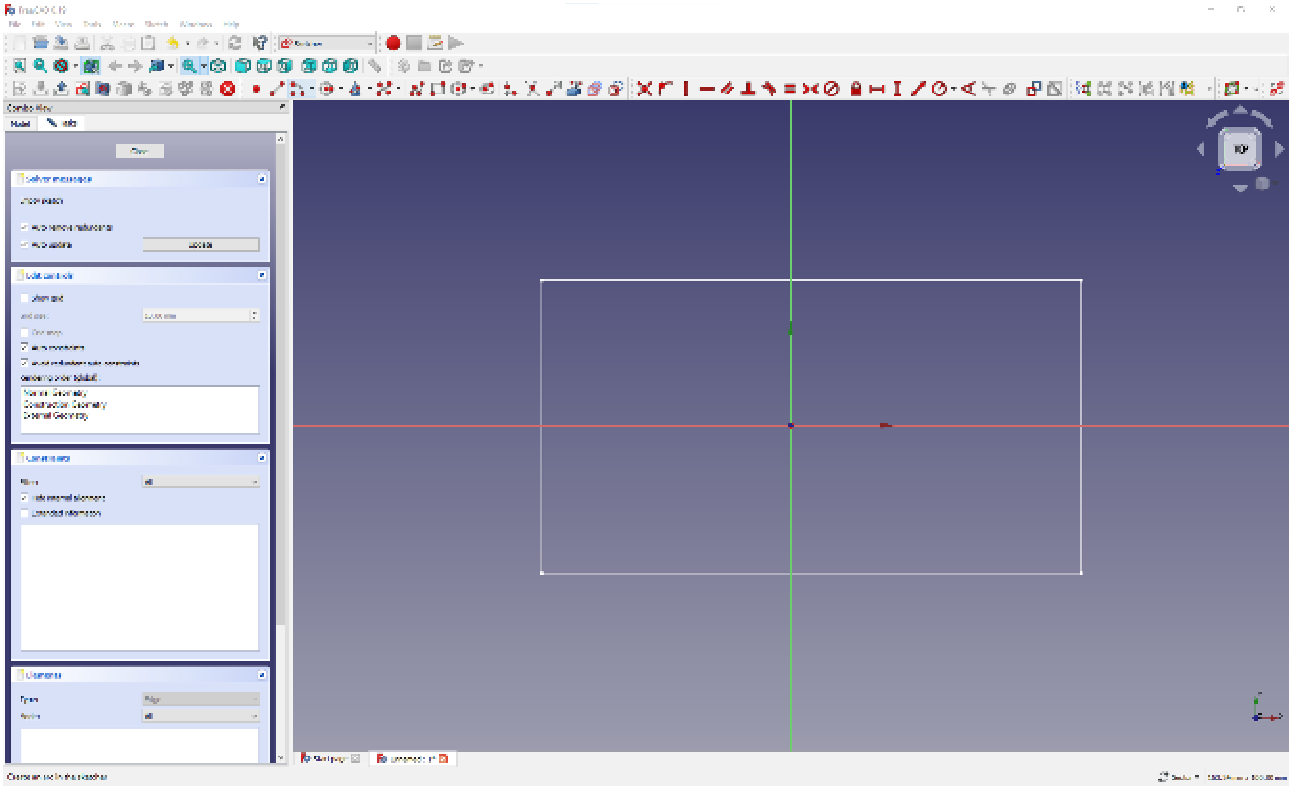
Lining up

Once you have your box generator set up, you can generate boxes of any dimensions easily. The layout on the XY plane means that the box and lid will be easy to export and 3D-print, as both parts are modelled in a way that emulates them sitting on the print bed. If you wanted to look at your box with the lid in position, again you could create a simple copy on the part workbench and move and rotate those parts into the correct position.

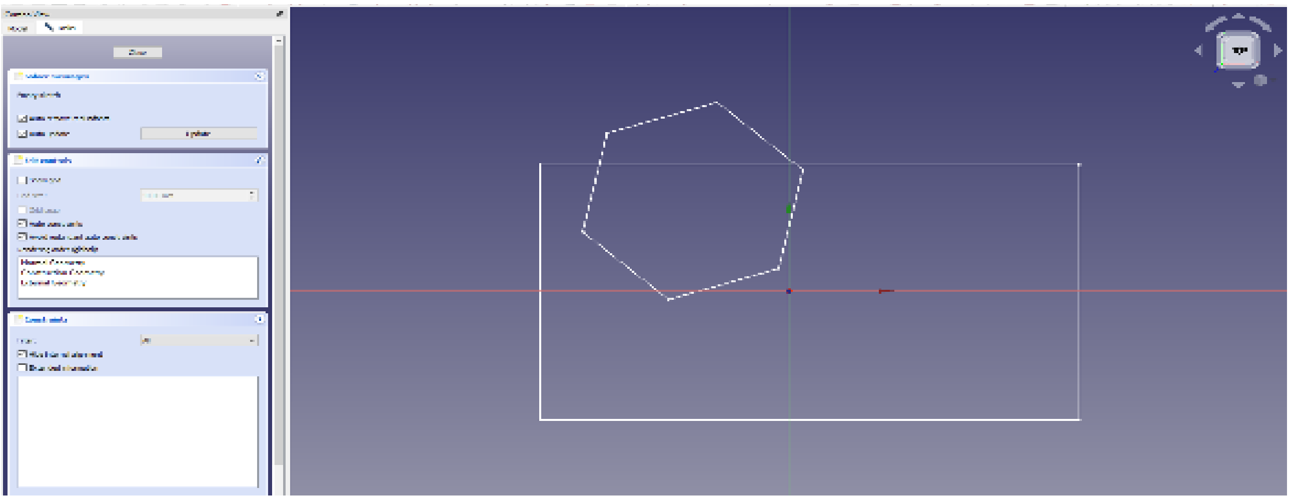
Curving all the way

First up is the Additive Loft function, which is a method that creates straight or curved transitions between sketches. It's a great way of creating complex flowing shapes. To start with, you will make a nonsense object just to see how it works, and then you will create a small ornamental bowl that can be 3D-printed.

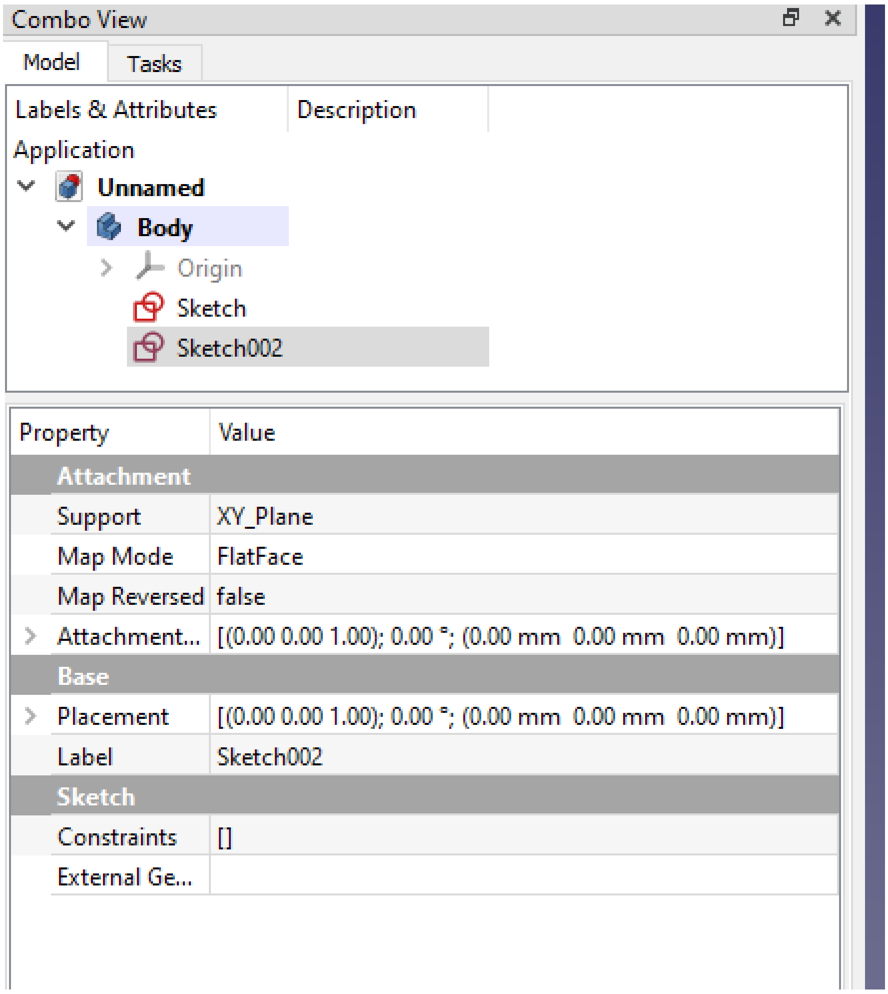
On the Part Design workbench, create a new body and then create a new sketch on the XY plane. This should move you onto the sketcher workbench. Select the rectangle tool and draw a rectangle in this sketch. As this is a test, you don't need to add constraints, so close this sketch now.



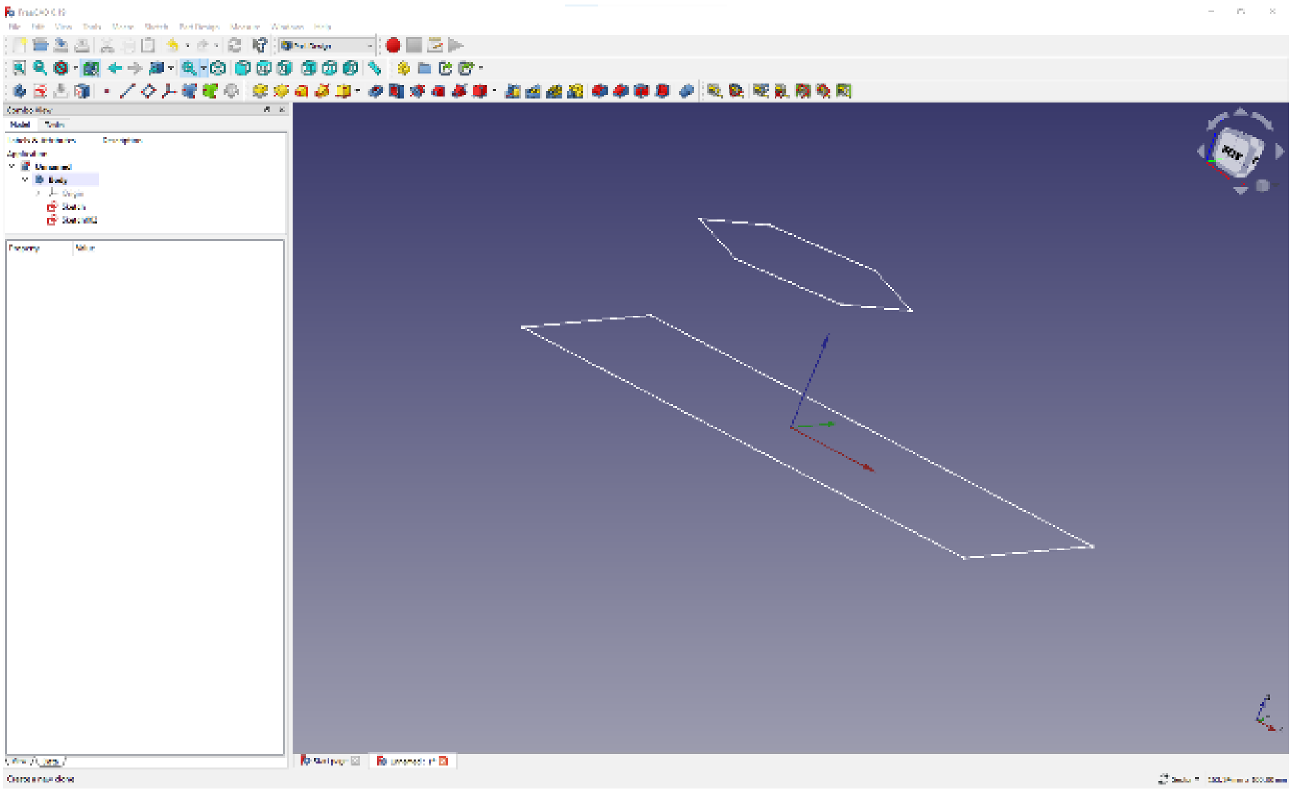
In the Part Design workbench again, within the same body, create a second sketch. Click the '`Create sketch`' button, and select the XY plane for your second sketch. In this second sketch, use the regular polygon tool to draw a hexagon that is smaller than the rectangle you drew – draw it inside the rectangle, and don't worry if it overlaps a little. Again, you don't need to constrain the hexagon, so close the sketch.



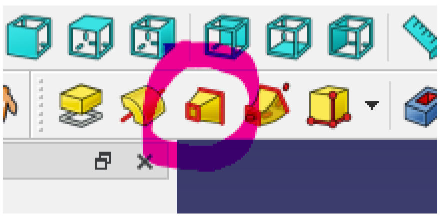
The next step is to move the hexagon sketch away from the rectangle sketch in the Z-axis. As both of the sketches are part of the same body, you can't move the basic position of the hexagon sketch, but you can move the relative attachment point of the sketch. With the sketch containing the hexagon selected, you should see the dialogue box for the sketch in the combo view panel. In this dialogue box, there are three sections of values separated under the headings '`Attachment`', '`Base`', and '`Sketch`'.



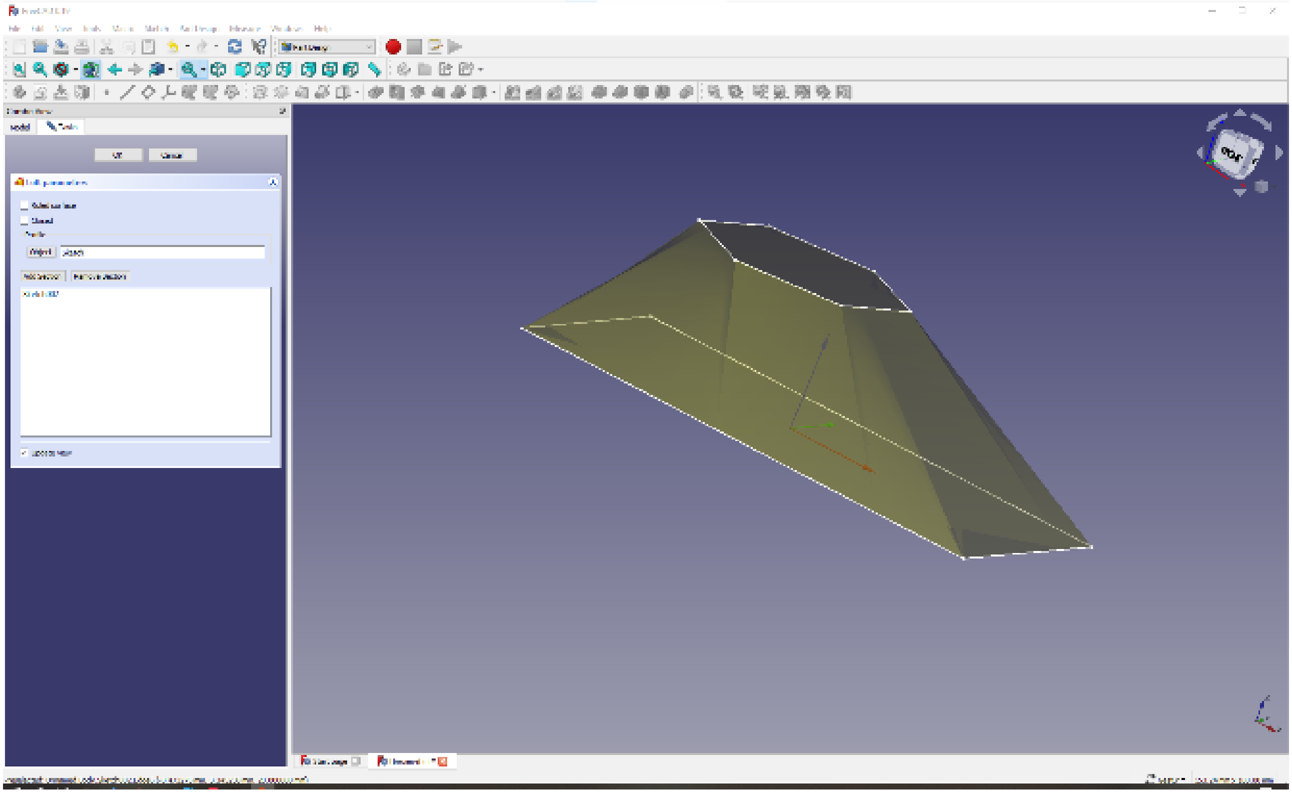
Making sure you are in the Attachment section, click the Attachment menu, and then click to expand the Position menu. In this drop-down menu, adjust the Z-axis to raise the hexagon sketch over the rectangle. This can be any amount, but I went for 20 mm. If you are in an isometric view in the Part Design workbench, you should see the hexagon sketch rise above the rectangle sketch.



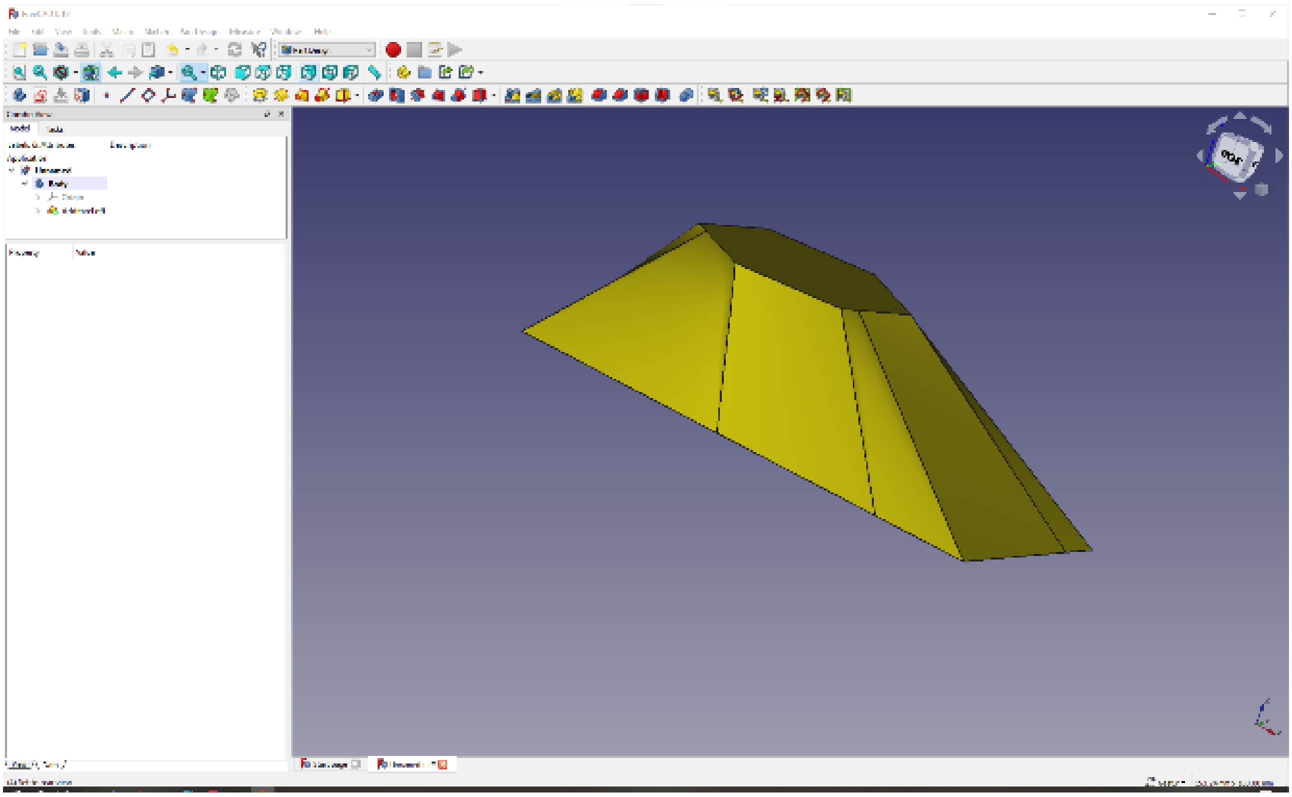
Next, select the original rectangle sketch in the file tree view, and then click the yellow and red '`loft a selected profile through other profile sections`' tool.



In the combo view panel, you should now see a window – click the 'Add section' button. Having clicked that, select the hexagon sketch in the preview window. As you do this, you should see a preview of the lofted part connected between the rectangle and the hexagon.



There are a couple of checkboxes you can click: one is called '`closed`' which will close the ends of the lofted object, and the other is 'ruled surface'. Clicking '`ruled surface`' toggles the lofted object between having either straight line/plane geometries or curved. It is not that obvious a difference with your simple example, but you will use this in your next lofting project.
To finish, click the '`closed`' checkbox and then OK to perform the loft. In the preview window, you will now see the resulting object.



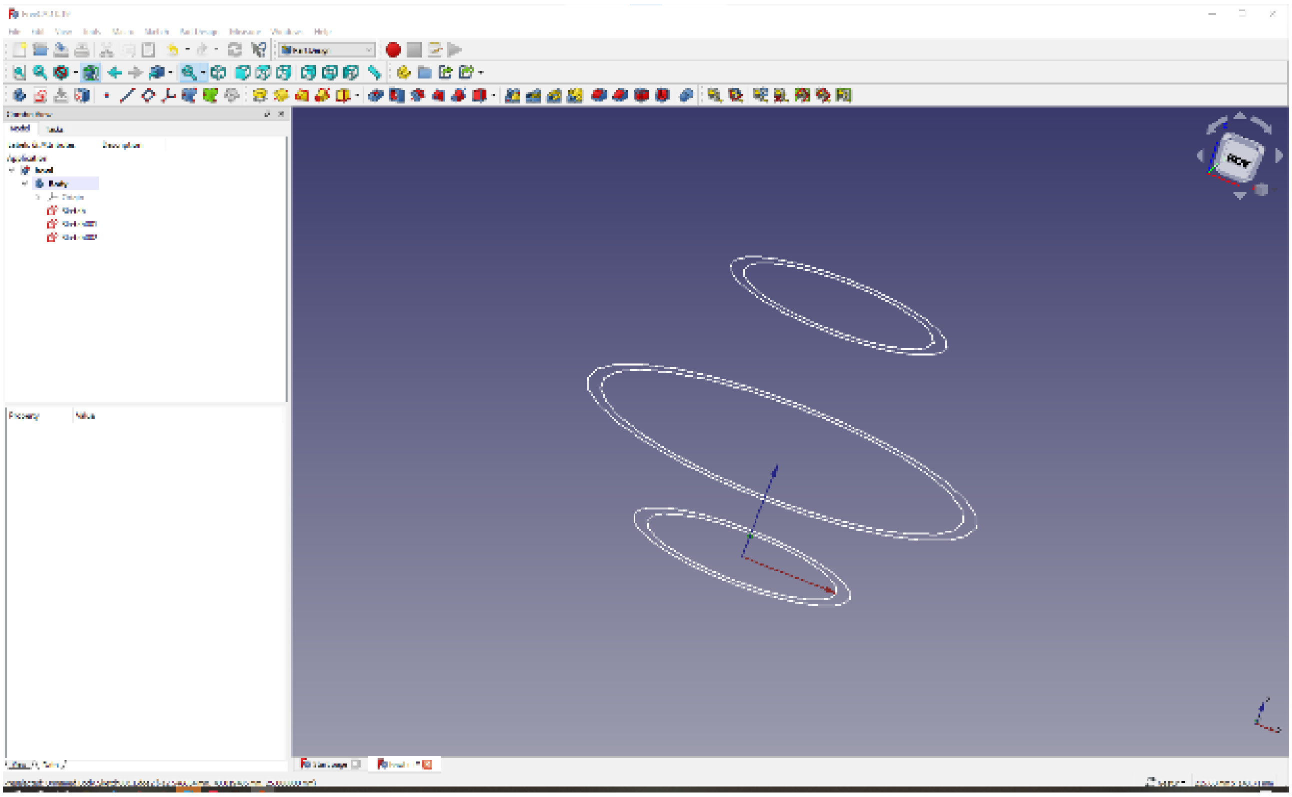
Lofted objects are like any other in that you can use other tools on the objects created with them, such as selecting edges for fillets. However, when the surfaces generated become complex and curved, you might find that some tools don't work and you need a different approach.

One thing people often try with the lofting tool is to apply a thickness to the resulting object to hollow it out. Sadly the thickness tools only work on simple geometries, so it’s not possible to work this way. However, let’s make a more interesting lofted object and show how you can create curved objects with defined thickness walls.

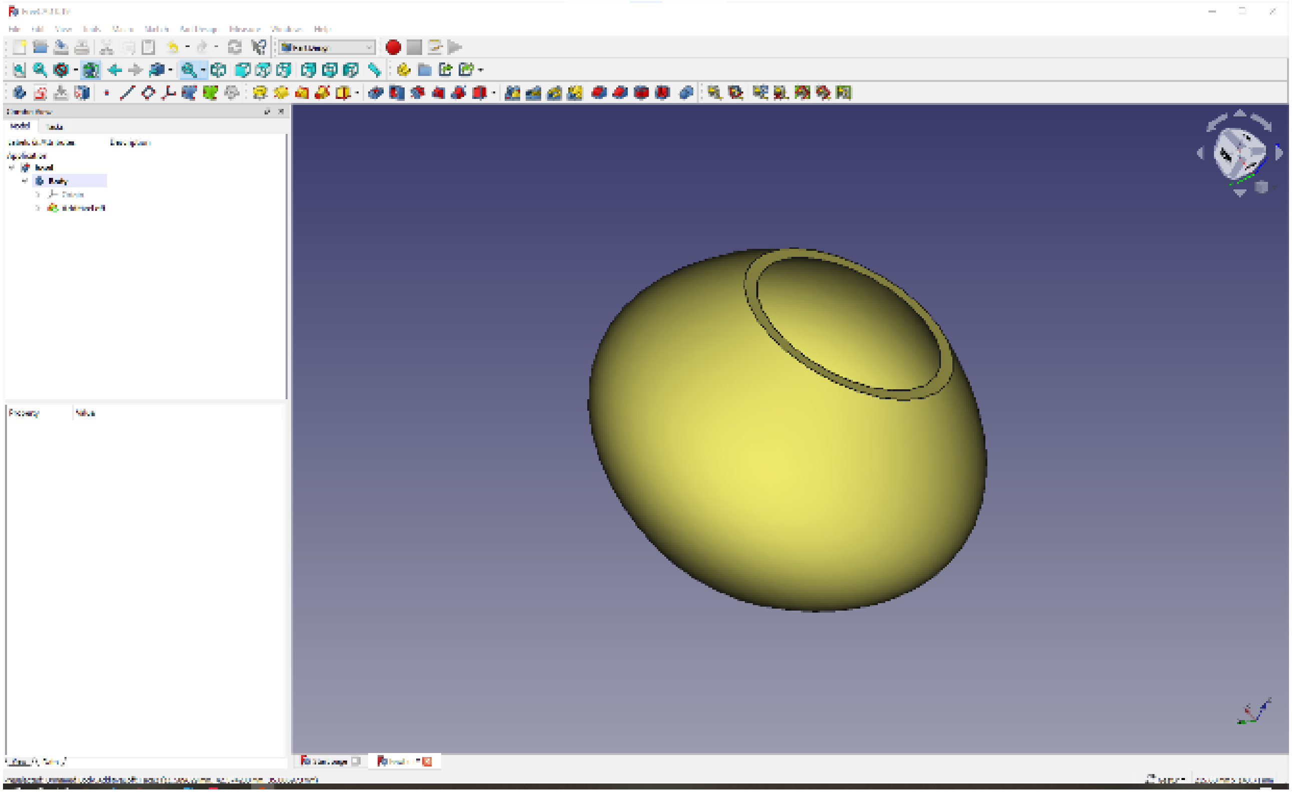
In a new project, create a body and create a sketch in the XY plane. Draw two circles constrained around the origin point and make one slightly smaller than the other; the distance between them will create the wall thickness of your object. You made your sketch with the outer circle having a diameter of 25 mm and the inner circle having a diameter of 22 mm, to give a wall thickness of 1.5 mm
Close the sketch and then create another sketch in the XY plane. Create a larger pair of circles, but with the same wall thickness – you went for 45 mm and 42 mm.
Close this sketch and then create a third sketch with a pair of circles constrained to match the first set you drew.

Quick tip : When you are drawing the third sketch, the circles will appear exactly on top of the firstsketch. To make it easier, make the original sketch invisible by highlighting it in the file tree, toggling visibility by pressing the space bar.

With the second sketch highlighted, move these circles up the Z-axis using the same method you used earlier with your hexagon. Move it up by 25 mm. Next, do the same with the third sketch, moving it up the Z-axis 60 mm.



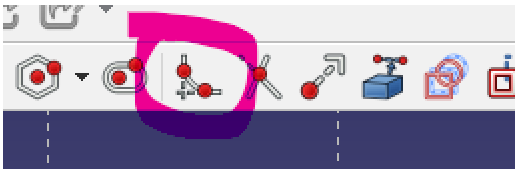
Create the loft in a similar way to the earlier example, starting by selecting the first sketch and then adding the second sketch and the third sketch in that order. If you uncheck the ruled option and the closed option, you should end up with a nice curvy shape.



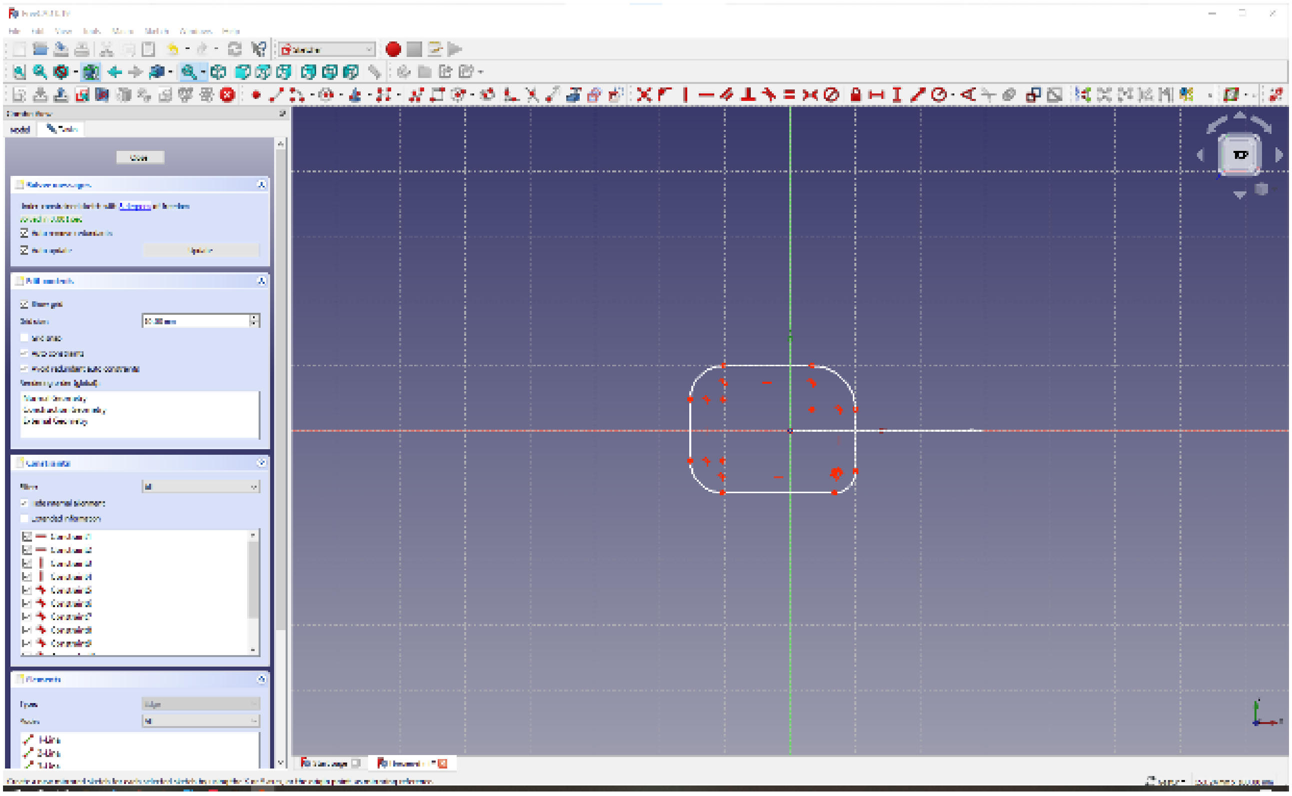
Sweeping along a path is a similar idea to revolving a sketch and also shares some attributes of lofting. However, sweeping uses a user-drawn path or geometry that the extrusion is swept along, rather than an automatically generated one. There are a couple of approaches to sweeping in FreeCAD: one using the part workbench, the other using the Part Design workbench. Let's look at both to have both methods in your skill set.

Curving in Part Design

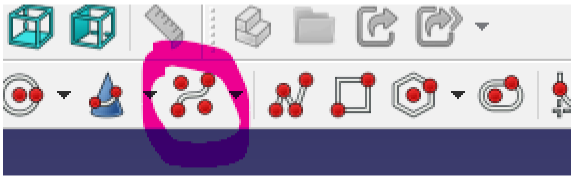
Starting on the Part Design workbench, create a new project, create a body, and create a sketch in the XY plane. In the blank sketch, make a small closed shape around the origin point. You drew a square and then added some fillets in the corners by clicking the '`create a fillet between two lines or at a coincident point`' tool.



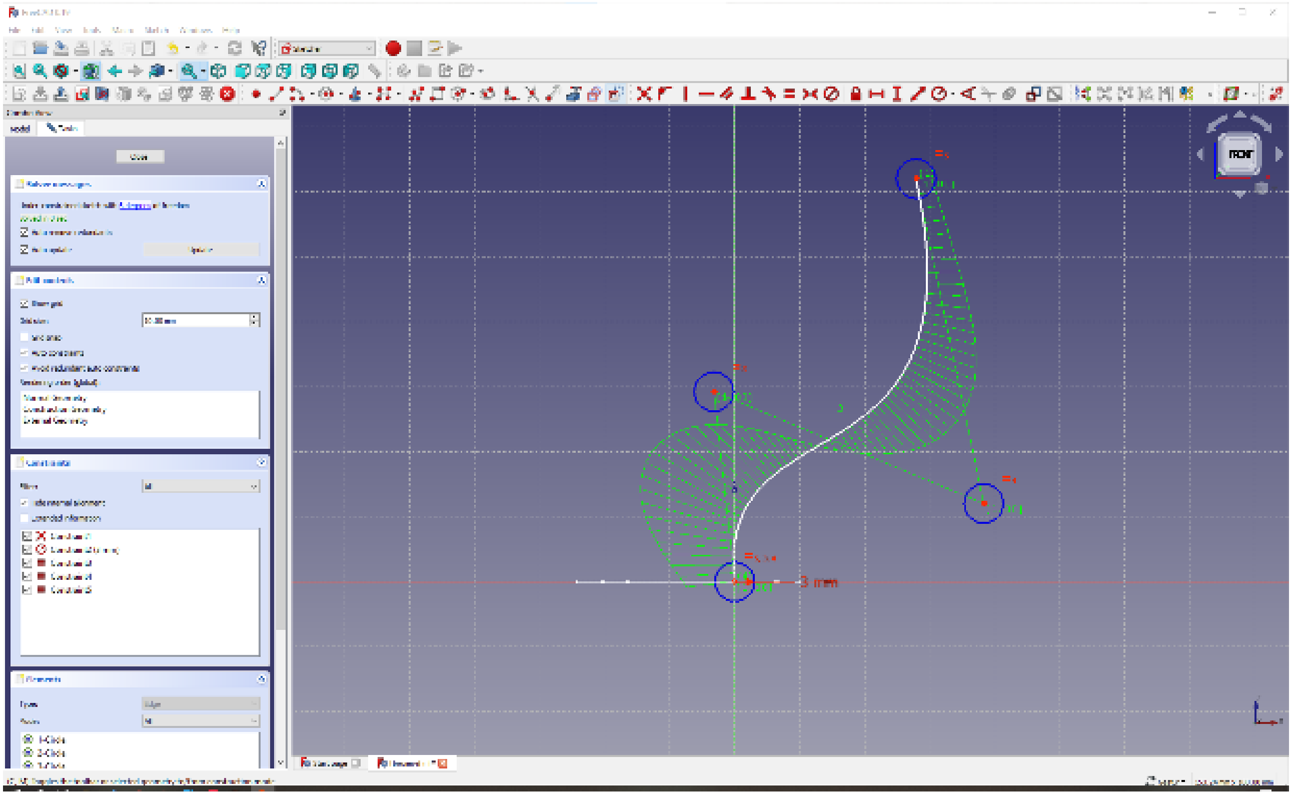
With the fillet tool selected, click on some corners of the square to add some fillets, and resize them to make an interesting shape. This shape will be the profile of your item you extrude by sweeping it along a path. Again, constraining the sketch is unimportant for a simple exploration of the tool.



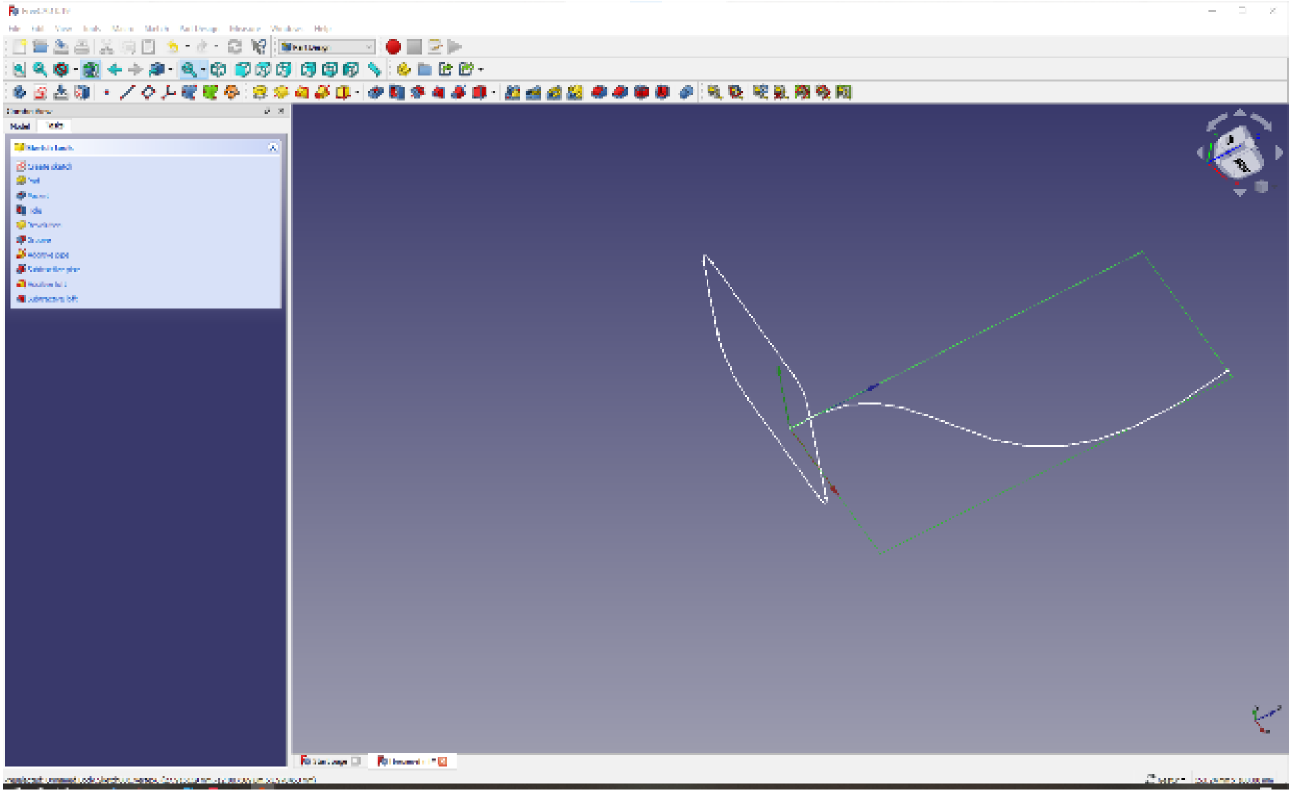
Close the sketch and then create a new empty sketch in the body. Create this new sketch in the XZ plane. In this sketch, you are going to create the path along which your first sketch profile will be swept. Let's use '`create a B-spline in the sketch`' tool.



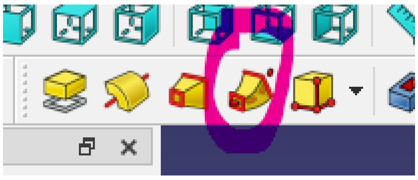
This tool allows us to create a combination of curves in a single line. With the B-spline tool selected, left-click over the origin point in the sketch and pull a line up and to the right or left of vertical, then left-click again and continue with the next line upwards and back towards the centre line. Left-click again and move back out to the right and upwards to create a skewed kind of Z-shape. Left-click to set the next point, and then right-click to finish the B-spline. The two lines you drew should now become curved, and you should see some blue circles. Simply, for now, moving the circles allows you to adjust the curve of the lines. Make a nice swooping curve, such as the one I created.



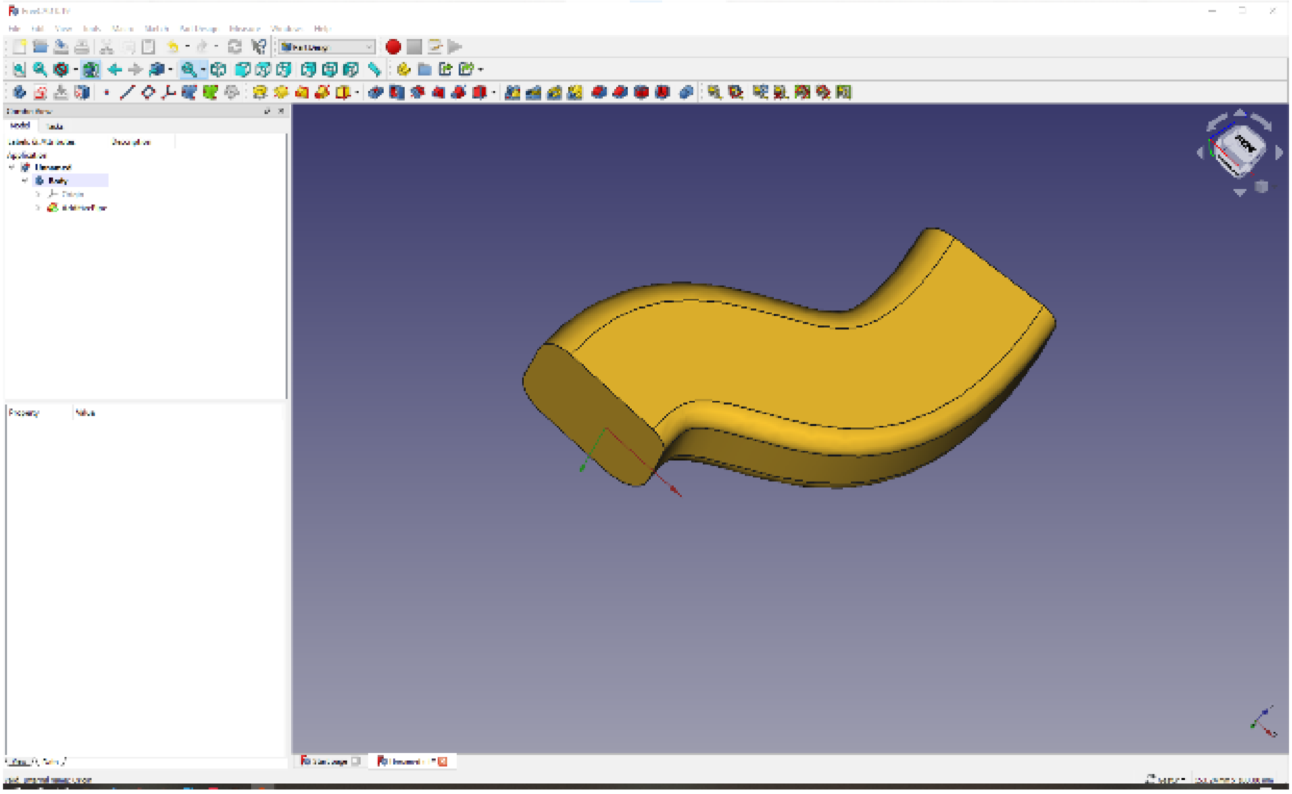
Close the sketch, and in the preview window, you can rotate your view a little and see that the curved edge you created starts at the origin point, which is in the middle of your profile sketch.



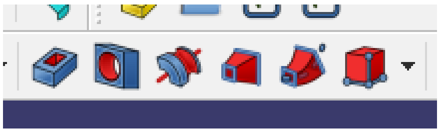
To sweep the profile along the curved path, highlight the first sketch you made and then click the yellow and red '`Sweep a selected sketch along a path`' tool.



You should see a dialogue box appear in the combo view panel. In the dialogue box, you need to select the path or object along which to sweep. You can do this in a couple of ways. You can either click the 'Object' button under the 'path to sweep along', or you can click the 'Add Edges' button. Having clicked either of these buttons, you can then select the curved path in the preview window. You should now see a brown preview of the profile swept along your path. Click 'OK' to finish the sweep. You should now have a nice swept object in the preview window.



There are many other options in the sweep utility in the Part Design workbench that can change the way that corners are handled in a swept object, and also how it handles transforming from one profile to another profile over the course of a sweep, which you will explore on the part workbench shortly. The other thing of note on the Part Design workbench is that there is a blue and red icon that is the same as the sweep tool you used, except that it's for subtractive sweeps.



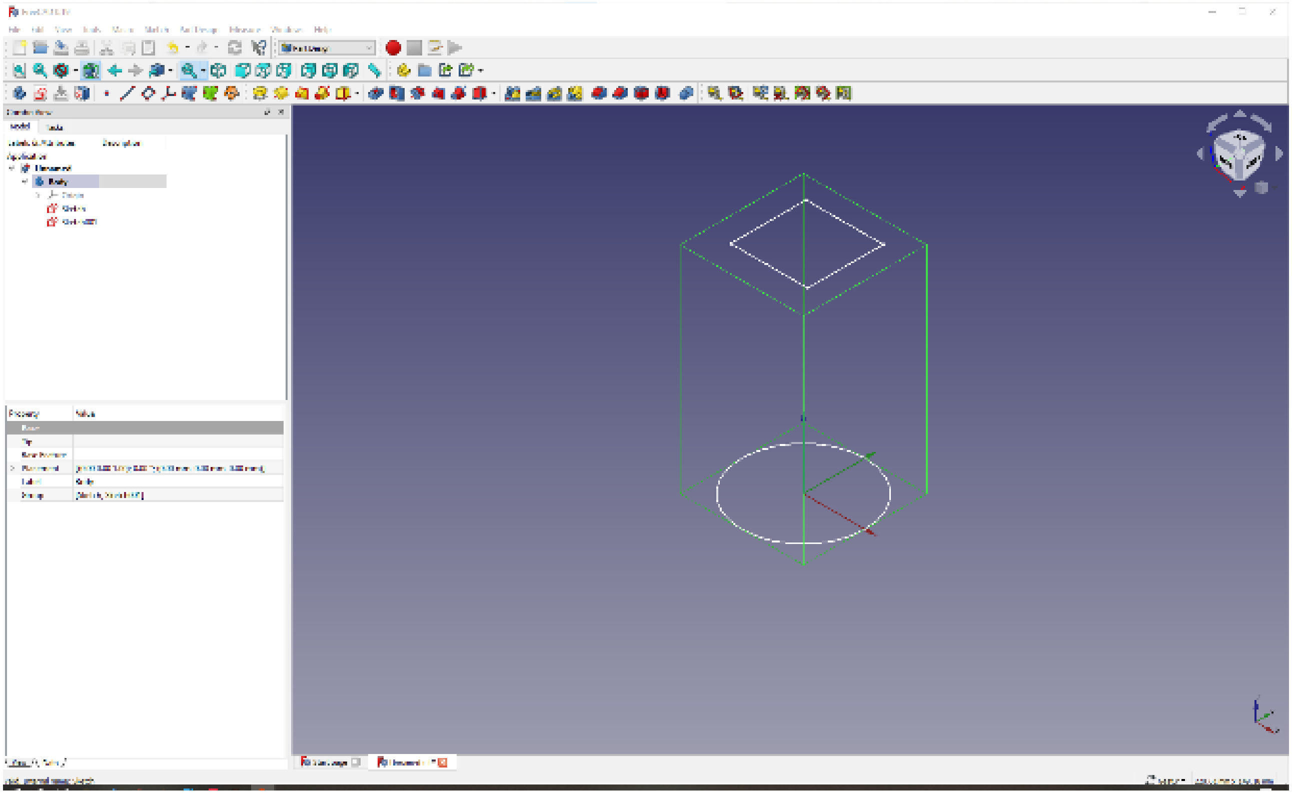
This means you could draw a solid object and then cut out and remove a swept path through it.

Quick Tip: The Part Design workbench sweep tool creates single solid body objects, but you can apply some techniques you learnt, where you used the '`Create a thickness tool`'. As a quick recap, select the face of one end of your swept solid object and click the thickness tool. In the dialogue box, you can set the thickness of the wall structure, and to make it a pipe object open at both ends, click '`Add face`' and then additionally select the opposite end of the swept object.

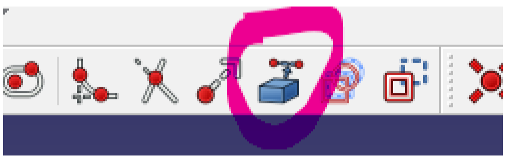
Curving in Part

Finally, let’s look at the sweep utilities on the Part workbench, which are laid out a little differently. You will also create two profiles and a path, and the profile will be transformed from one shape to the other along the path. Let's create a new project in Part Design workbench , create a new body. Create a new sketch in the XY plane and then draw a circle anchored around the origin point. It is up to you if you want to constrain it or not for this example. Close that sketch and then create a second sketch, again in the XY plane. In this sketch, let's draw a square around the origin point which should be a similar size to the circle you drew in the first sketch. Close this sketch, but highlight it in the combo view panel so that the dialogue box options for the sketch appear below it. You are going to move the position of the sketch so that it shifts vertically 70 mm up the Z-axis. In the sketch dialogue box under the '`Attachment`' heading, open the drop-down menu labelled '`Attachment`' then, in turn, open the drop-down menu labelled '`Position`'. In this menu, change the Z-axis value to 70 mm.

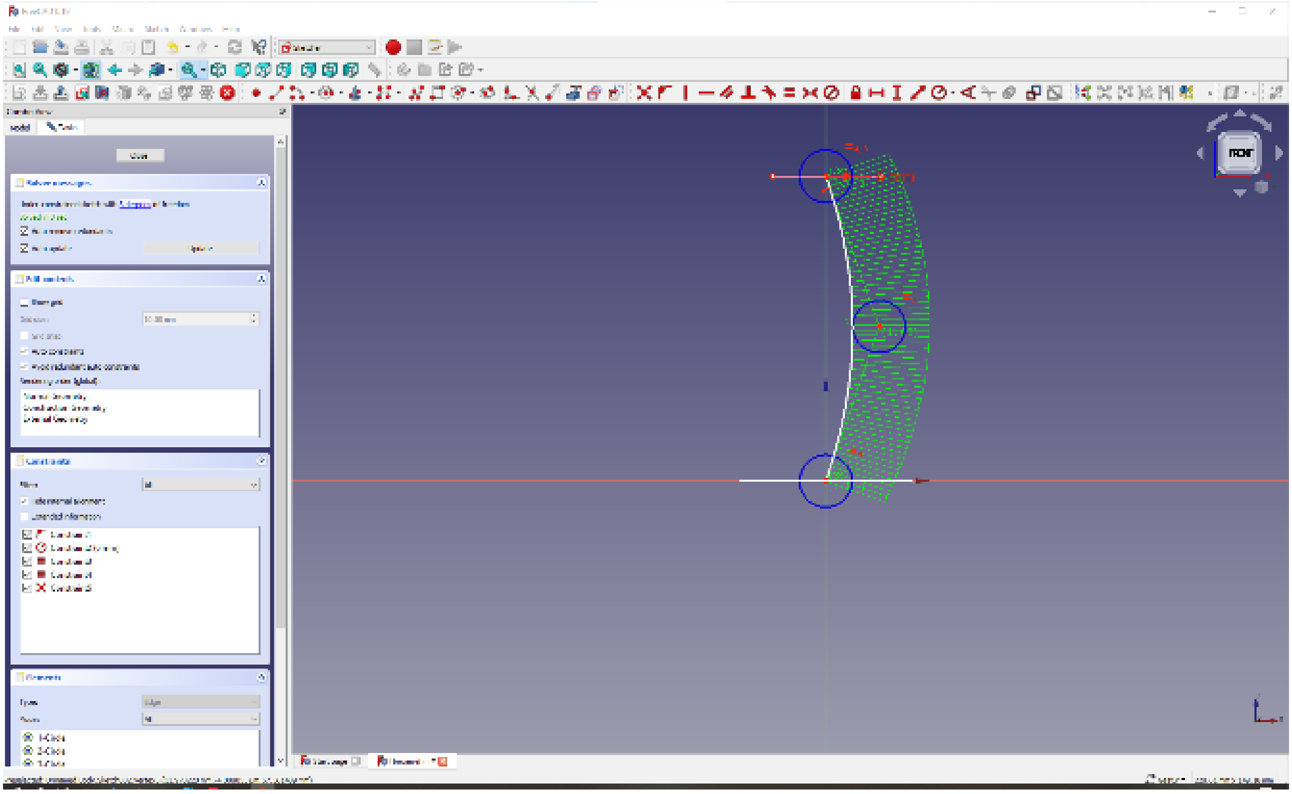
You should be able to see in the preview window that the square is now above the circle.



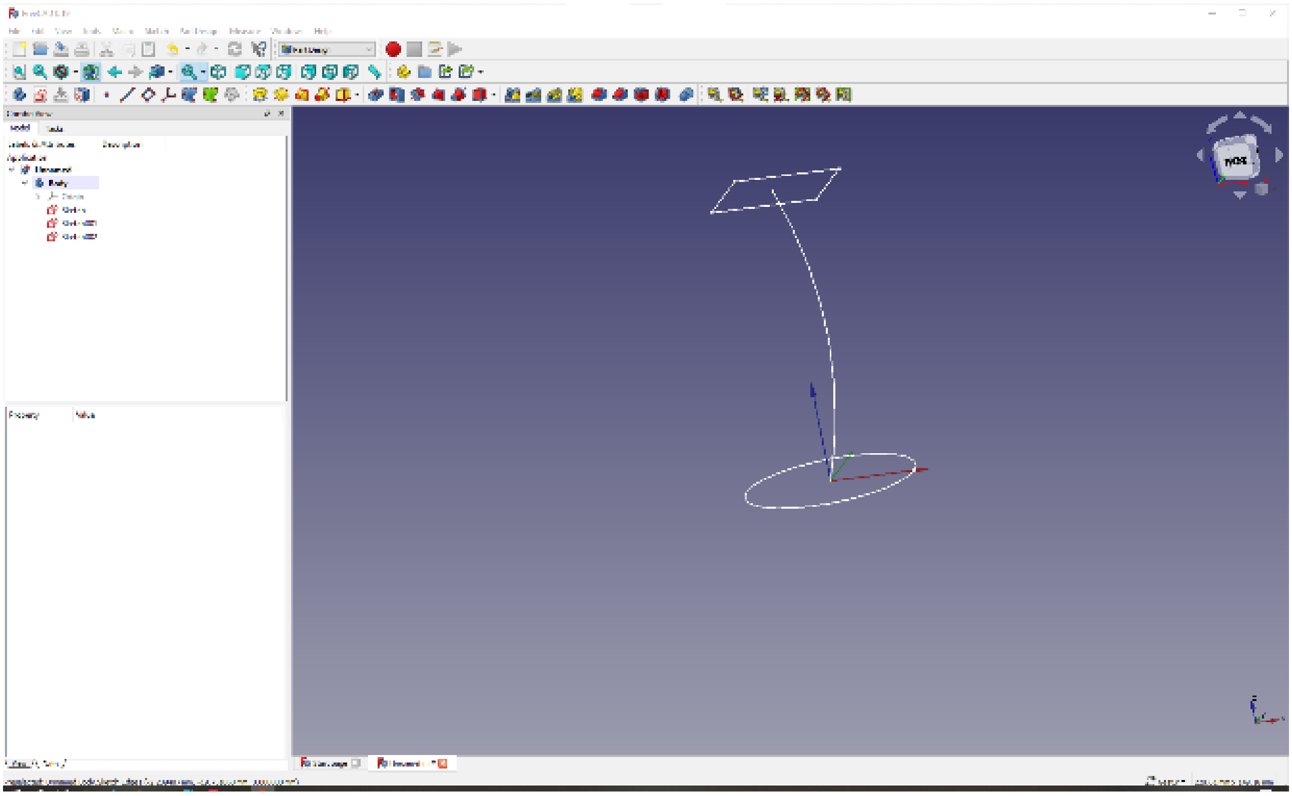
Next, making sure nothing is selected in the file tree, add a third sketch with this sketch drawn on the XZ plane. Using the '`create an edge linked to an external geometry`' tool,



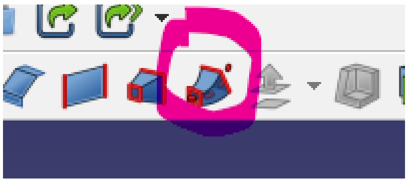
click the horizontal line that is on the upper square sketch. Next, use the B-spline tool to create a nice curved line which starts at the origin point and curves up to reach the centre of the upper square sketch.



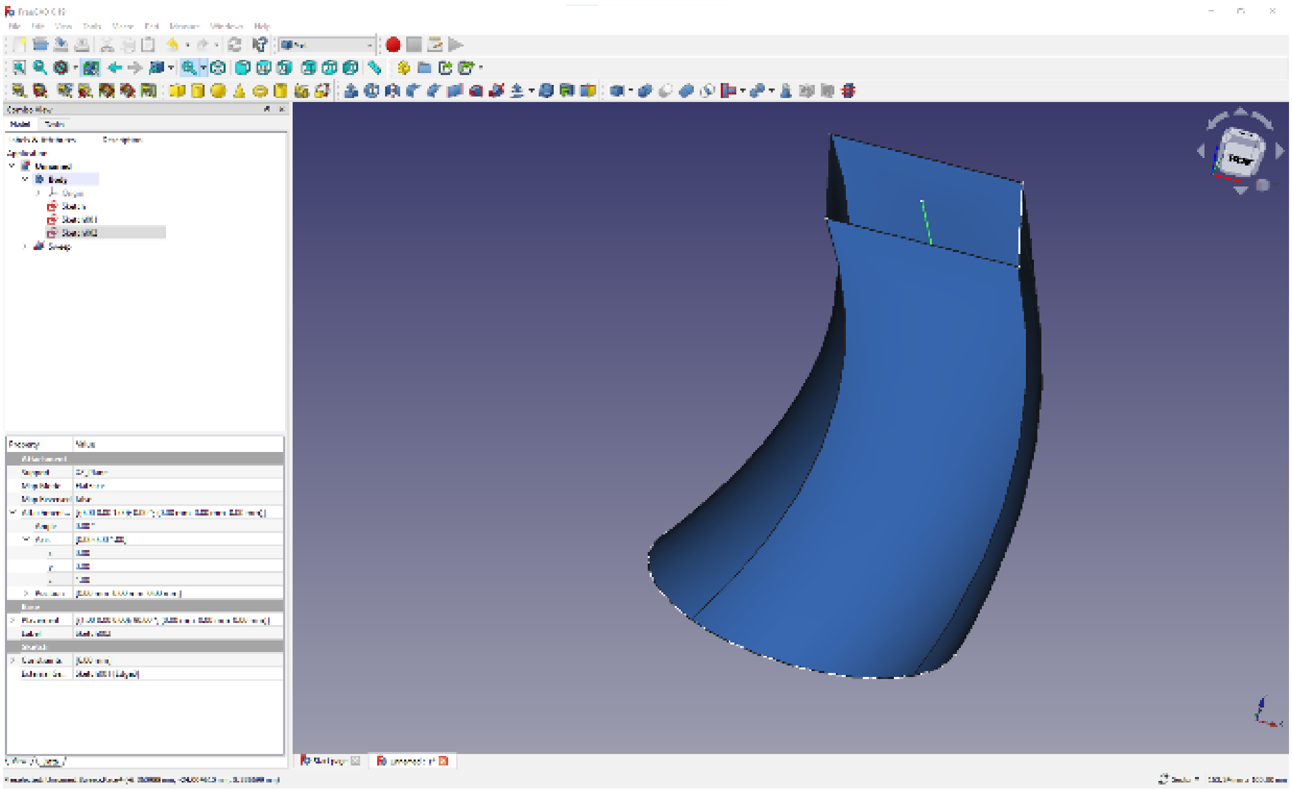
Then close the sketch, and you should have three sketches roughly similar to this.



Moving to the part workbench, click the '`utility to sweep`' tool.



In the dialogue box, you should see the list of sketches on the left-hand side. In turn, select the first sketch containing the circle, and click the right-facing arrow that becomes highlighted to move the sketch into the '`selected profiles`' column. Repeat this for the second sketch – the sketch that contains the square. Next, click the '`Sweep path`' button, and the dialogue box will become greyed out. Click the path you want to sweep along in the preview window to turn it green, and then click the curved path you created earlier. Next, click '`Done`' and then click the '`OK`' button. You should now see the circle profile has been swept towards the square on your curved path, and it has transformed throughout the curve to conform to the square profile.

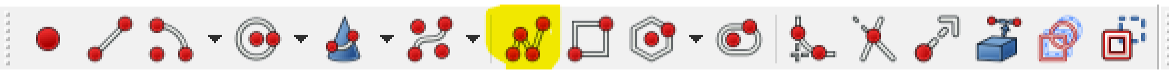


In the file tree, this is now a '`sweep`' object, and double-clicking the sweep object doesn't allow you to edit the sweep, but rather, opens the dialogue box to move and rotate the part. If you want to change any sweep parameters, you must select the sweep object and delete it, which will return you to just having the three separate sketches, and then repeat the sweep utility process. In the dialogue box for the sweep utility, you can select to make the object a solid. Then, you could create a simple copy and then use the thickness and other tools and utilities.

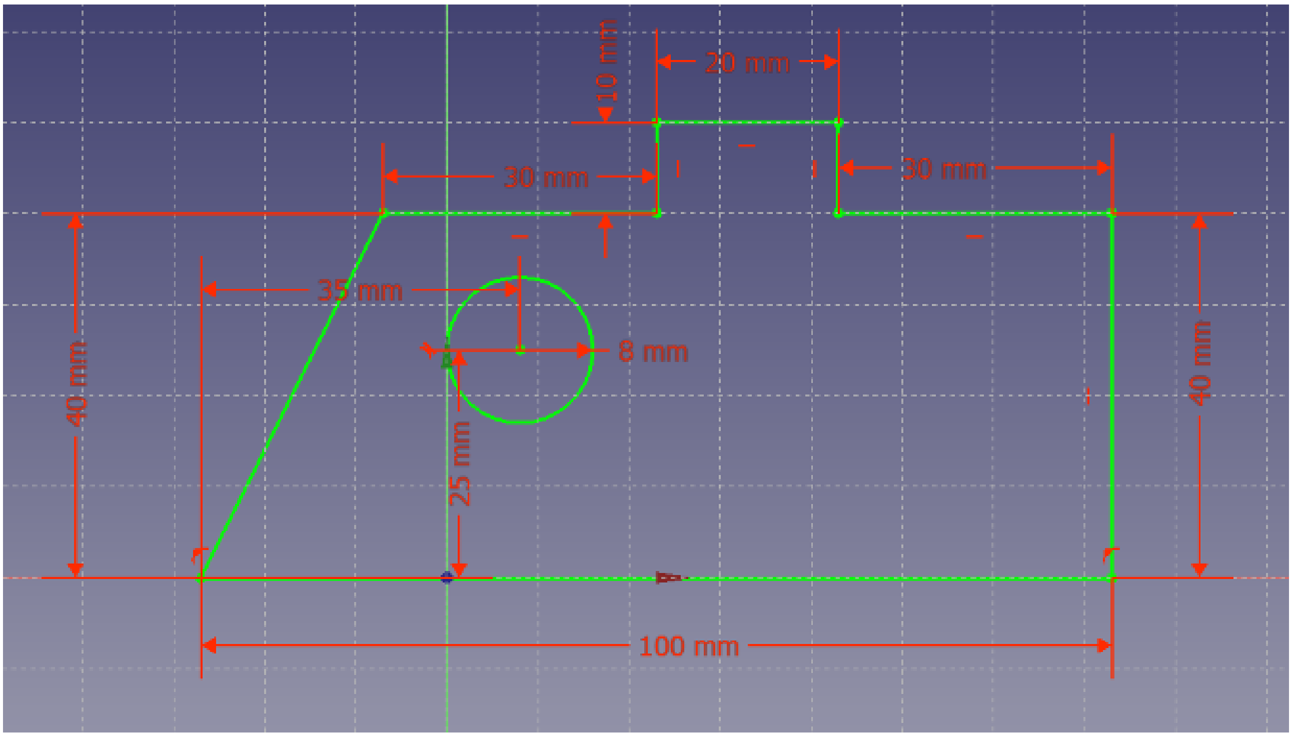
Technical drawings

Many users will be using FreeCAD to model in 3D to produce files that can be sliced and printed on a 3D printer. Other users, however, will be perhaps designing parts that they want to create using subtractive methods, like lathes or milling machines. Some will be using bench-fitting techniques to cut, drill, and file to hew parts from materials. To share the information needed to manufacture a part, the standard practice is to use technical drawings.
Drafting a technical drawing using pencil and paper and a drawing board is a fantastic skill to have, but it can be difficult to master. FreeCAD simplifies the creation of technical drawings from 3D models by having a dedicated technical drawing workbench called 'TechDraw'.

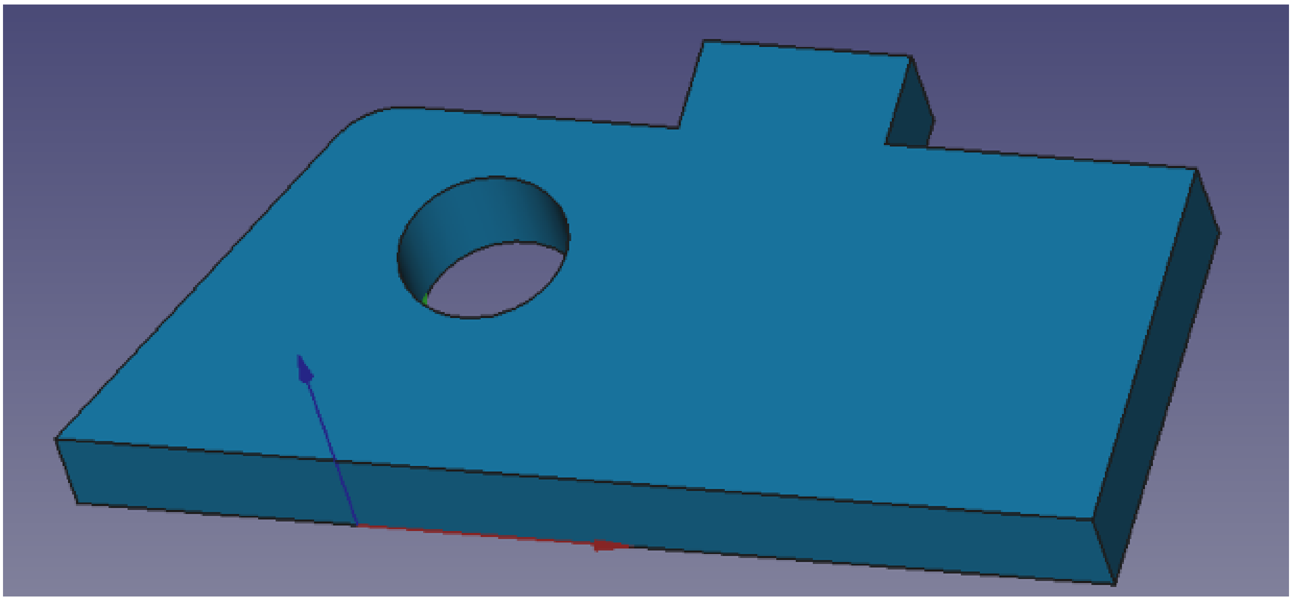
To work through an example of creating a technical drawing, you need a 3D model to use. Let's quickly draw an item using simple techniques you learnt using the Part Design workbench. Create a new project and, on the 'Part Design' workbench, create a body and then a sketch in the XY plane. Using the 'Create Polyline' tool, create a shape similar as below.



You don't necessarily need to constrain the entire model or sketch to use the TechDraw tools, but it's useful, in this case, to have all the dimensions constrained so you can compare the dimensions on the drawing (more on why that's important later).



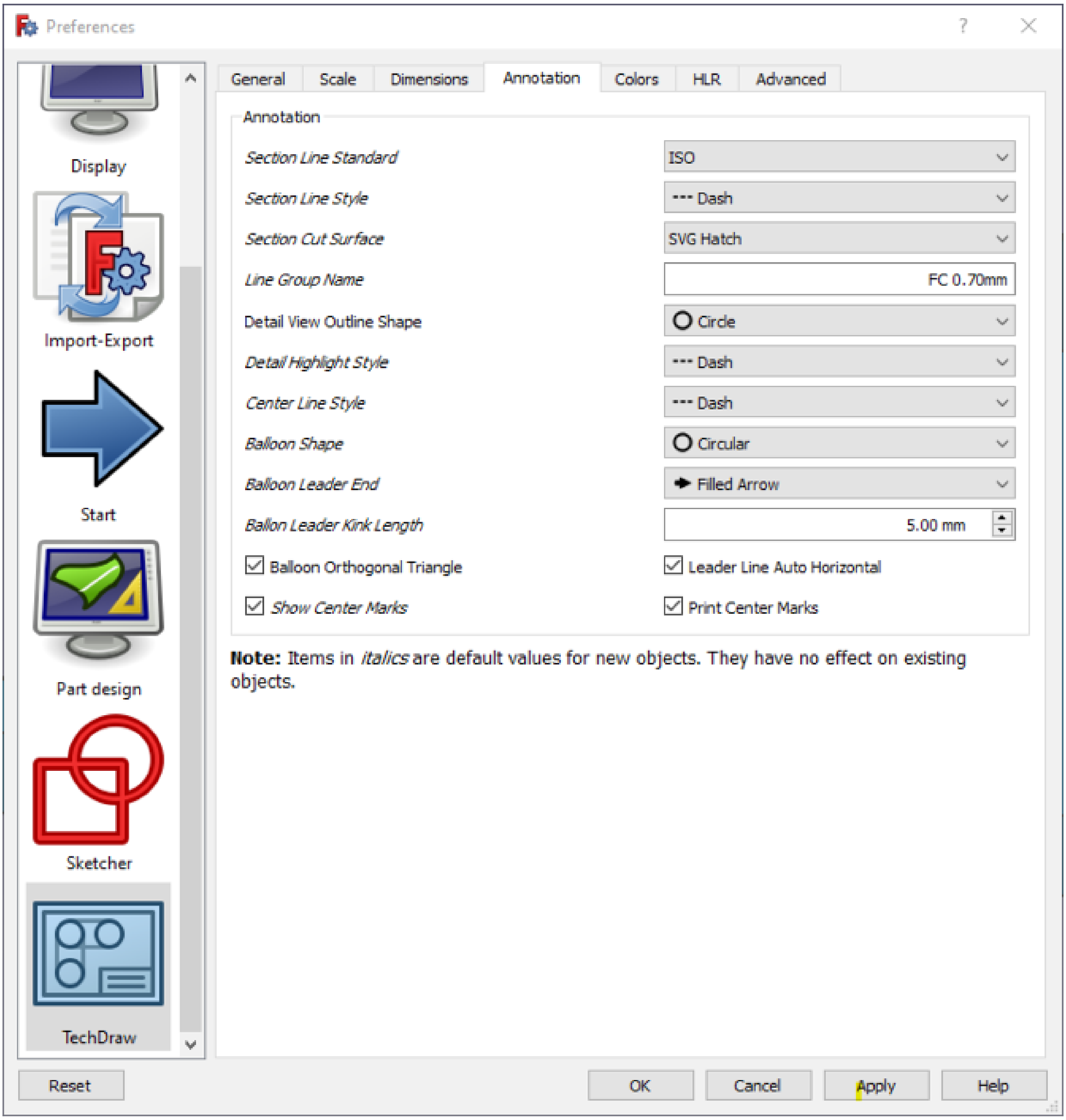
Having drawn the outline, add a circle in the object, and then close the sketch and perform a pad operation of 10 mm on the sketch to create your part. Finally, before you jump into the TechDraw workbench, add a fillet to an external edge with a 7 mm radius.



Having an internal hole and an external radius and an angled edge gives us enough details to do a good demonstration of the TechDraw workbench tools.

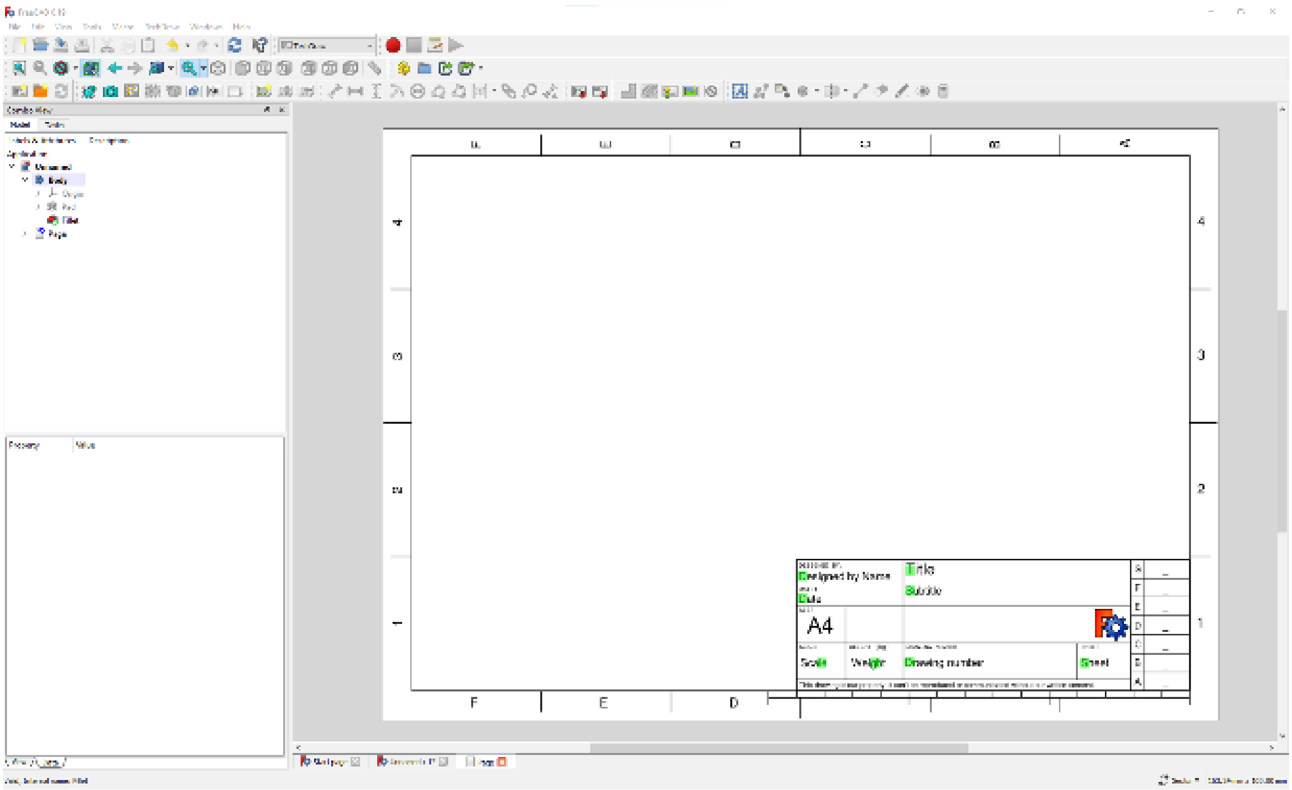
Getting Set Up

Before you add any views to your new page, you need to make an important change in some preferences so that the centres of circles and arcs are marked when they are drawn in TechDraw views. Making sure you are on the TechDraw workbench, go to Edit > Preferences and then scroll down the left-hand side to the TechDraw icon. On the TechDraw preferences window, click the 'Annotation' tab and make sure the 'Show centre marks' and 'Print centre marks' boxes are checked. Click Apply, and then OK to close Preferences.

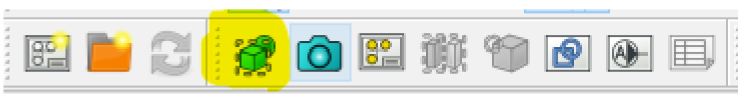


Making a techinal drawing
To begin playing with TechDraw, select the TechDraw workbench from the drop-down menu. Click the 'Insert Default Page' tool icon and you should have a new tab in the preview window with an empty technical drawing page in it.

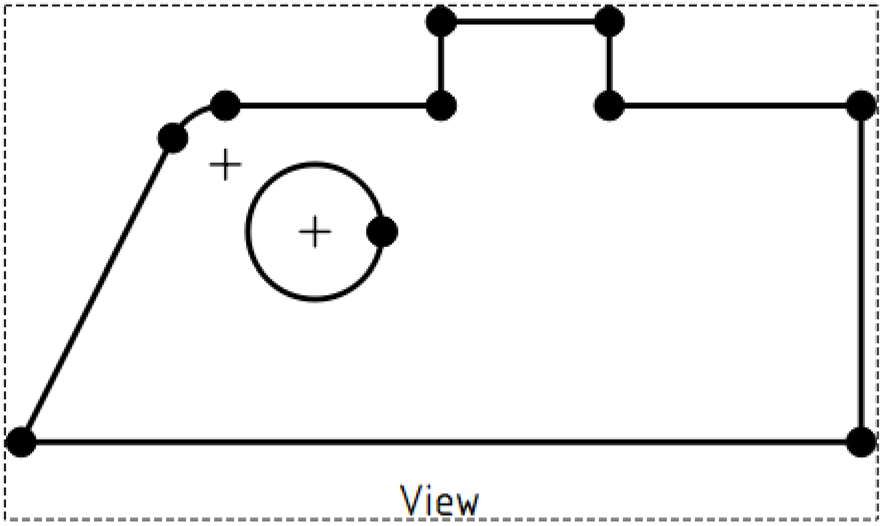
You can see that the new 'page' is listed as an object in the file tree.



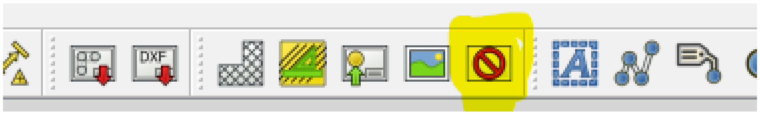
Switch back to the tab that contains the 3D model and you'll notice that, despite switching tabs, you are still in the TechDraw workbench. As an experiment, move the model so that it is at an angle rather than in one of the 'front, top, back' type views. Next, ensuring the body is highlighted in the file tree, click the 'Insert a View' tool icon.



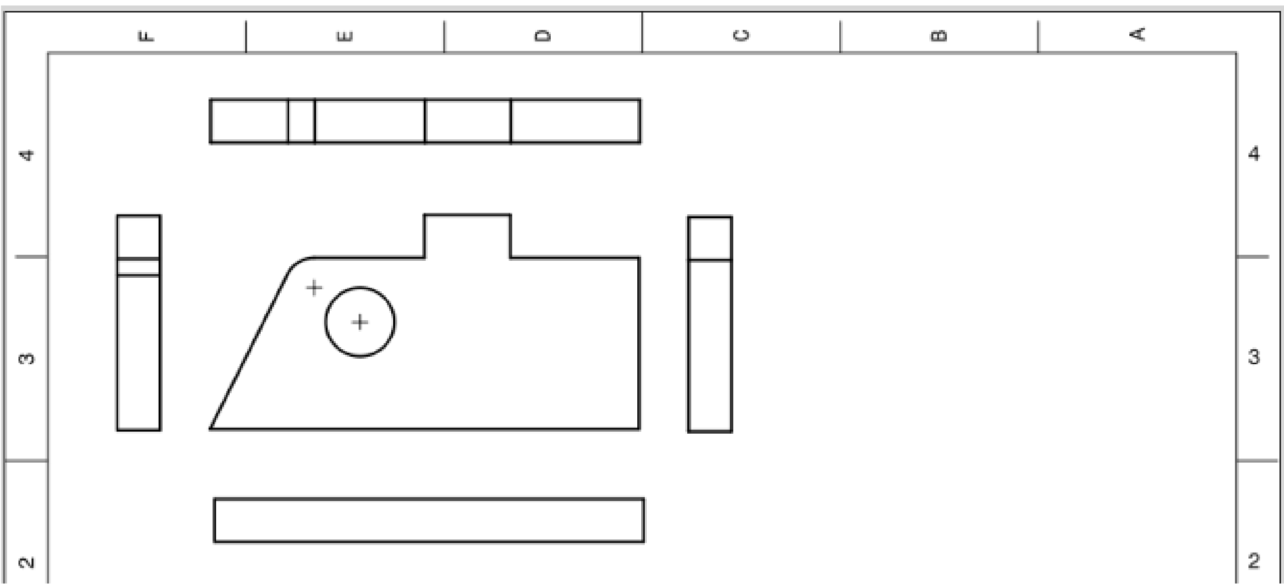
If you switch to the 'page' tab, you should see the 3D object has been added to the drawing in its exact view, at the angle it appears on the live preview.
It's important you know that it does this, because if your object is slightly at an angle, you might not notice, but it can affect dimensions in technical drawings added later. On the 'page' tab, you should have a dashed line around the 'view' – these are referred to as the 'View Frames'. Clicking the frame and dragging allows you to move the view around the technical drawing page. Highlighting the frame and clicking delete allows you to remove it. Click and delete this angled view and let's have another go. Return to the preview window and, using either the view icons, or the rotation block, select the 'top' view. Repeat the process of clicking the 'Insert a View' tool to add the view to the page.



You'll notice that all the lines in the drawing have the vertices marked as dots (similar to the Sketcher layout). The vertices help us to add dimensions and other tasks, but they are part of the view frame. You can toggle the view frames on and off by clicking the 'Turn View Frames On/Off' icon, or by right-clicking and selecting 'Toggle Frames'.

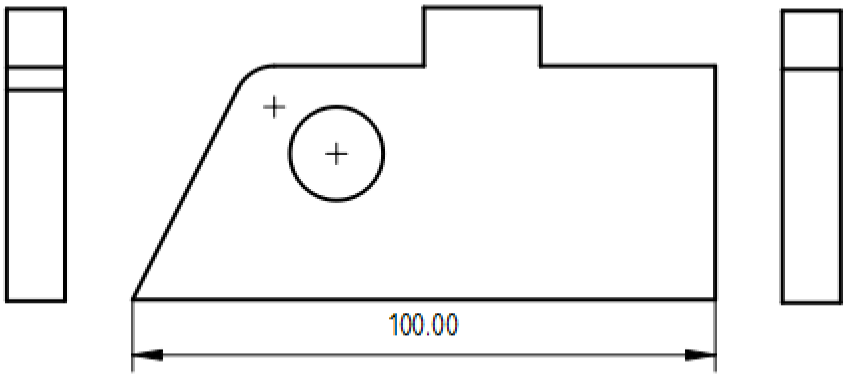


Your simple test piece design doesn't need masses of views to be able to show all of its attributes and dimensions, but you would definitely need another view adding to show the thickness of the part. Later, you will look at automatically generating multiple views of a part which adhere to technical drawing. Do this manually as you did with your top view. Return to the 3D model preview, and move the model so you are in a left- or right-hand view, then click the 'Insert View' button.

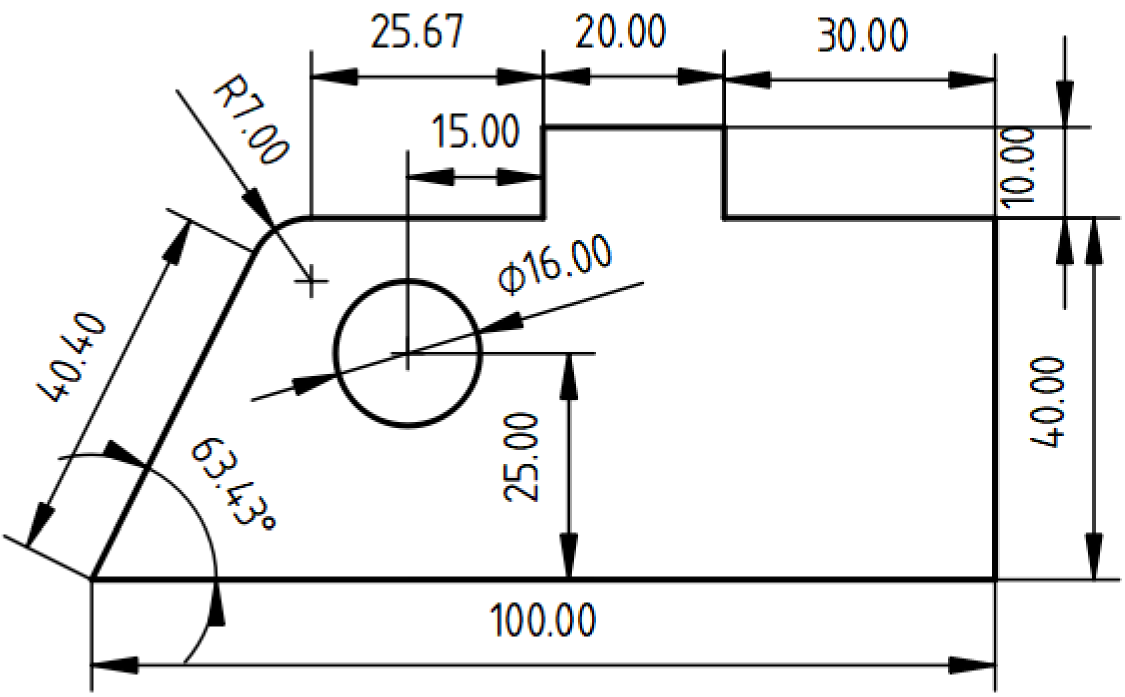


Next, let's add some dimensions to parts of the views so that someone could make this part accurately. It's similar in feel to adding constraints in the Sketcher workbench. First, let's select the long line/edge at the base of your object in the front view on the technical drawing page. Then click the yellow 'Insert Horizontal Dimension' tool icon.

You should now see a dimension appear in the drawing. Similar to the Sketcher workbench, you can click and drag this dimension to different areas to organise where the labels are.



Adding the height of the 3D object is a similar task, except you don't have a single complete line from the bottom to the top of the object, so you will use vertices instead. Similar to the vertical and horizontal constraints in the Sketcher workbench, TechDraw can create a vertical dimension between two points, even if they don't sit on the same edge.



Slightly differently than the Sketcher workbench, when you want to select multiple vertices, you need to hold the CTRL key down. Select the right-hand vertex on the bottom edge of your object and the highest vertex on the right of the small bit that sticks up at the top of the design. Click the 'Insert Vertical Dimension' tool to add the dimension and note that it correctly adds the vertical distance between the points – not the distance along a straight line between the points.

Holes in designs in technical drawings are commonly annotated with the diameter rather than the radius, often as the maker will be wanting to know what size drill or reamer to use. Adding a dimension for the diameter of your hole is similar to adding a radius constraint when you drew the circle in Sketcher. You simply select the circle and then click the yellow 'Insert Diameter Dimension' and the diameter of your hole is displayed.

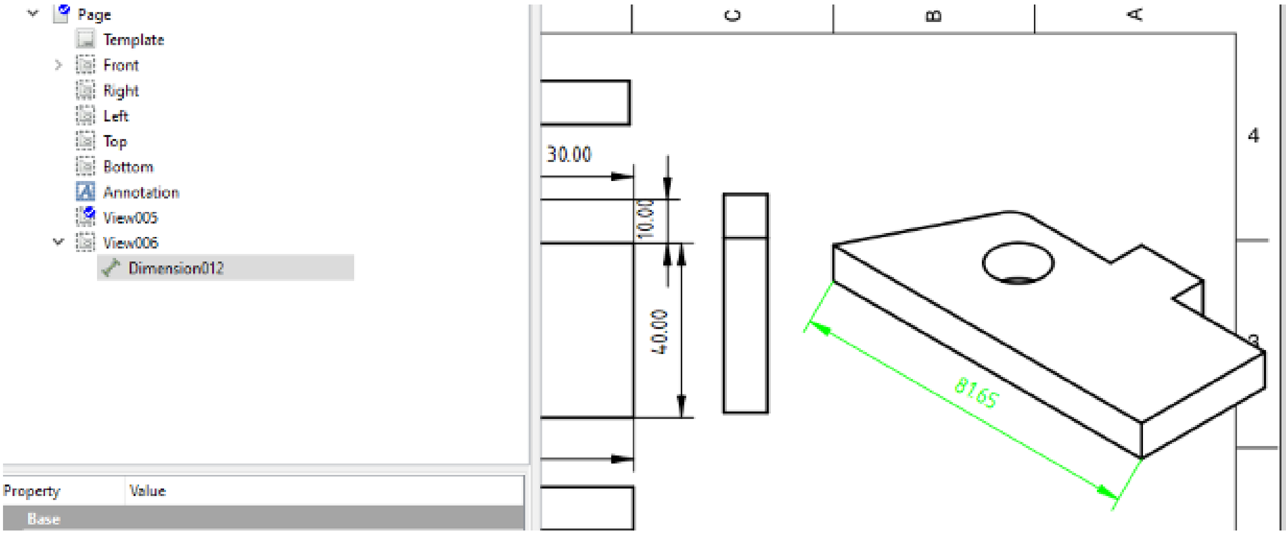
To indicate the co-ordinate position of the hole centre, you can select the centre mark and apply horizontal and vertical dimensions relative to another vertex. You have selected the lower left-hand corner of the object to act as this datum point and, of course, you could repeat this for the centre marks of the arc of the fillet. Most technical drawings would have a radius of the arc marked, rather than a diameter, and this would apply to your fillet arc. To add this, simply click the arc line and then click the 'Insert Radius Dimension' tool

To add a dimension showing the angle of the left-hand side line, relative to the base of the object, select both these lines and click the yellow 'Insert Angle Dimension' tool icon. Whilst it's not often necessary, sometimes you might want to add the actual length of a line rather than a vertical or horizontal and, to do this, let's select the angled line and click the yellow 'Insert Length Dimension' tool.

This is a very useful function at times, but also it hints at the way TechDraw handles line/edge length and a possible problem. If you return to your 3D object in the live preview, click the 'Set to Isometric View' icon from the collection of blue view icon tools.

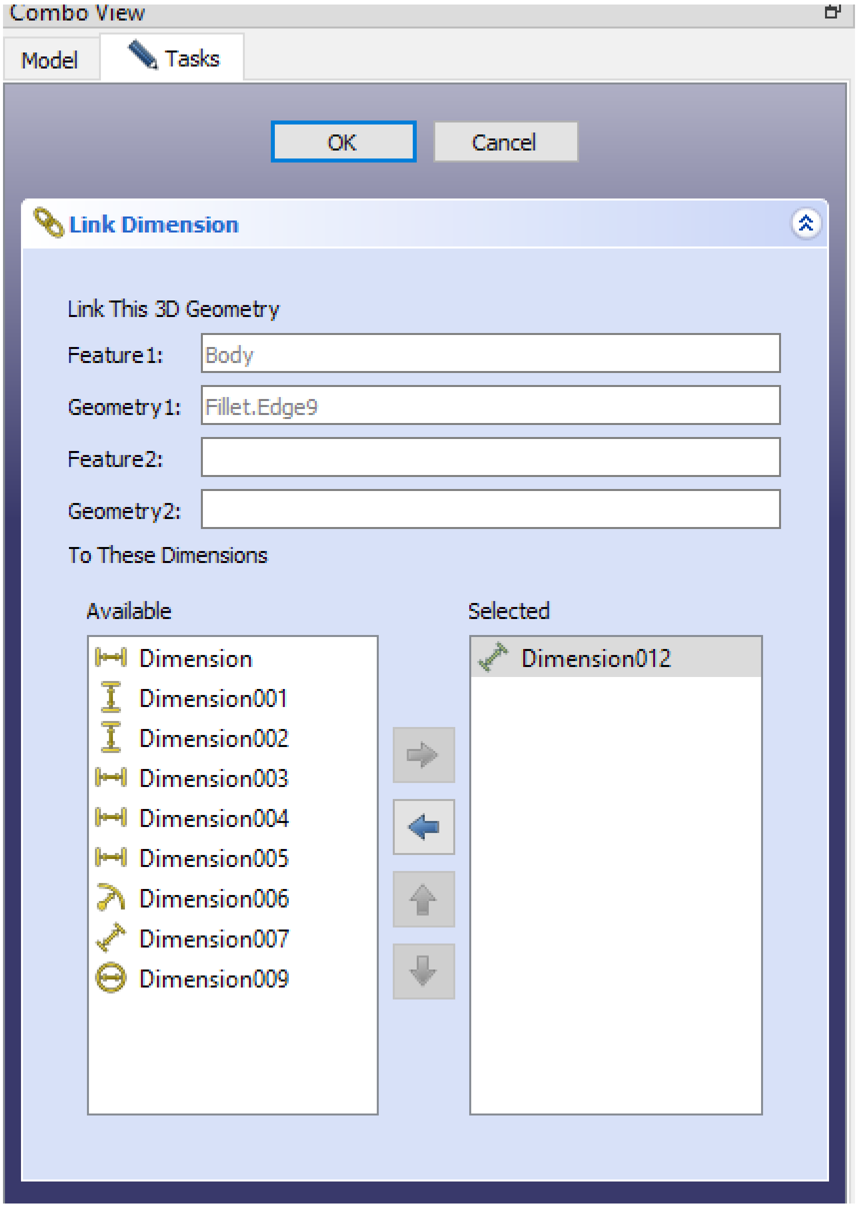
With the object now at an angled isometric view, let's add a new view to your TechDraw page. Move to the technical drawing page and move the new view so that it's clear of other views. Select one of the long lines that is the base of your object and click the 'Insert Length Dimension' tool. If you compare the dimension you have just added to the very first horizontal dimension you added on your first view of the object, you'll notice that they are different. This is

because the TechDraw dimensions default to being based off the view drawing rather than the actual 3D object, so the actual length of the angled line in the isometric view drawing is shorter.
If you wanted to add an isometric view with some dimensions added, you can overcome this problem by linking a dimension to the 3D object. First, highlight the incorrect dimension you just added to the isometric view, and look in the file tree view to identify the name of this dimension. In your example, this was ‘Dimension012’.



Having noted that, click over to the 3D object tab and highlight the same edge on the 3D object. Next, click the yellow ‘Link Dimension to 3D Geometry’ tool.

In the combo view, you will see a window with a list of dimensions in a left-hand column. Select the dimension name that you noted earlier and then click the right-facing arrow to bring that dimension name over to the right-hand column.



Click OK at the top of the menu and return to the technical drawing page tab.
You should now see that the dimension has updated to the correct length of the 3D object.

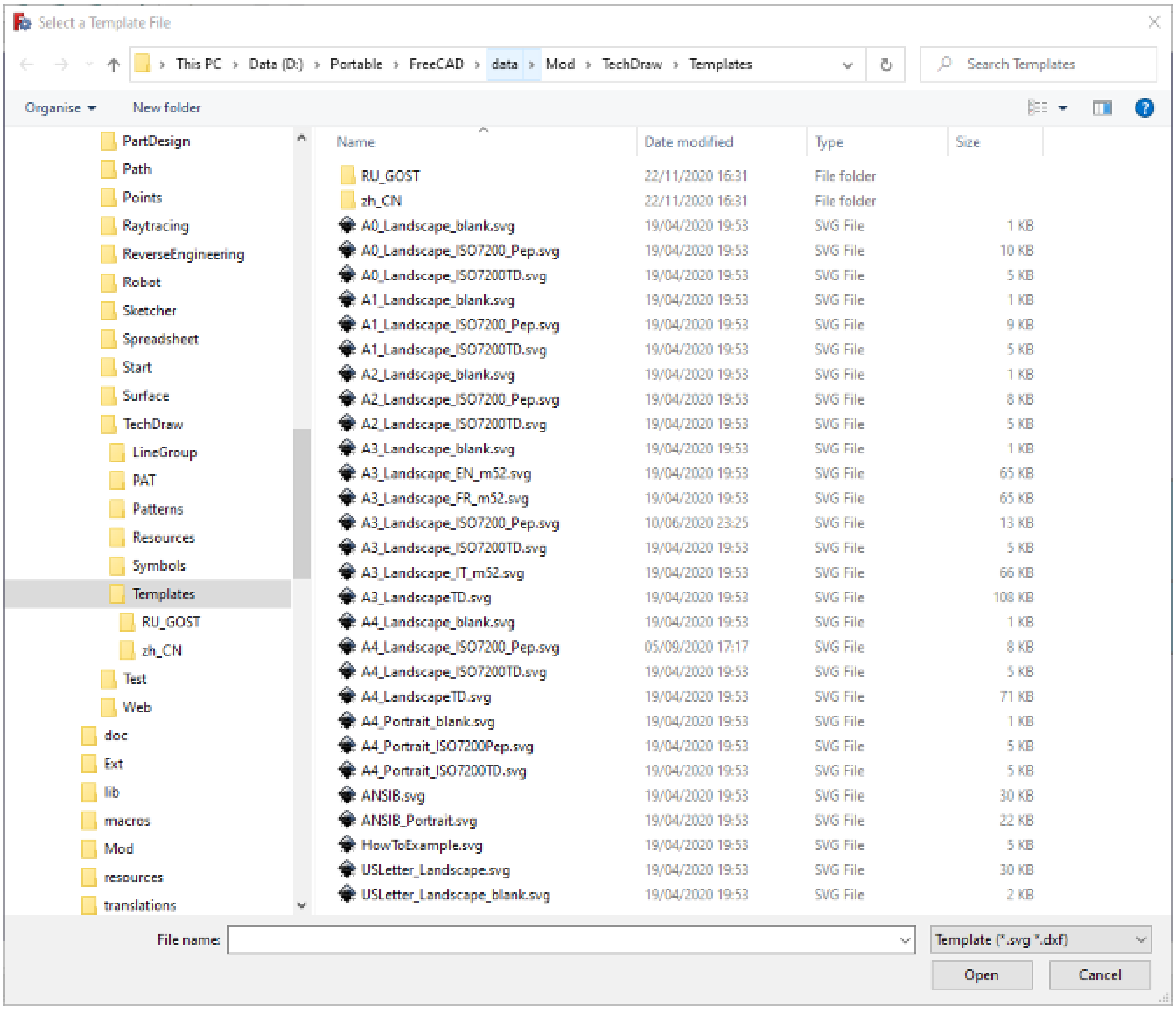
Note: if it did not update right-away, click on the Dimension-line in the tree-view.

All-over again but better and faster

Having played with the TechDraw tools a little, let's delete your practice drawing and start again, looking at a couple of things that might make the process quicker now you know how some of the tools work.

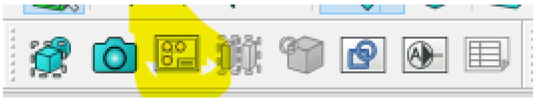
Select the page you were working on in the file tree and delete it. To create a new page for your technical drawing, let's click a different icon: the 'Insert Page Using Template' icon that looks like an orange folder.

This opens a folder containing different templates for technical drawings.

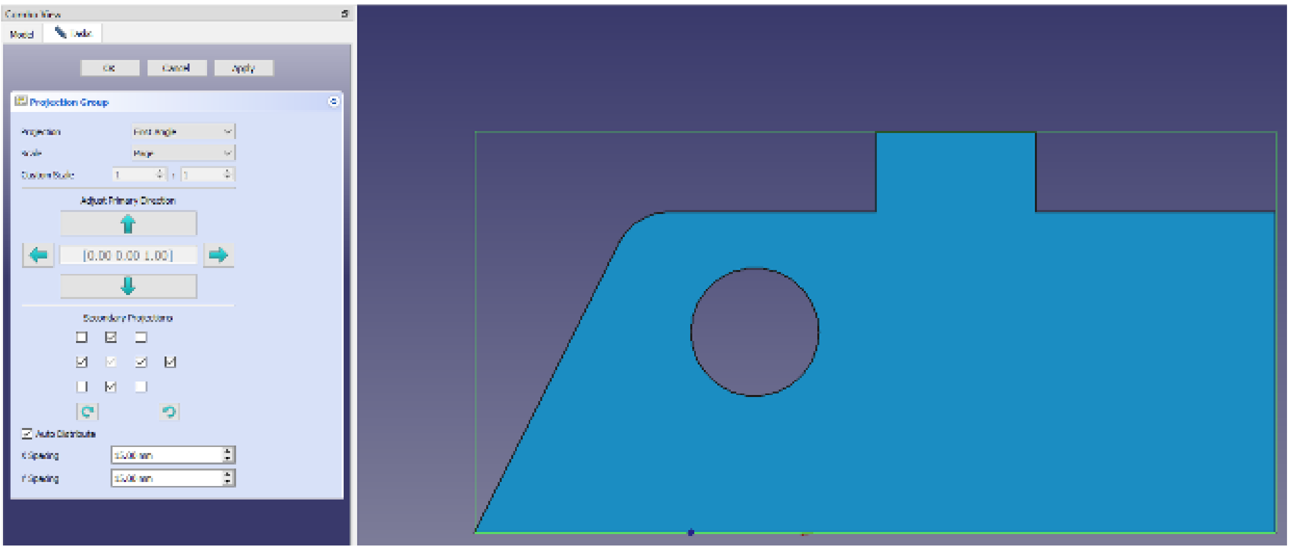


There are some things that will appear obvious to you, like there are different page sizes: A4, A3, A1, etc. There are also some templates that adhere to particular technical standards, such as the American National Standards Institute (ANSI) and the International Organisation for Standardisation (ISO), which may be of use if you want to make standard-compliant drawings. Let's use a page called 'A3_LandscapeTD.svg'.

Make sure that the 3D object is in the top view in the preview and highlight the body in the file tree. This time, instead of clicking to insert a single view, let's use the 'Insert Multiple Linked Views Of Drawable Objects' tool icon, which should be next to the template folder icon.

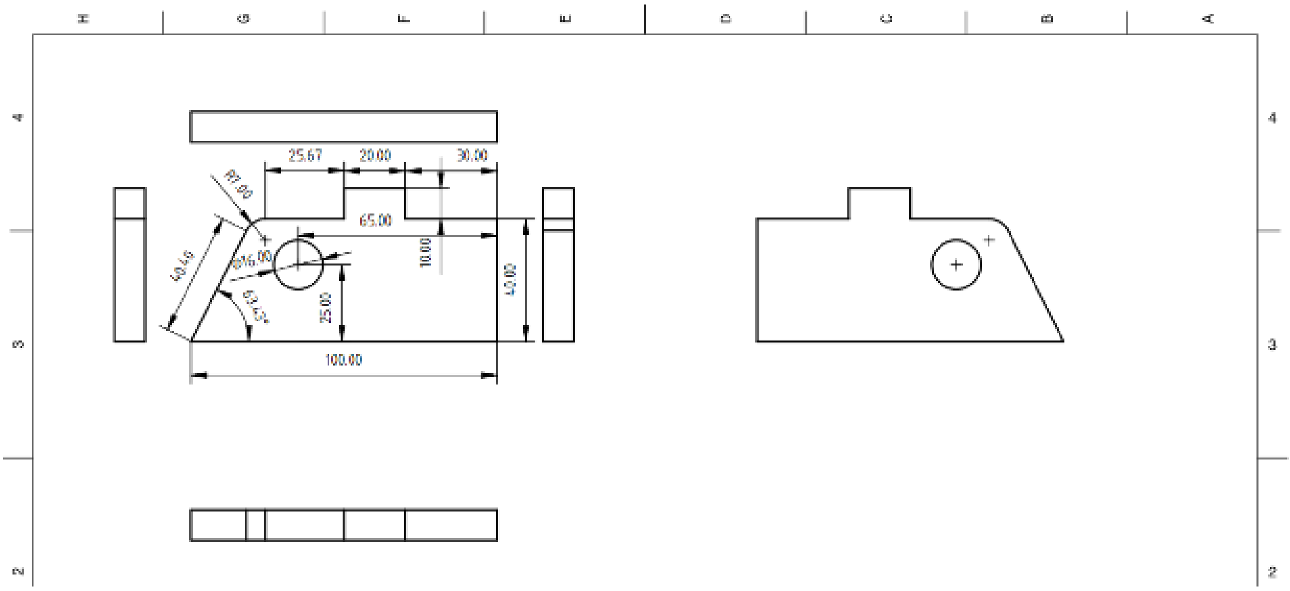


If you now click to the 'page' tab, you can see any changes you make in the dialog box live on the template.



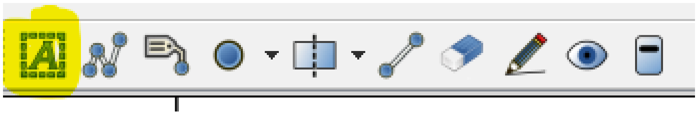
In the dialog box, the first thing you will see is a projection menu. Let's use first angle, as your template indicates the drawing will be in first angle projection.
For this example, you don't need to scale the drawings as it's quite a small object. However, you can play with selecting a custom scale and changing the values, which is useful if you want to scale down a large object on the plans. Further down you will see an arrangement of checkboxes that correspond with the multiple views that will be added to your drawing.
Select the pattern you have selected and click OK. On the technical drawing, you should now have the corresponding views automatically inserted.

To move linked views is slightly different as it always aligns the projection of the main views to the centre view. To move all the views as a group, move the centre view; to move the upper, lower, and side views, you can push and pull them relative to the centre view. The exception is your isometric view, which you can move anywhere in the page.

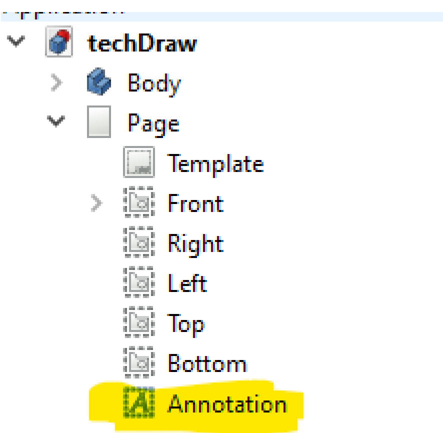


Annotations

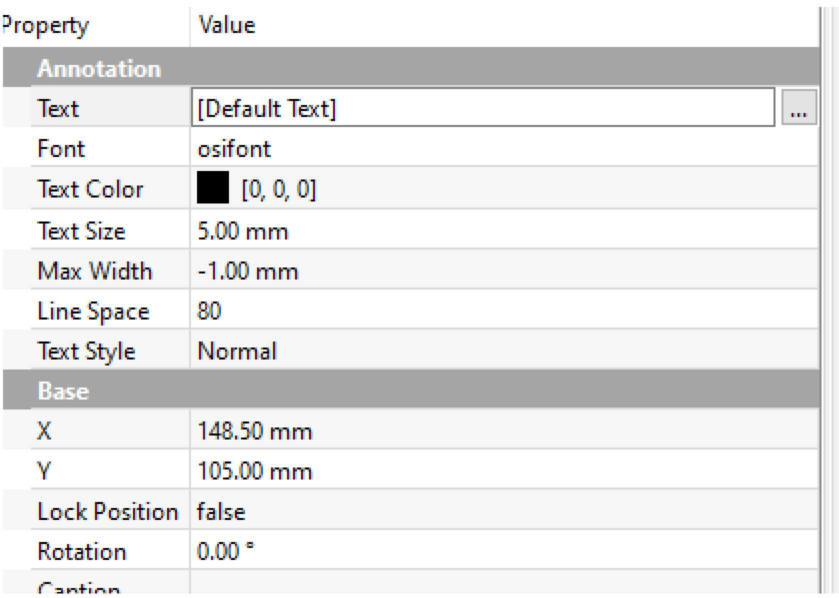
Most technical drawings will have some details added as text. In your default template, you have some text boxes set up in the lower right-hand corner. These include common details you might wish to add to a technical drawing, including the title, designer name, date, scale of the drawing, and more. With the view frames turned on, you should notice that each text area has a small green block. Clicking on the green block allows you to edit and insert the text that you want to add. You'll also notice that there are some blank boxes with no text inserted. You can add text anywhere on the document by using the 'Insert Annotation' tool.



Clicking the tool, you should see a view frame appear labelled 'annotation' with 'default text' written in it.

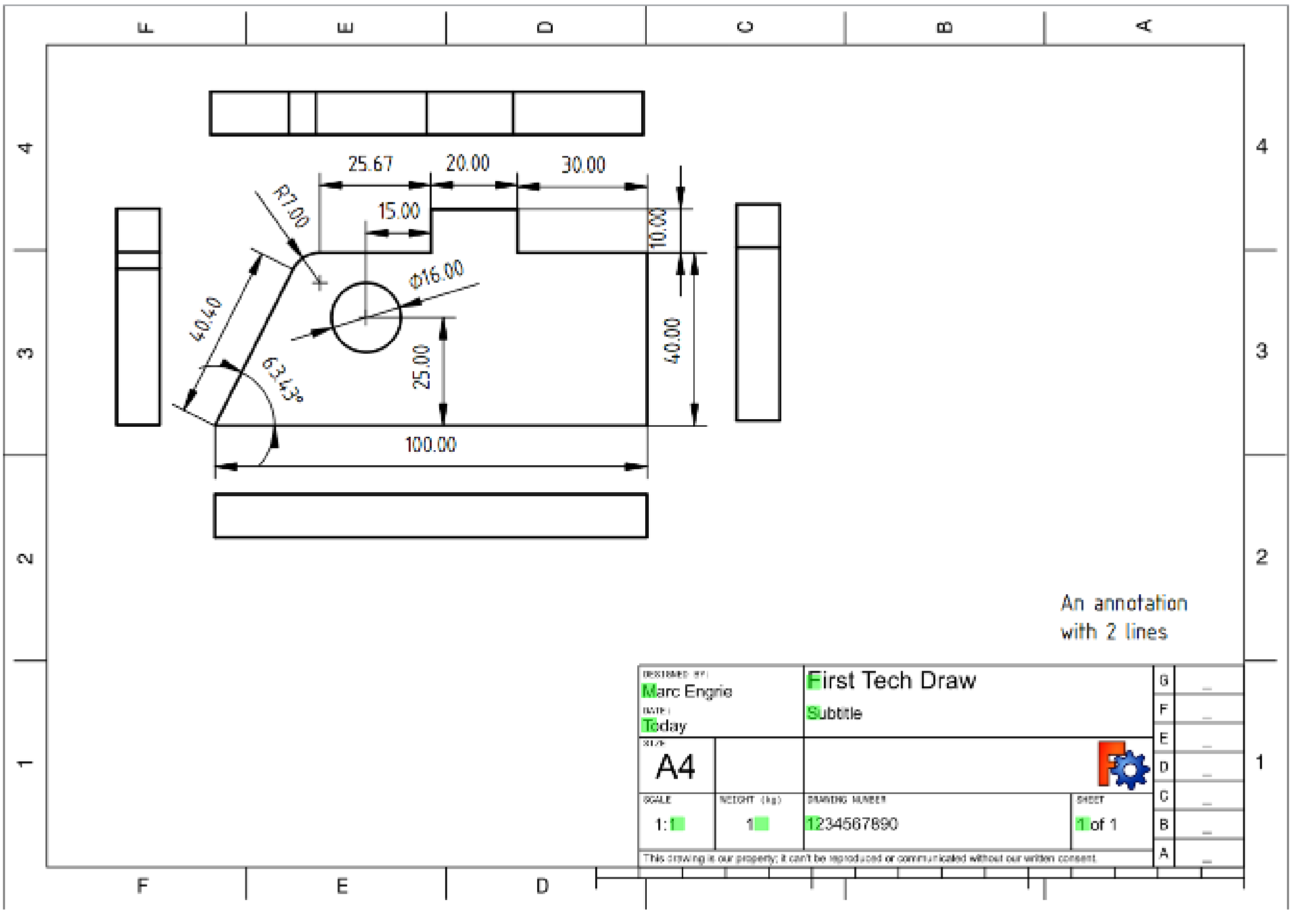


To edit the annotation, select it in the file tree view. In the combo view, you should now see the editable parameters of the annotation, which include common text editing options like font and size. At the top of the annotation parameters, you should see 'Text' and the 'Default Text' that is stored within it. Click the ellipsis icon (the three dots to the right) and in the pop-up window you can input text.



Property	Value
Annotation	
Text	[Default Text] ...
Font	osifont
Text Color	[0, 0, 0]
Text Size	5.00 mm
Max Width	-1.00 mm
Line Space	80
Text Style	Normal
Base	
X	148.50 mm
Y	105.00 mm
Lock Position	false
Rotation	0.00 °

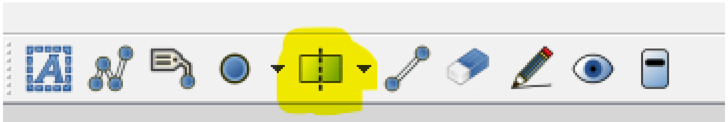
Note that it doesn't use text wrapping, and you need to hit your keyboard Enter-key to create a new line of text. In the image above, you have filled in the default drawing details with the editable text boxes, but also added some text in an annotation box outside the default drawing text boxes, as you can add an annotation frame anywhere in the drawing.



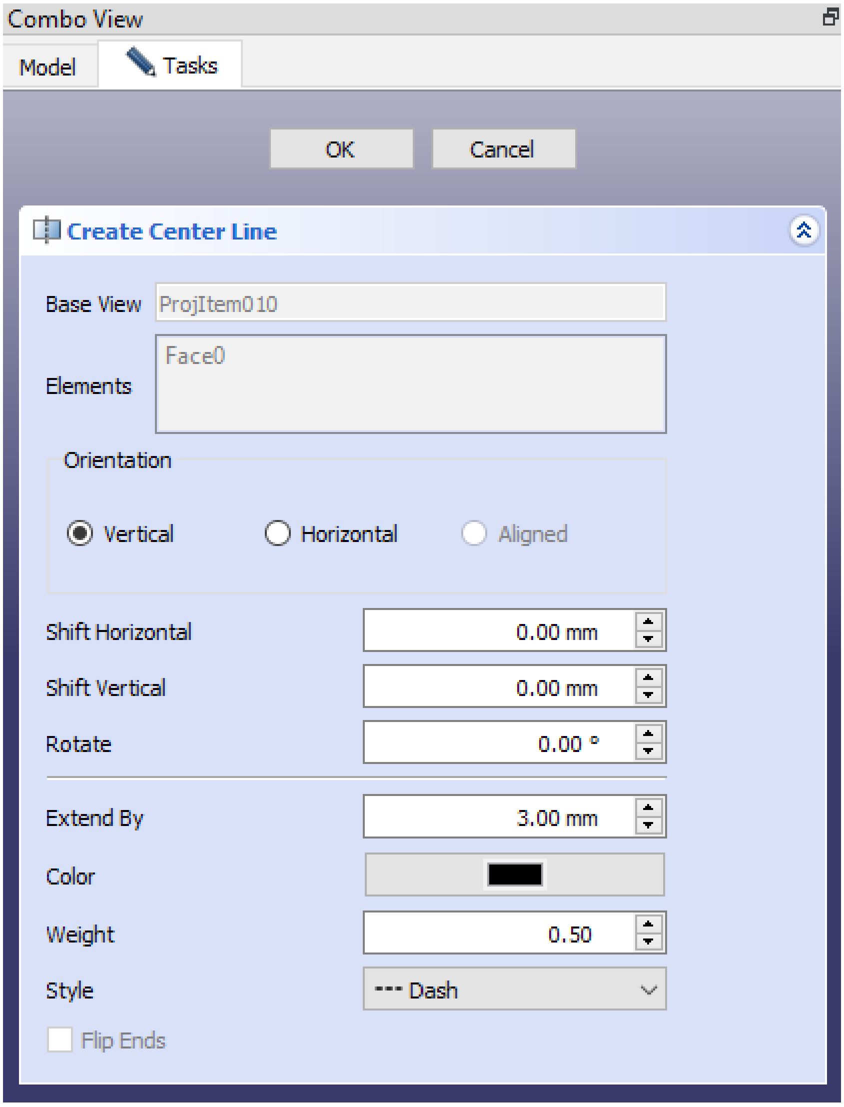
Note: To move any object in the TechDraw, also annotations, turn View Frames on and you can drag the desired object with the mouse.

Getting centred

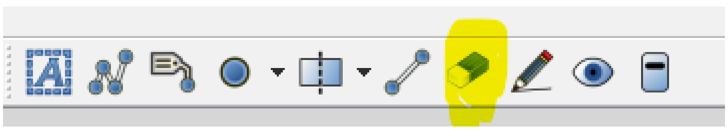
Technical drawings often use centre lines and hatching and other effects to make them more readable. Adding a patterned centre line is possible, and you can use a face, two vertices, or two lines as a reference. As a quick example, click the face of your object in the front view to select it and then click the 'Insert Centre Line' tool icon.



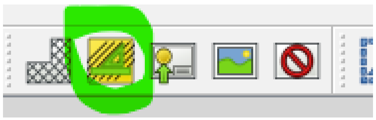
In the dialog box you can adjust the position and appearance, but for now just click OK and a dashed line should appear vertically through your object.

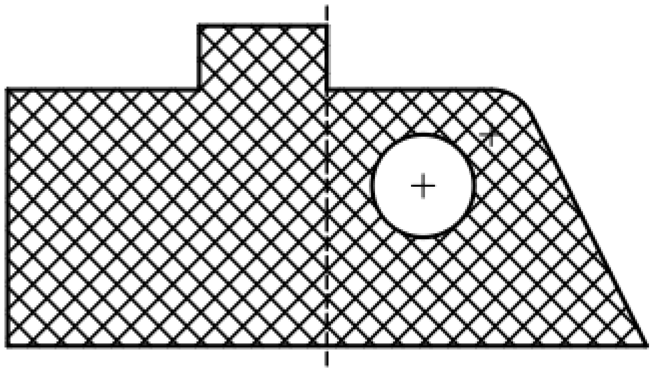


A centre line is classed as a 'cosmetic object' and as such doesn't appear in the file tree. To remove the centre line you just created, you need to select the centre line and then click the 'Remove Cosmetic Object' icon, which looks like an eraser.



To add hatching to a face, select the face and then click the 'Apply Geometric Hatch To Face'. In the dialog you can change various settings such as the pattern, line weight, and colour.





Quick Tip: To export and print your technical drawing at any time, simply right-click anywhere in the drawing

As ever with FreeCAD, there's heaps more you could look at on this workbench alone, but hopefully you are well on the path to creating beautiful and useful technical drawings for all your project needs.

Extra Information on Angles

Technical drawing in itself is a massive field to learn about, and there are numerous standards and systems in place governing technical drawings. One set of standards relate to the projection angles which affect where views are positioned in a technical drawing. These two projection views families, 'First Angle' and 'Third Angle', are both used but create a technical drawing laid out differently.

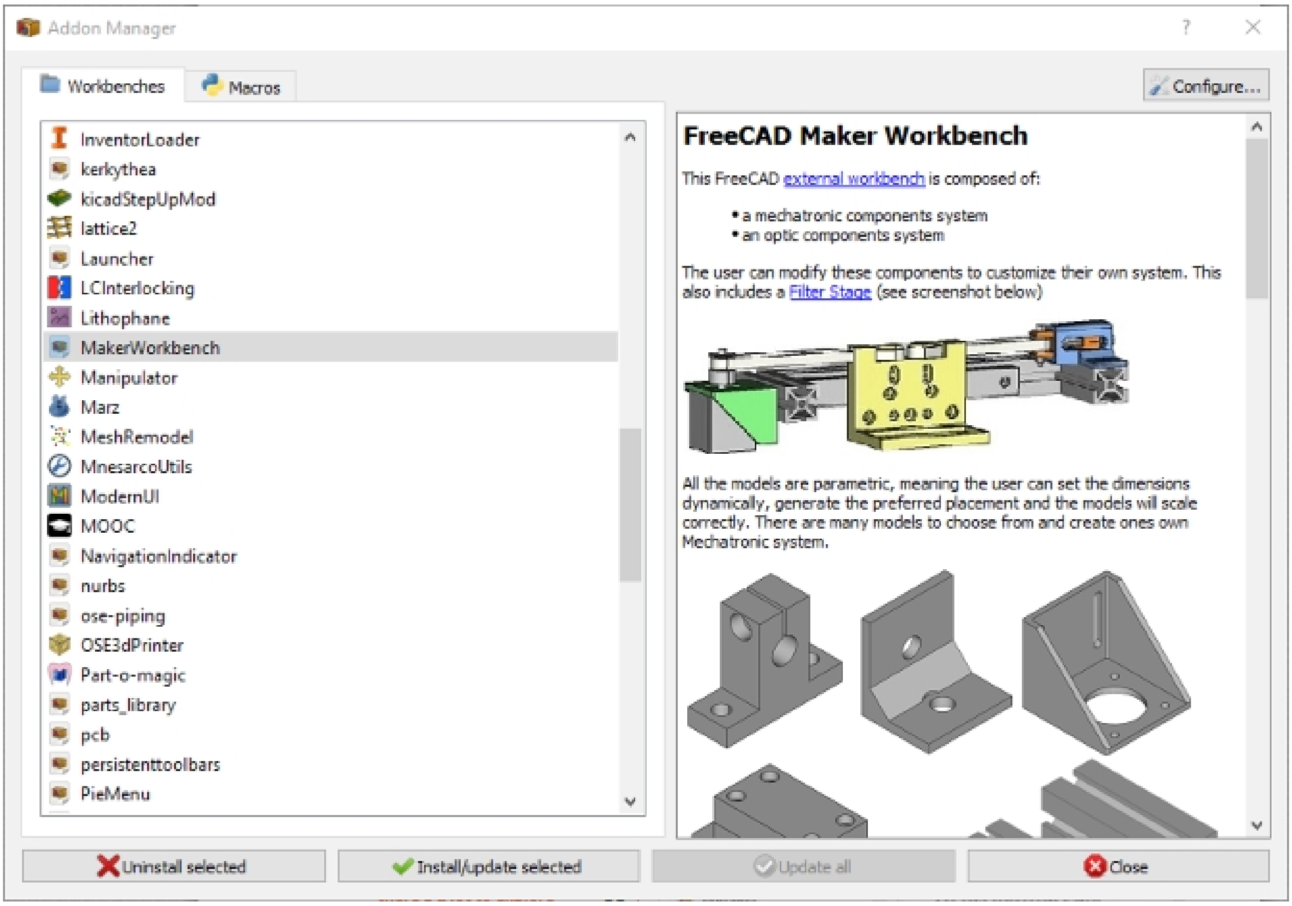
First angle drawings are more common outside of North America, whereas in North America third angle is more common. It's important that you indicate that you are using a first or third angle projection on a technical drawing and that you only use that type of view and don't mix them. Commonly on technical drawings, you indicate whether a drawing is first or third angle by adding the correct icon, as seen in the image below.

It's quite complex to explain the difference, and there are numerous different approaches to explaining first and third angle. A good place to start is this Wikipedia page: https://en.wikipedia.org/wiki/Multiview_projection#Third-angle_projection

Add-on workbenches and Laser Cut Interlocking

After looking at how you install additional workbenches, you will focus on using an excellent workbench called 'Laser Cut Interlocking' or 'LCInterlocking', which offers some interesting and useful tools for those working on designs destined for a laser cutter or CNC router.

To explore the amazing array of add-on workbenches, you'll need to be connected to the internet. You can view the available add-on workbenches from any workbench in FreeCAD by simply clicking Tools > Addon Manager.
The Addon Manager will then check all the available add-on workbenches – it may take a few seconds for the list to appear as it checks it's up to date. You should end up with a list that looks like this.



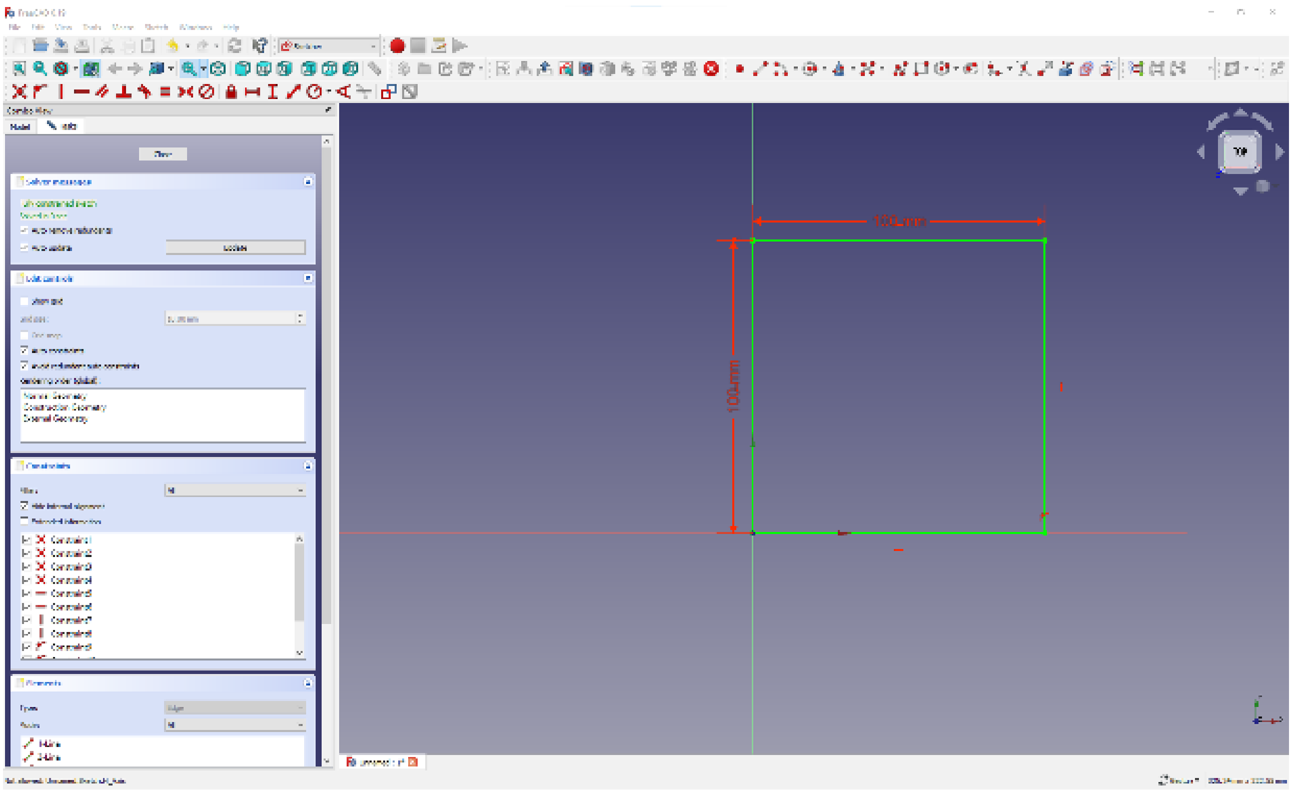
Before you install the workbench we'll be using, scroll around the list. If you highlight a workbench on the list, information about it will appear on the right-hand side of the window. It's incredible what workbenches there are! Like all of FreeCAD, the add-on workbenches listed in the Addon Manager represent many thousands of hours of voluntary contributions by FreeCAD community developers, covering a vast array of subjects and interests.
From aeroplane design tools to computational fluid dynamics to 3D print slicing engines – there's a lot to explore.

Installing a workbench is as simple as selecting it in the list and then clicking Install. Behind the scenes, FreeCAD connects to the workbench code repository online and downloads and installs the latest version. It's then simple to update a workbench when new features are released. It's just a case of clicking 'Install/update selected' on the relevant workbench in the Addon Manager.

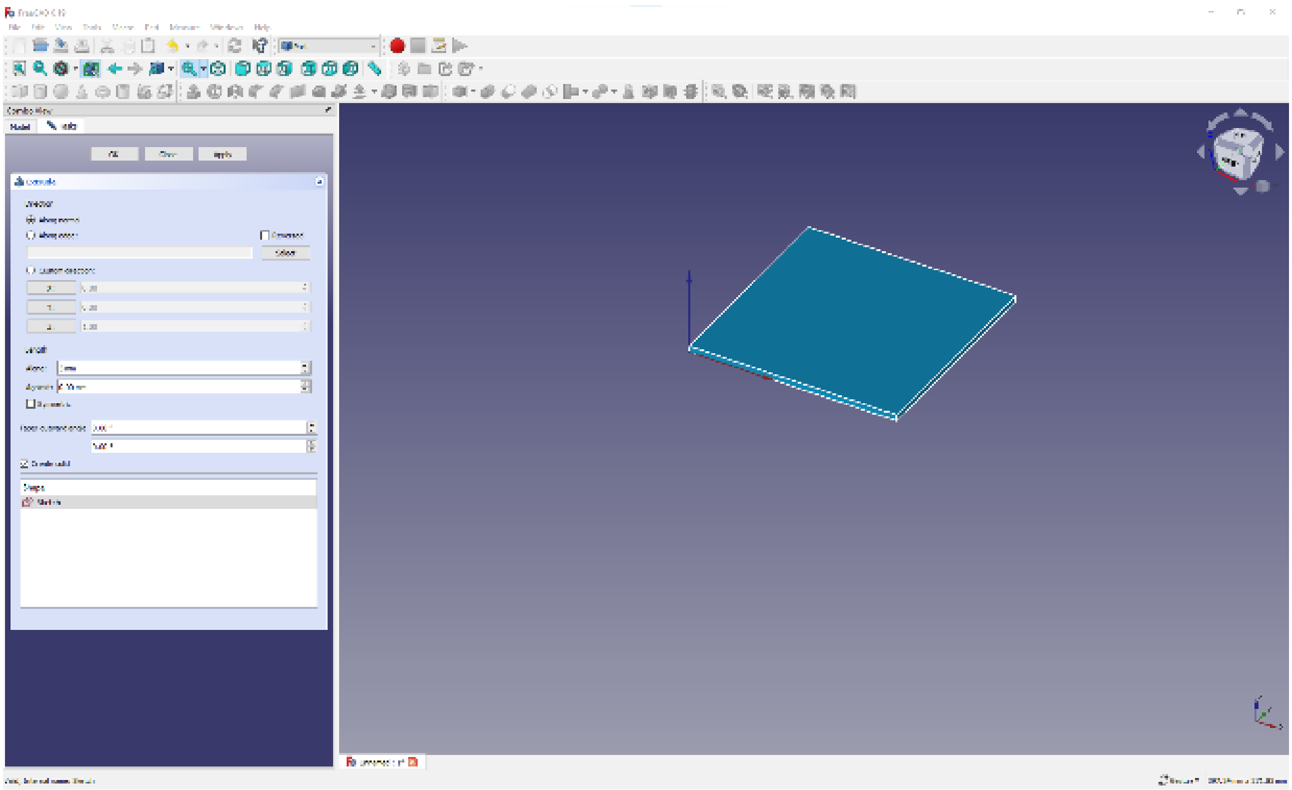
The workbench you are going to look at is called ’LCInterlocking’ – it’s described as the ’FreeCAD Laser Cut Interlocking Module’. Go ahead and click install. Once installed, the Addon Manager will prompt you to close and restart FreeCAD to finish the installation. Once FreeCAD has restarted, let’s create a small project to explore the tools on this new workbench.

The LCInterlocking workbench has tools that allow a collection of parts in FreeCAD to be pulled apart and laid out flat. It then has tools that enable us to create a two-dimensional projection of the part shapes that you can export in various file formats (DXF, SVG, etc.). While it’s predominantly, as the name suggests, aimed at laser cutting, it also can be useful for other processes, such as preparing files and objects to set up toolpaths for CNC routing.

The ’interlocking’ part of the LCInterlocking name hints at another handy feature: the workbench has tools that can automatically create tabbed joints in objects where one part has tabs and the other part has receiving slots.
To explore these tools, let’s create a collection of parts that form the base and sides of a simple box. In the Sketcher workbench, draw a sketch in the XY plane of a square that is 100 mm on each side, and let’s constrain it with the lower-left corner onto the 0,0 point of the axis.



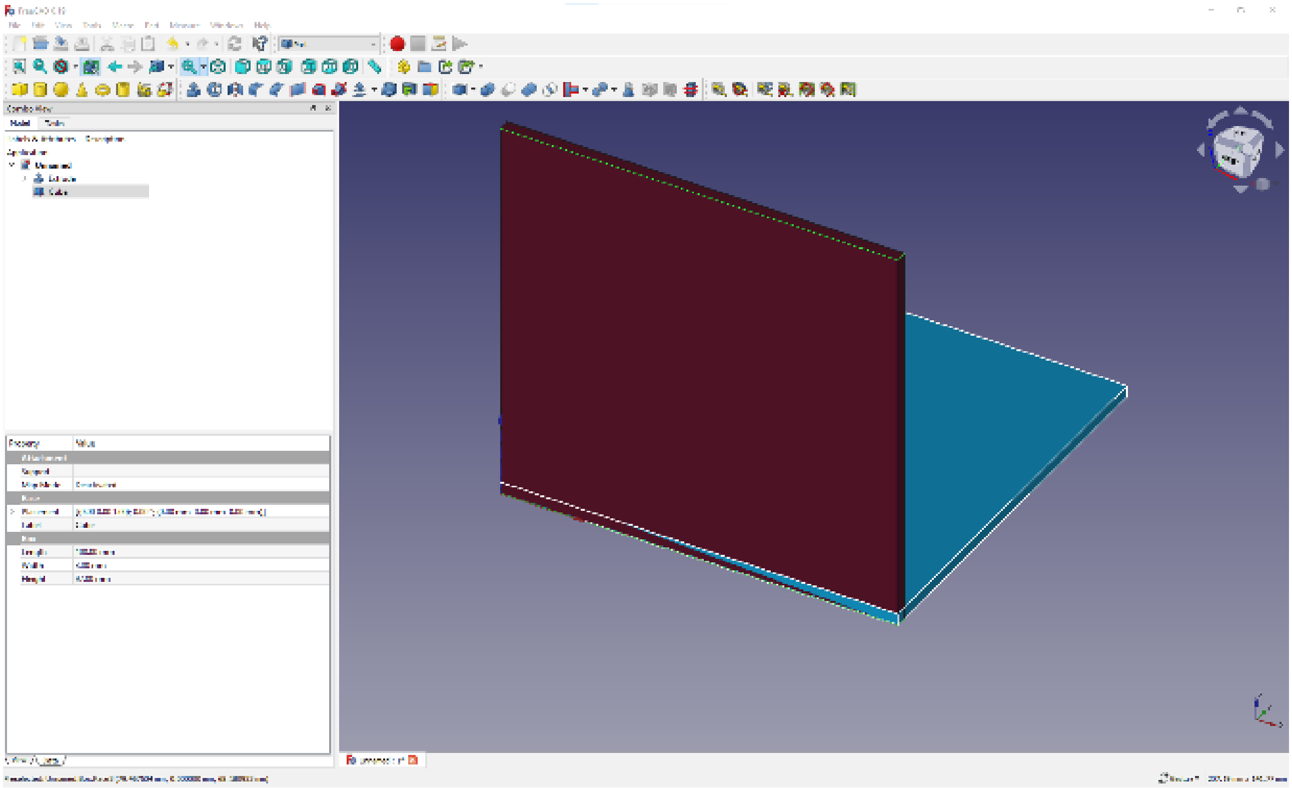
Having drawn the square sketch, use the ’extrude/pad’ tool on the Part workbench to extrude the sketch to a thickness of 3 mm. Using 3 mm means you can imagine you are going to laser-cut your tabbed box out of 3 mm plastic or plywood.



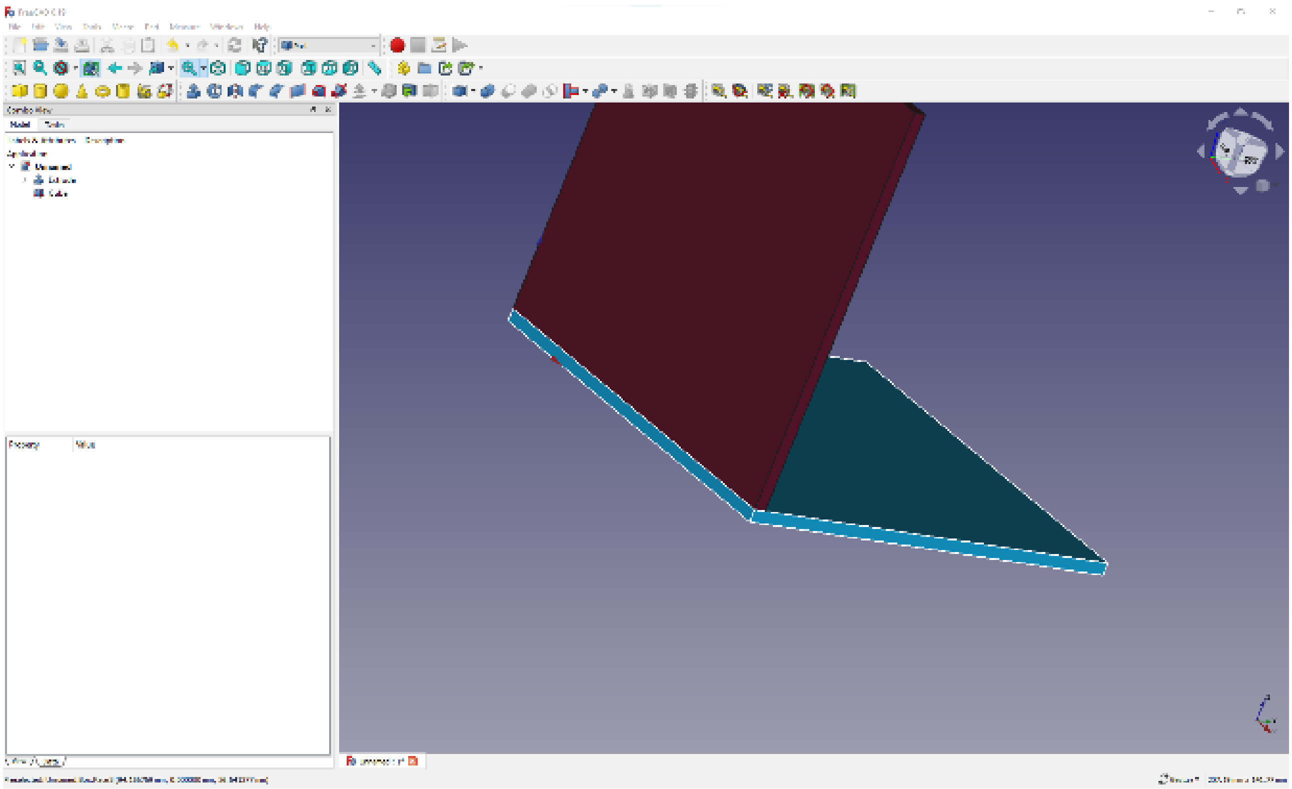
You could make this base part in a much easier way – namely, you could use the ’Create a cube solid’ tool. However, as an example, you wanted one part to be made from an extrusion as you need to treat extruded parts a little differently when you get to the LCInterlocking workbench.

You can use the cube generator to create the sides of your box, however. First, let’s make a cube by clicking the ’Create a cube solid’ tool.

Double-click the object in the Combo View file tree and then change its dimensional parameters. Let’s make the length 100 mm, height 97 mm, and width 3 mm.

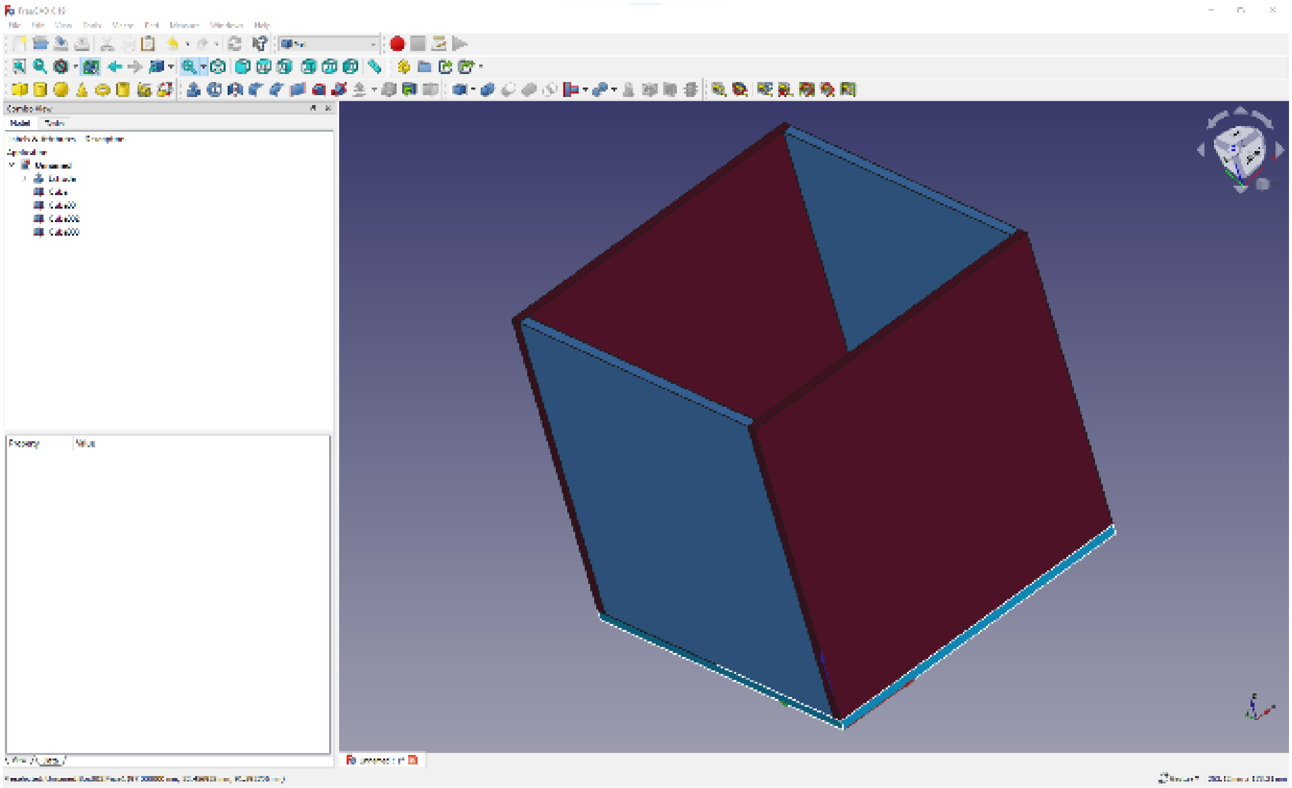


Next, let's right-click this cube in the file tree and then select Transform – use the arrows in the preview window to bring the side up on the Z-axis by 3 mm so that it sits exactly on top of the base you made earlier.

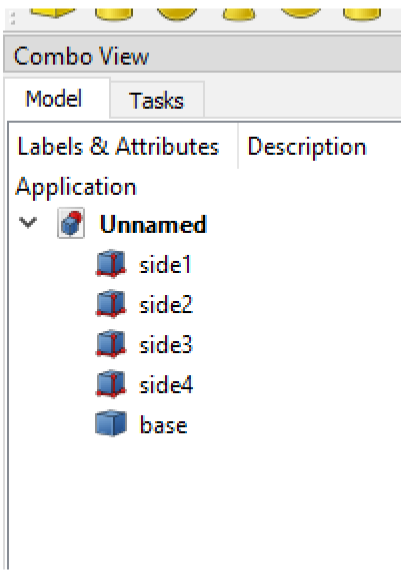


Make the opposite side of your box by either repeating the process and moving it to the opposite side, or simply copy and paste and move a duplicate of the first side.

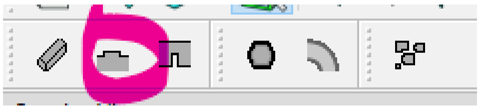
Next, repeat the process to make the final two sides of the simple box, making the height 97 mm, width 94 mm, and length 3 mm. Move these in a similar way using the 'Transform' tool to move these sides into the correct positions lined up with the edges of (and sat on) the base.



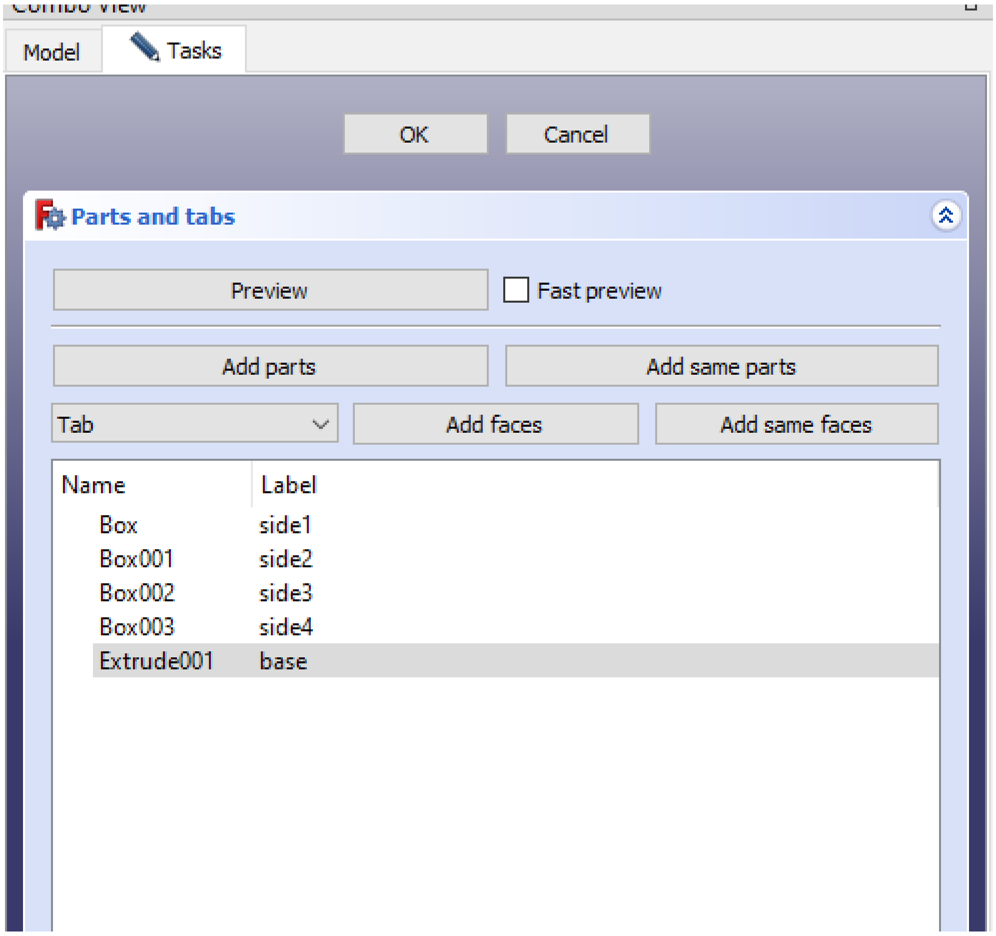
Before you move over to the LCInterlocking workbench, you need to convert the base into a simple part. LCInterlocking prefers to work with parts that are simple objects rather than parts that appear as a stack of operations in the file tree view. You could just draw the base with a cube, but if you're working with more complex shapes, this might help you remember to convert parts into simple copies before opening the LCInterlocking workbench. So, before proceeding, highlight the base part in the file tree, and click Part > Create a Copy > Create a Simple Copy. Delete the original extrusion and sketch in the file tree to keep things simple and clutter-free. It's a good idea to right-click on the simple copy of the base and rename it to something recognisable, such as 'Base'.



Next, let’s move to the LCInterlocking workbench by selecting it from the drop-down menu. The first task in the new workbench is to create some tabs and slots to ensure strong joints in the box.
To get started, highlight the five parts that form your open box in the file tree, and then click the grey ’Interlocking’ tool icon.

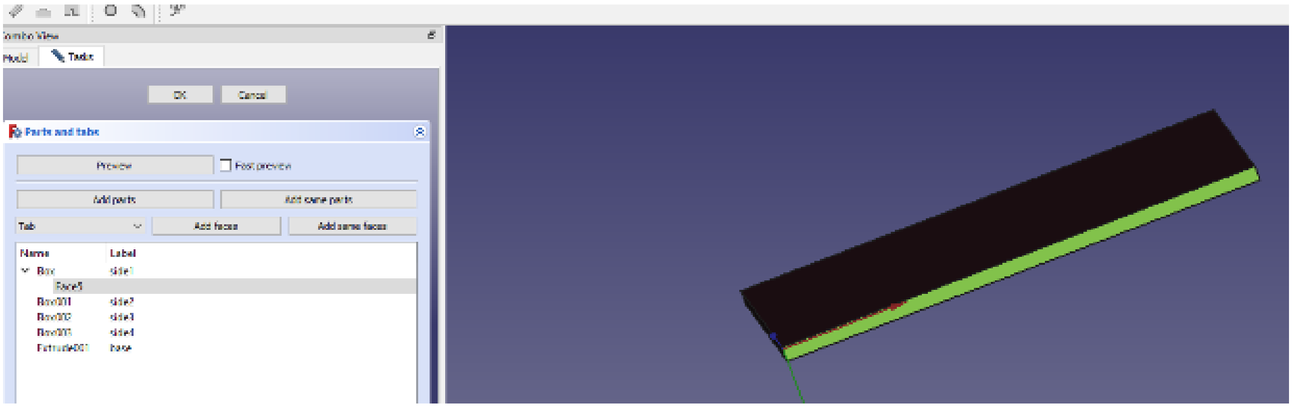


In the dialog box that appears, you will see a button that says ’Add parts’.
Click this button and you will see the five selected parts appear in the window.



This list of parts in the dialog box is similar to the file tree in the combo view in that you can highlight anything in the list – you can press the SPACE bar to toggle visibility. This is very handy for the next task. You need to tell the ’Interlocking’ tool which faces of your box are going to interlock. Let’s begin with the faces that will form the tabs that extend into the base. Select the ’Base’ part and then press the SPACE bar to make it disappear – do this for three of the sides.

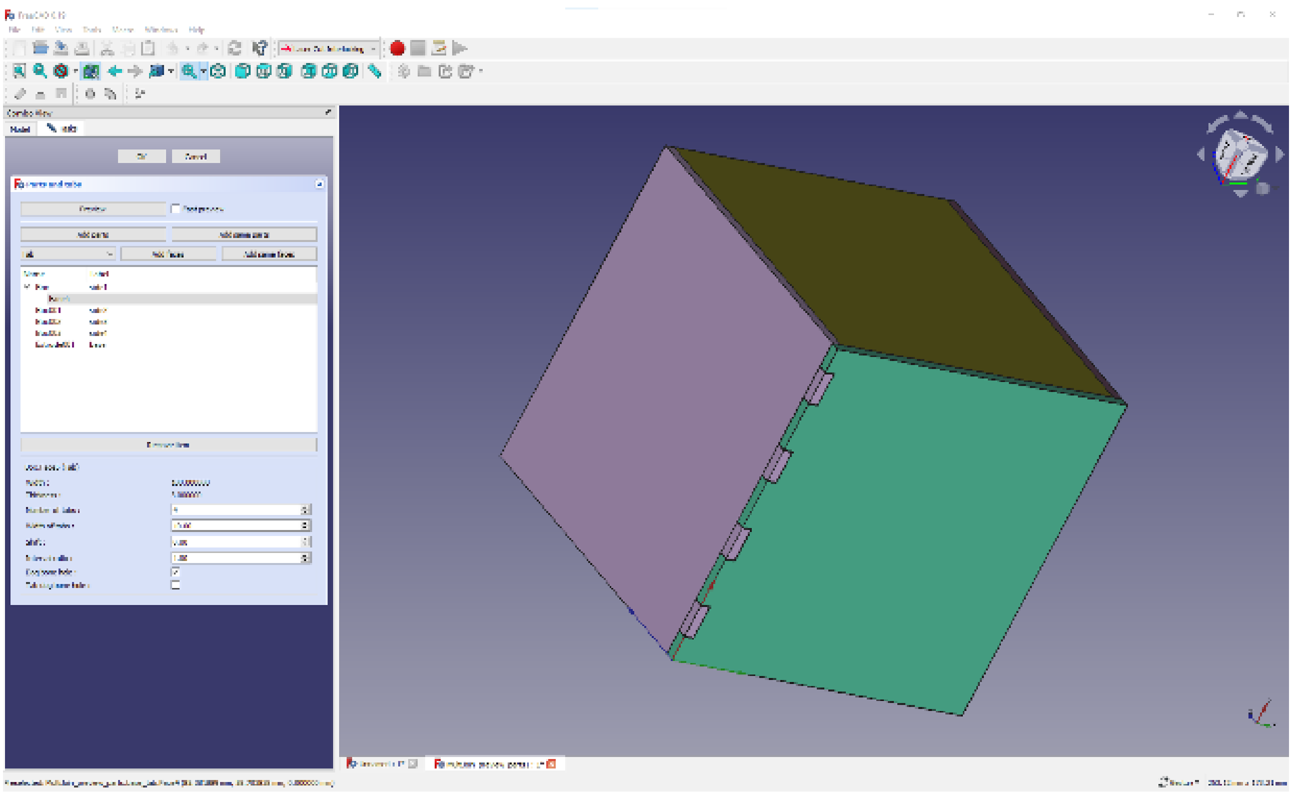
Next, select one of the thin faces on the remaining side of the box, making sure it is the face that is touching the base when the base is visible. With that face highlighted, click the ’Add faces’ button in the dialog. You should see the added face highlighted in the list, filed under the part that it is attached to.



If you look down in the dialog box, you will see a menu relating to that face.
In this, you can input the details of the tabs that you wish to generate on this face.
Add the number of tabs – you can make your own choices, but you went with four. You set the 'width of tabs' to 10 mm, 'shift' was left at 0.00, and the 'interval ratio' was set to 1.00. You'll see there's a couple of checkbox items: 'Dog bone hole' and 'Tab dog bone hole'.

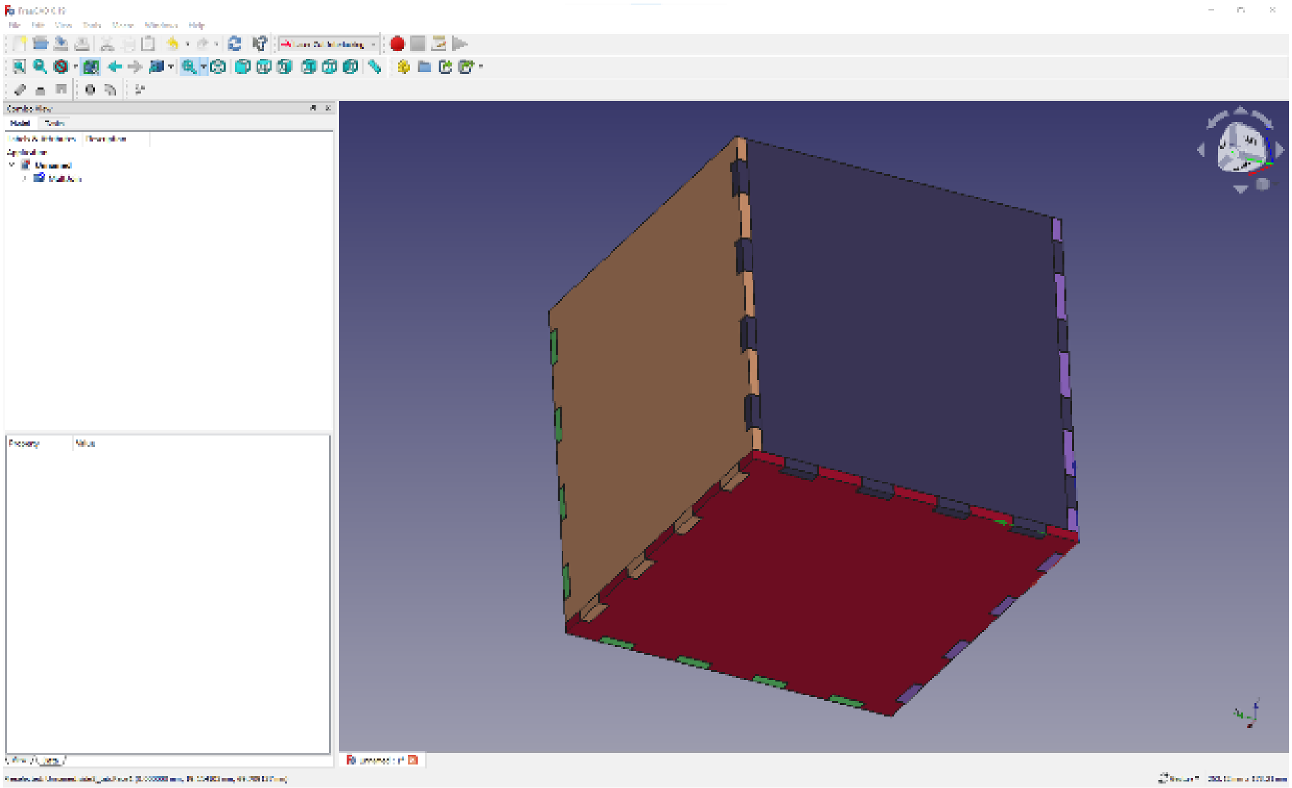
'Dog bone holes' are a method often used in parts that are CNC routed to get rid of a problem created by the fact that most router tools are round. If you send a round end mill cutter along a path to cut a 90-degree internal corner, you are left with the radius of the tool in the corner of the cut. If you are cutting tabs into material in this way, then the tabs won't fully seat into the slots due to this excess material. Dog bone holes are where the machine is instructed to cut a little further into the corners on both axes; this slight overcut creates two small, round cuts in a corner that look like a stereotypical dog's bone, giving the process its unusual name. You deselected both of the checkboxes as you don't need them for this example.

Having input the tab parameters, go back up to the top of the interlocking dialog box and click the 'Preview' button. This will launch a new tab in the preview window, and you should see that your box appears regardless of which parts are visible or invisible in your other tab. If you look at the joint between the base and your tabbed face, there are now four tabs interlocking into the base.

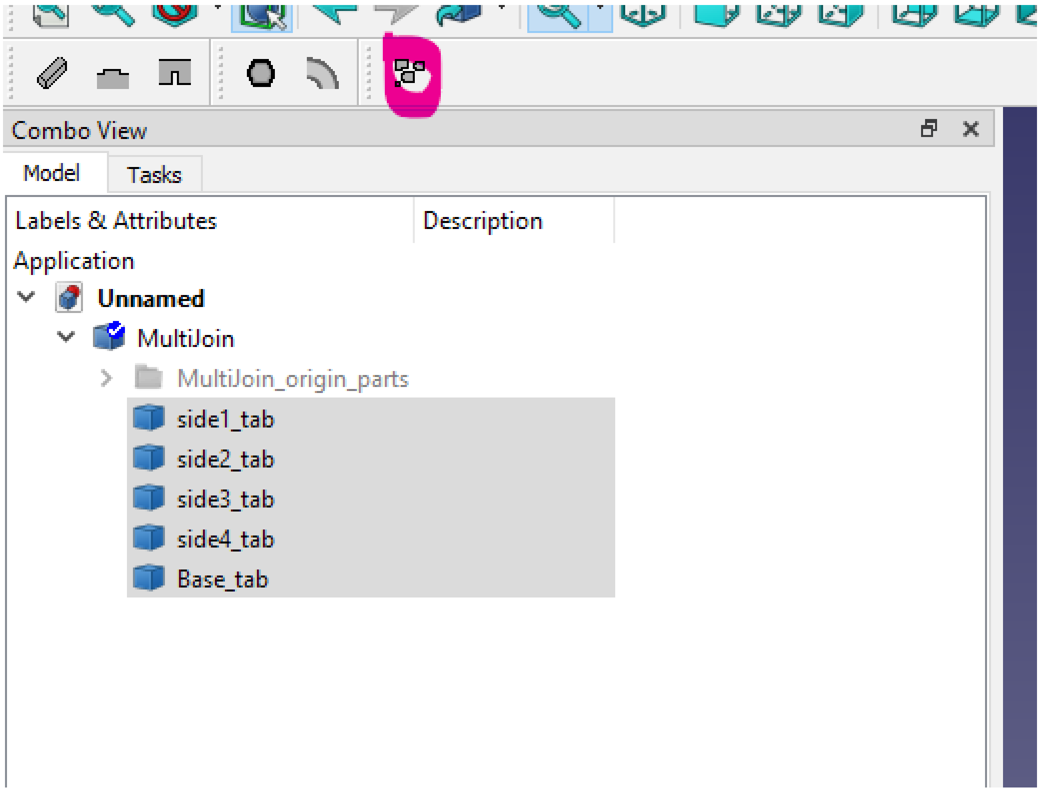


You can now repeat the above process of selecting and adding faces to the interlocking dialog and configuring tabs. Working your way through the model, you added tabs to the other three faces that interlock with the base, and then added tabs to two of the side panels to create tabbed joints on every interface of your box.

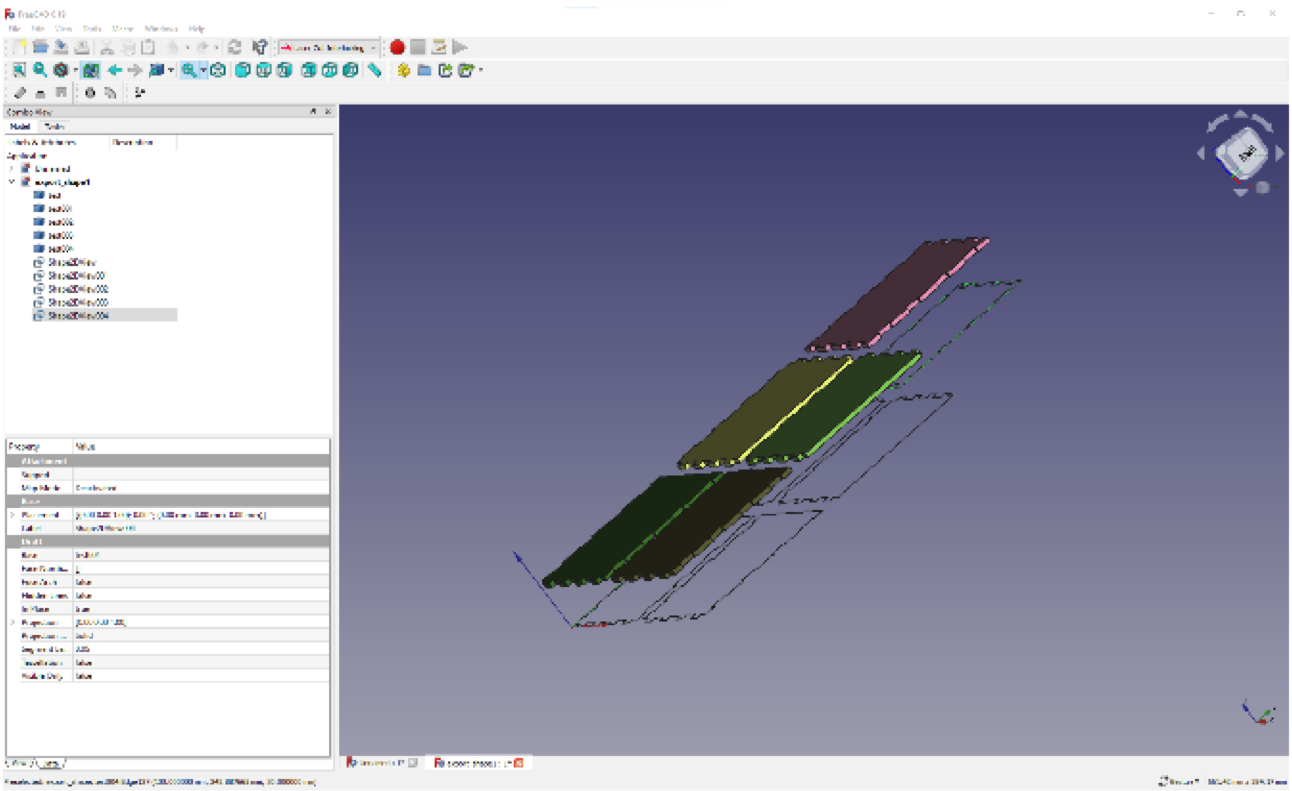
Once you are satisfied with your added tabs in the preview, click the 'OK' button to apply the interlocking tool to the parts. In the Combo View under your main document, there is now an object called 'MultiJoin'. Within this file tree, there are now parts that are the tabbed version of your original parts, and there is also an invisible folder that contains the original non-tabbed parts.
The preview tab also stays available – you can save this as a separate project if you like; however, you can close the preview tab without saving it for this example.



The next task is to apply the 'Export' tool to the tabbed parts you have generated. Rather than actually export a file, this tool explodes your box apart, lays the parts out as flat parts, and automatically creates a drawn shape projection of the parts. In the file tree, select all the new tabbed parts you created under the MultiJoin object, and then click the 'Export' tool icon.

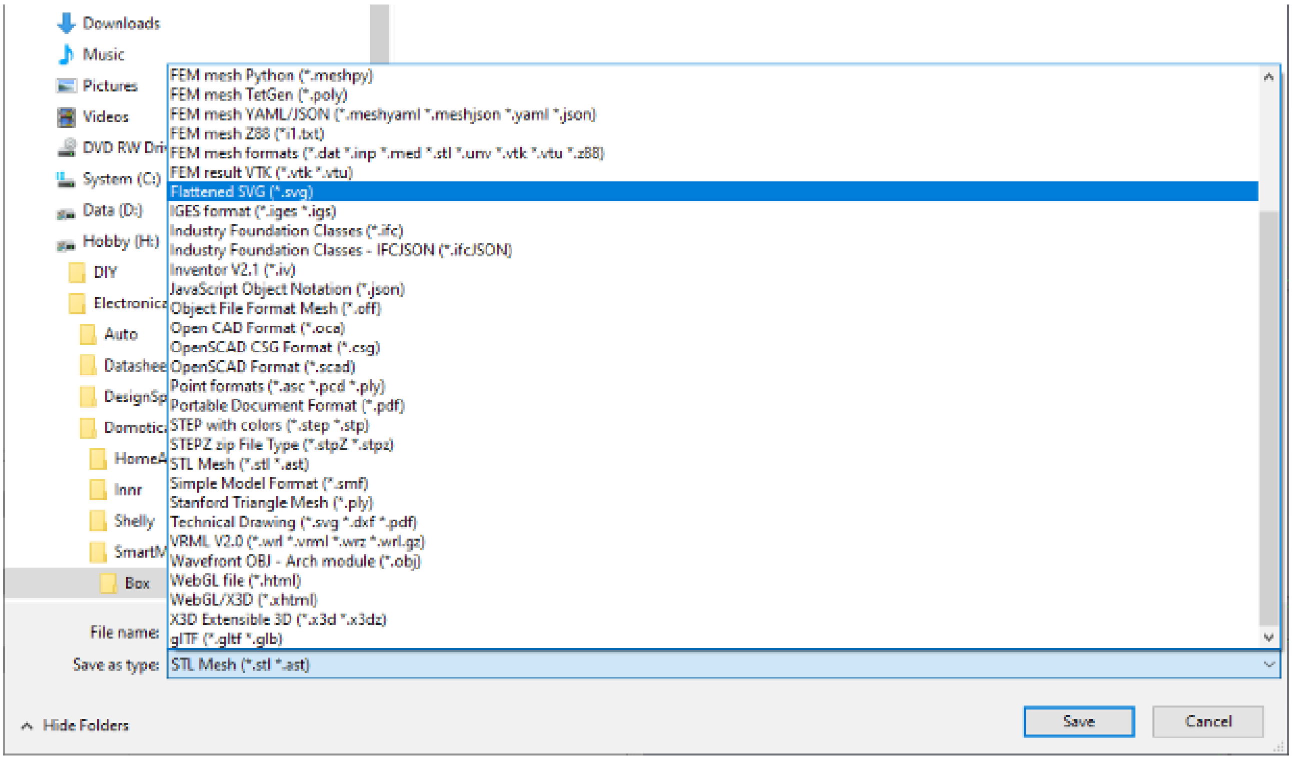


Once again, FreeCAD will open a new preview window tab – this will contain all the parts of your box separated and laid out on a single plane. You will also see some new objects in the file tree if you rotate the plane in the preview, as you have like this.

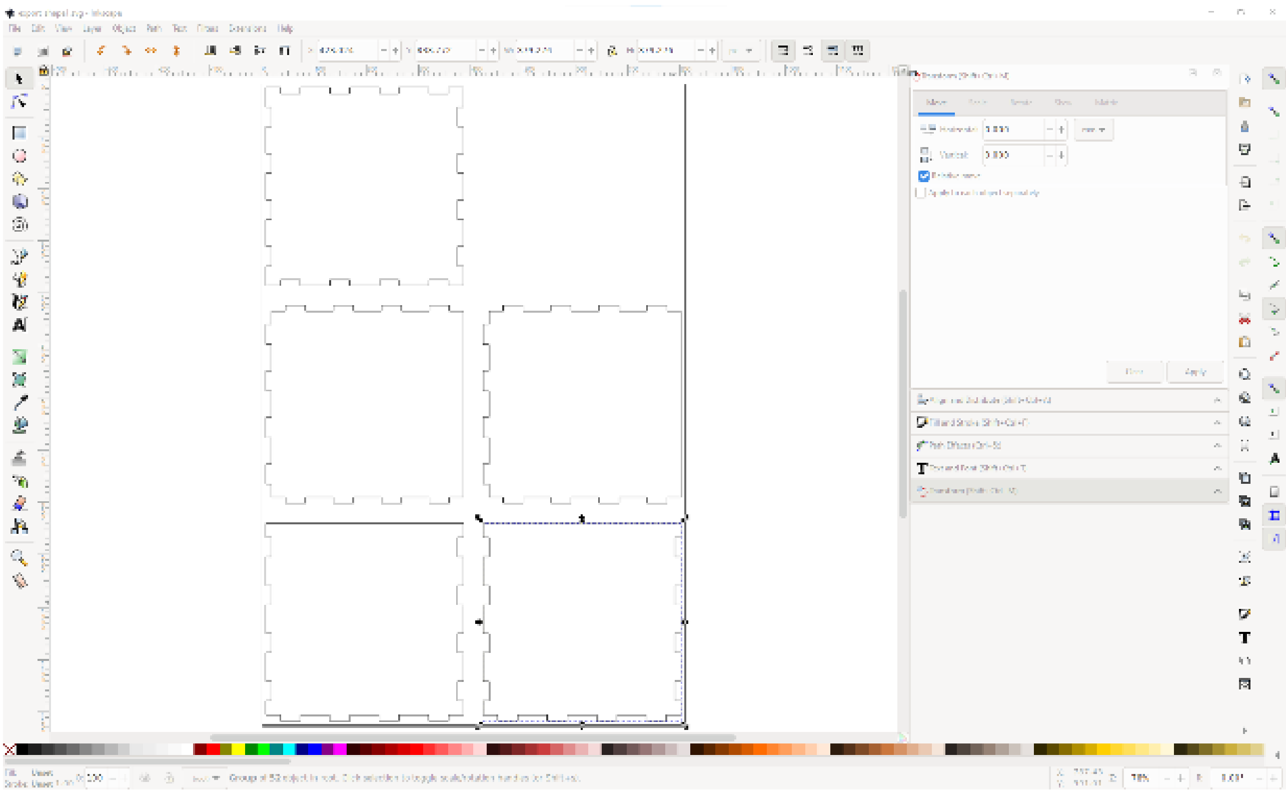


These are the ’Shape2DView’ objects – line projections of the outline of your parts. While this might initially seem confusing, having both versions of the parts is incredibly useful, depending on what you would like to do to create the parts.

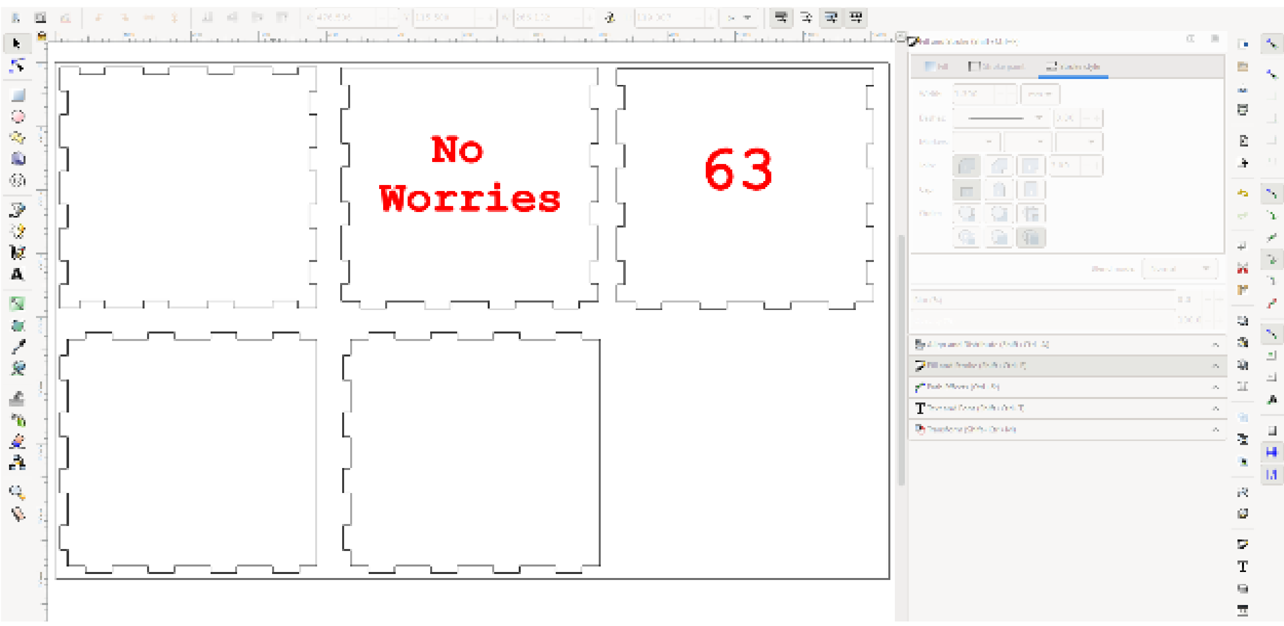
Using the Shape2DView parts, you can create 2D output files that are suitable to cut this design using a laser cutter. First, in the file tree, let’s toggle the visibility of the 3D parts so that you just have the Shape2DView objects visible in the preview. (Hint: select part and hit spacebar)
Next, highlight all the Shape2DView parts in the file tree, and then click File > Export.
In the export dialog, select a location to export to, and then from the drop-down menu, select ’Flattened SVG’ as the file type. Give your export a name followed by ’.svg’ and then click the Save button.



You should now have saved an SVG file containing a vector line drawing of the parts you created. The beauty of an SVG export is that you don’t lose any dimensional accuracy, and you can open this file for editing in many graphics packages. You can use open-source Inkscape package to take a quick look at your SVG.
Find your file and open it in Inkscape.



Note that you can select and move each of the parts individually. This is excellent, as often you will want to move parts around on an Inkscape canvas to arrange your components so that you are as economical as possible when cutting the parts on your laser cutter.
That might be the only edit you need to make, but often you may need to customise the SVG file to work with the settings of your laser cutter. You can think of a couple of different laser cutters that you have access to that have slightly different requirements. One needs any line that the laser is going to perform a cutting/vector action to be a 'hairline' thickness of 0.0254 mm or thinner. Another machine isn't worried about the thickness applied to a stroke on a path but wants all cut lines to be red in colour. These changes are trivial to make in Inkscape.
Of course, as a fully-fledged vector graphics package, Inkscape can also be useful to embellish your parts with decorations that the laser cutter will engrave/raster; again, this is simple to realise in Inkscape, as you have done.

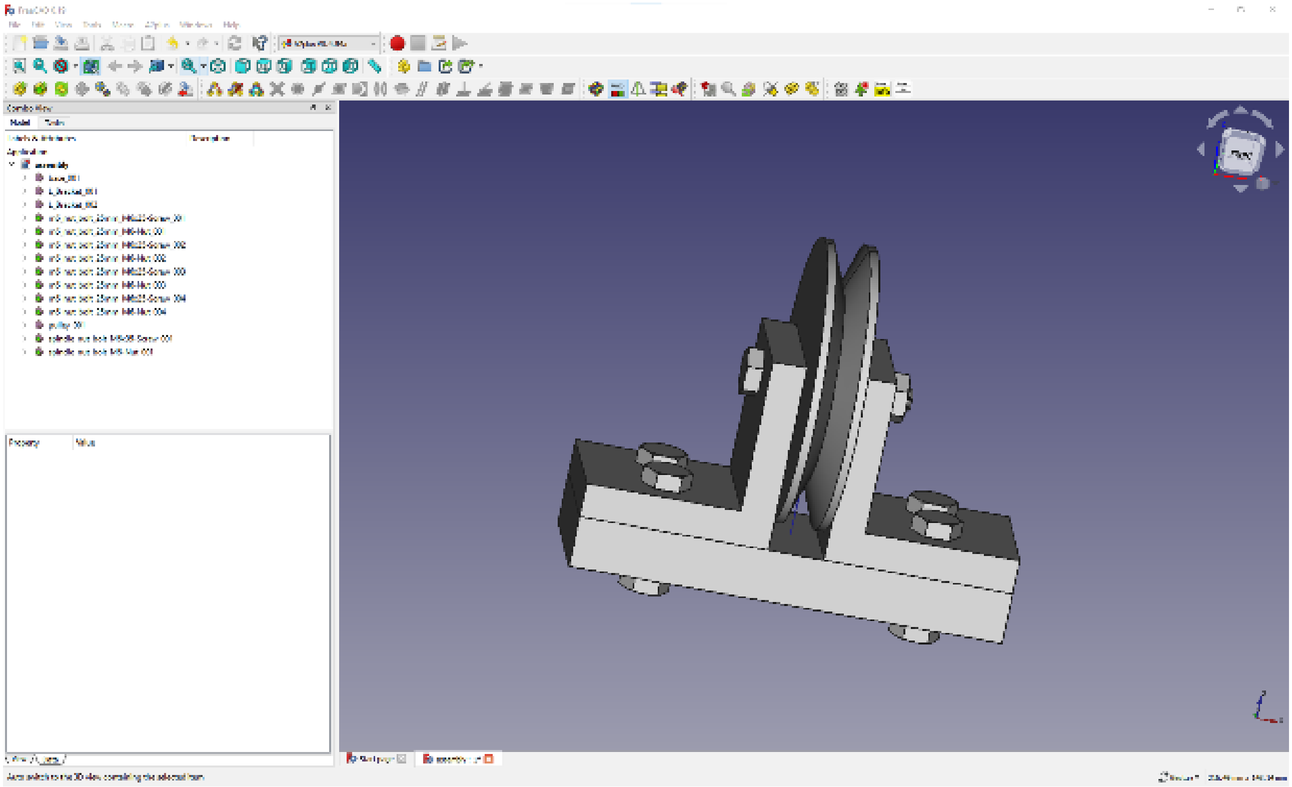


Having made your changes in Inkscape, you can resave the file as an SVG or, of course, Inkscape is capable of producing many other file types that may be of use to you depending on your making process. Remember, sometimes simple is best – you could always print the box part design out on paper as a template to cut out!

Assemblies

You will use two other add-on workbenches, which we'll need to install. Open FreeCAD, go to `Tools > Addon Manager`, and click to install the `A2plus` workbench. Once installed, follow the prompt to restart FreeCAD, and then repeat the above task to install the `Fasteners` workbench.

Creating an assembly in CAD is useful for many reasons: it allows us to design more complex projects, to check that parts fit, and it allows us to use the same parts multiple times. It also can be used to check clearances and movements of dynamic systems and more. You are going to make and assemble an example of a pulley wheel mounted in some brackets, sat on top of a base, and held together with M6 nuts and bolts. As you are focusing on the assembly, you aren't going to step through how you made every separate component. This isn't supposed to be a particularly well-designed pulley block. It would need some bushings and bearings in real life, as well as a proper spindle, but as an example, it's a good project to play with.



When using Part Design to create a body, that body is a single continuous solid object. The classic example is a nut and a bolt – which would require two bodies: one for the nut and one for the bolt – because although the items can be assembled, they are both single solid items. You could make two separate bodies within a project and then painstakingly move them into position relative to each other, but this is long-winded, and also, if you make a change or move the bolt, suddenly the nut is in the wrong place. Using A2plus, you will make assemblies where components are constrained to the specific parts of other components that they are attached to. Beyond parts simply being lined up positionally, the assembly constraints describe the relationship of one part to another. For example, if something can turn around an axis, or if one component is always parallel to another.

You begin by making your components using Part Design to create bodies for each part. You create a base, two L-brackets, some nuts and bolts, a simple V-pulley, and a longer nut and bolt to act as a shaft. For each part you create a single project and file.
You'll see that you are going to import these items into the A2plus workbench, and when you do, you could import each body either from an individual project, a project with just that one body in it, or you can import multiple bodies from the same project.
Most of your files are single items, but you created one file containing two parts to allow us to explore how to import an individual part from a multipart project.
This is useful for many reasons. You can imagine borrowing items from unrelated projects in your library.You made a base which is a simple block and is the file 'base'. The base has holes to mount the L-brackets using M6 nuts and bolts. The sketch for the base is drawn in the XY plane and is centred around the 0,0 point – the bolt-holes are constrained positionally relative to that origin point. This means that if you change the size of the base, the bolt-holes will remain in the correct position, which is important because they define the L-bracket position and the width of the gap for the pulley wheel.
The L-brackets are again a simple part with holes that match the hole coordinates on the base and a hole on the upright to receive the spindle bolt.

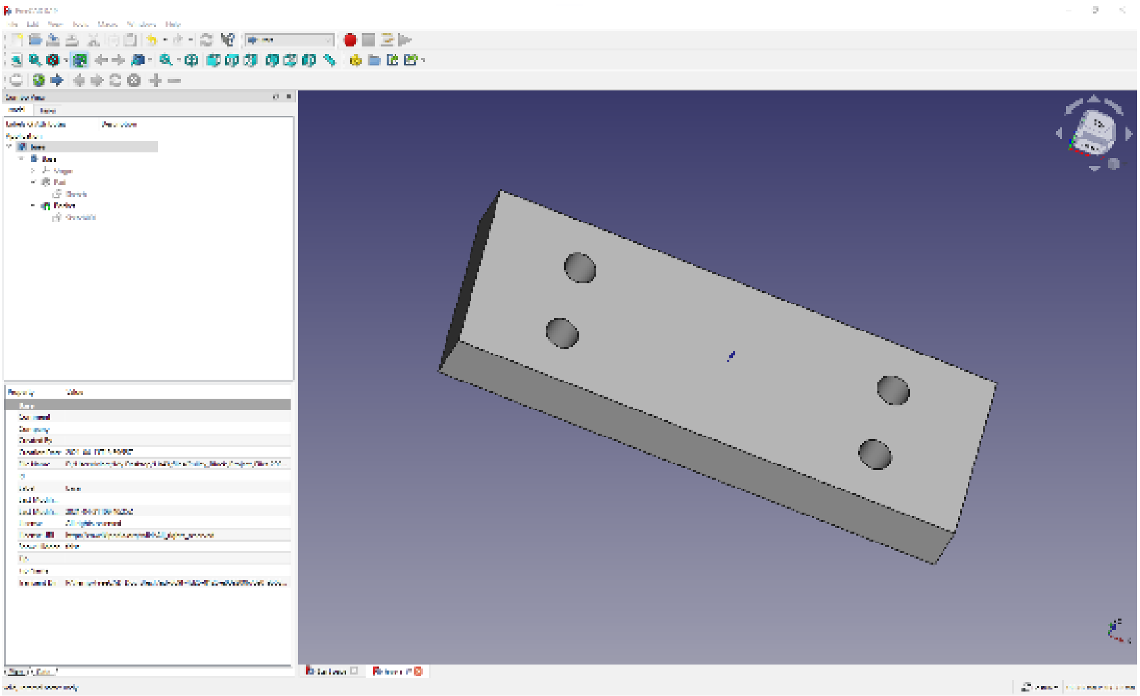
Join Together
You are going to make some nuts and bolts for your assembly. For this, you will use the `Fasteners` workbench you installed. Create a new project and move to the Fasteners workbench. The Fasteners workbench makes it easy to model metric and imperial nuts and bolts and screws – it even has some rarer fasteners like PEM insertion nuts, PCB standoffs, and more. You are going to use four M6 nuts and bolts that are 25 mm long to attach the L-brackets to the baseplate; however, you only need to model one and then import four copies to the A2plus assembly project.

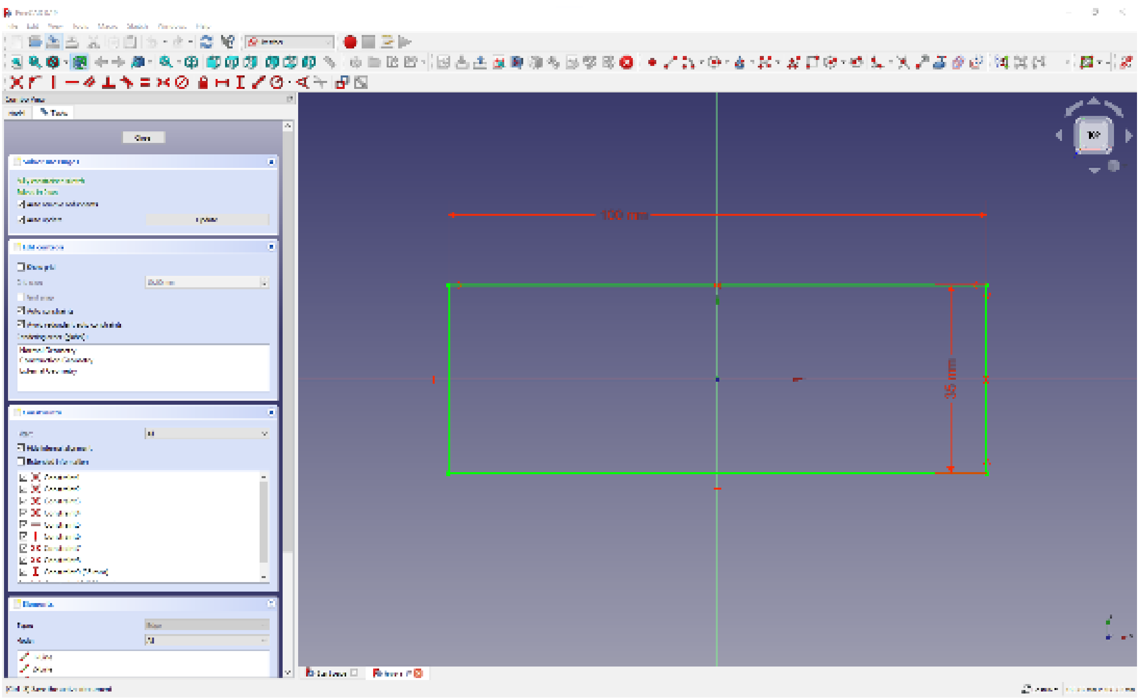
To create your bolt, first click the yellow icon that looks like a hexagonal-headed bolt, which when you hover over it says 'ISO 4014 Hex Head Bolt'. M6 is the default diameter for this bolt type on the Fasteners workbench – it should create a 30 mm M6 bolt model. You'll notice that the model doesn't have any thread modelled onto it. This is common across most CAD platforms when modelling threaded fasteners because the thread is a complex geometry. If you have lots of fasteners, it can increase the project complexity and speed of loading or recomputing.
Most of the time in CAD, you don't need to model the thread. However, you can simply add the thread to this model by highlighting the item in the file tree and, in the dialog box, setting the 'thread' property to 'true'. Depending on your computer, this may take some time to generate the thread on the model. Turn the threads back to 'false' and shorten your bolt to 25 mm by selecting 'custom' in the 'length' property in the dialog box and typing in 25 mm.

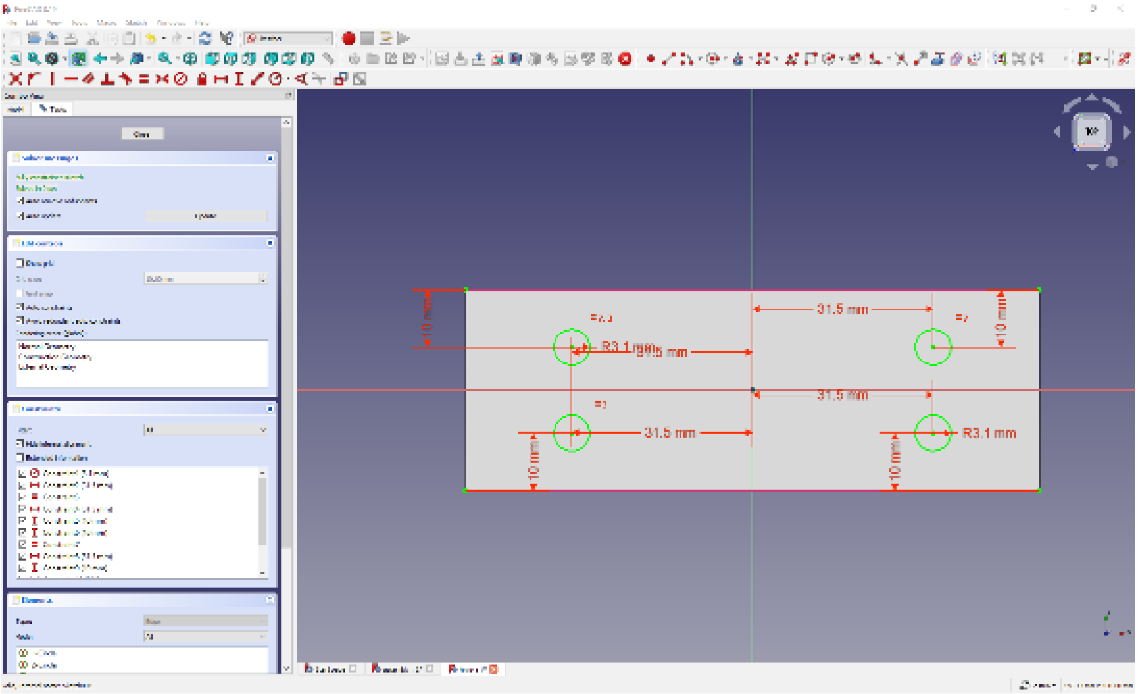
Let's create a nut for your bolt in the same project file. Depending on your screen, you might need to expand the toolbar to select the yellow icon that looks like a hexagonal nut labelled 'ISO 4032 Hexagon nuts, Style 1'. Again, the Fasteners workbench will automatically create a nut, but it will be positioned on the origin point in the same place as the bolt-head. To move this, double-click the nut in the file tree and then use the arrows in the preview window to move it down and away from the bolt. Making sure you have turned off the thread setting for each item, save this project ready for use in your assembly.

Some measurements:

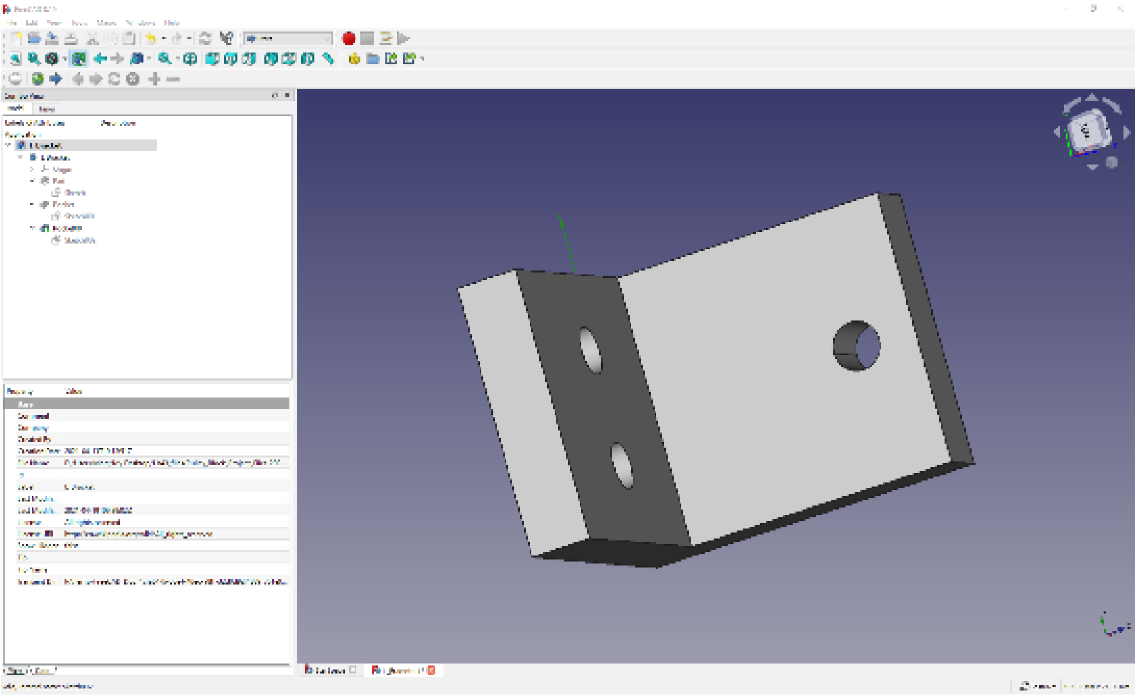
- Base is 100 x 35 x 12 mm with holes of 6.2 mm located at 10 mm from long side and 31.5 mm from the middle.

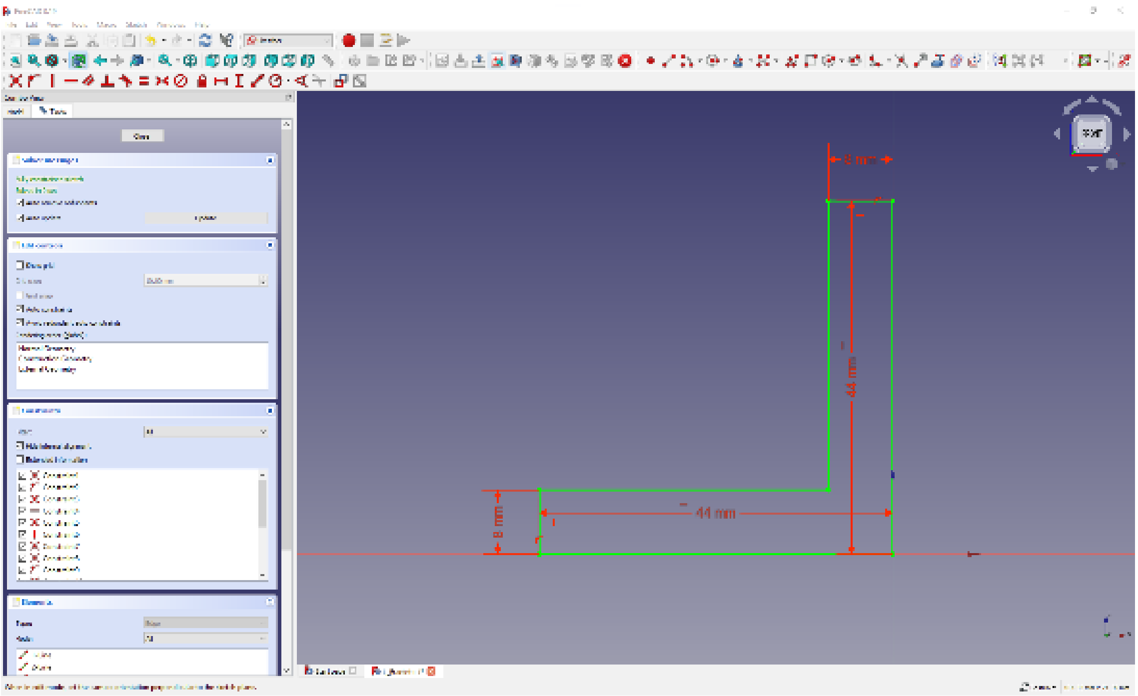


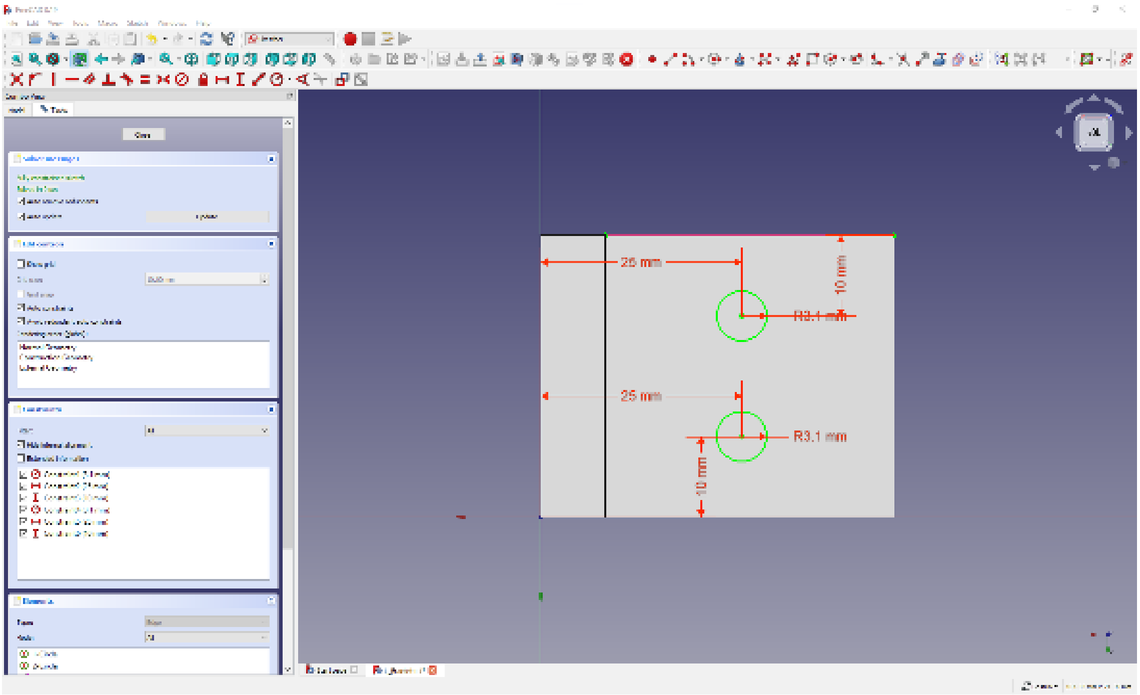



31.5 mm
R3.1 mm
10 mm
R3.1 mm

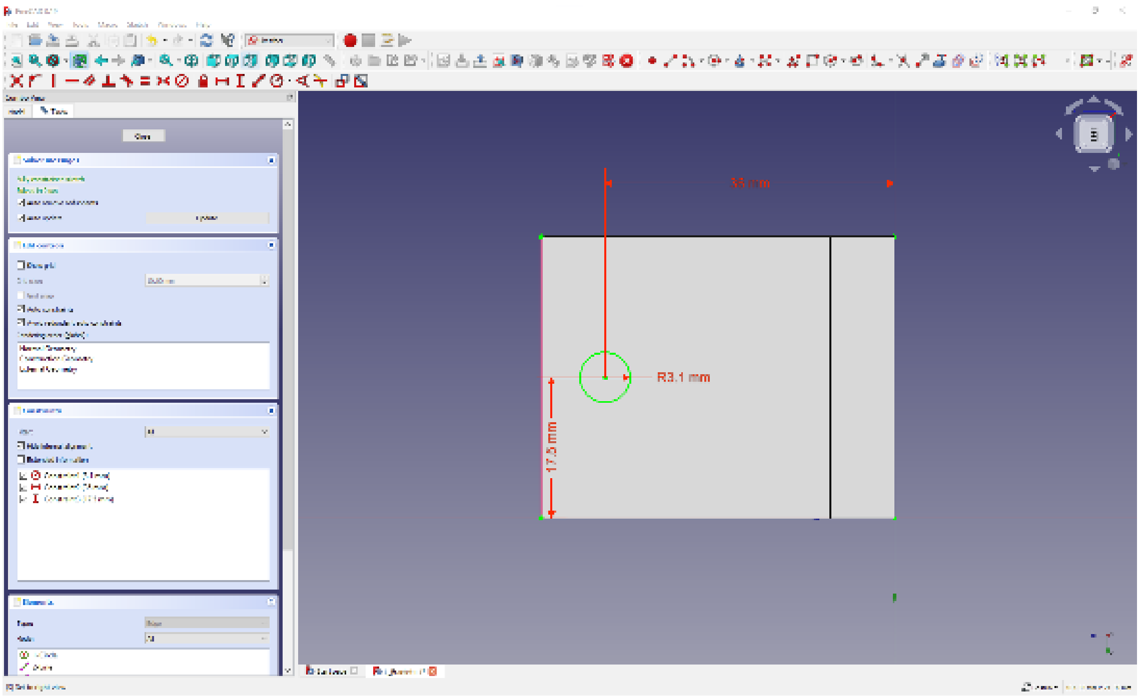
- L-bracket is 44 mm x 44 mm x 35 mm x 8 mm with holes of 6.2 mm located 10 mm from long side and 25 mm from short side while the pulley-hole is located in the middle at 8 mm from top





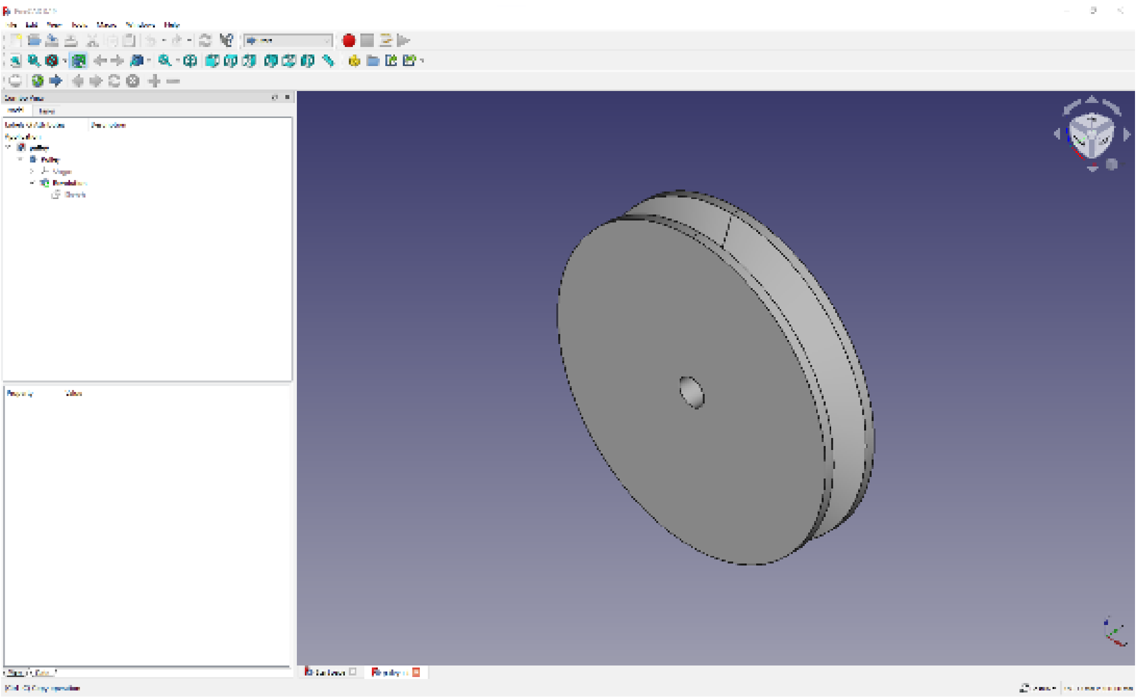


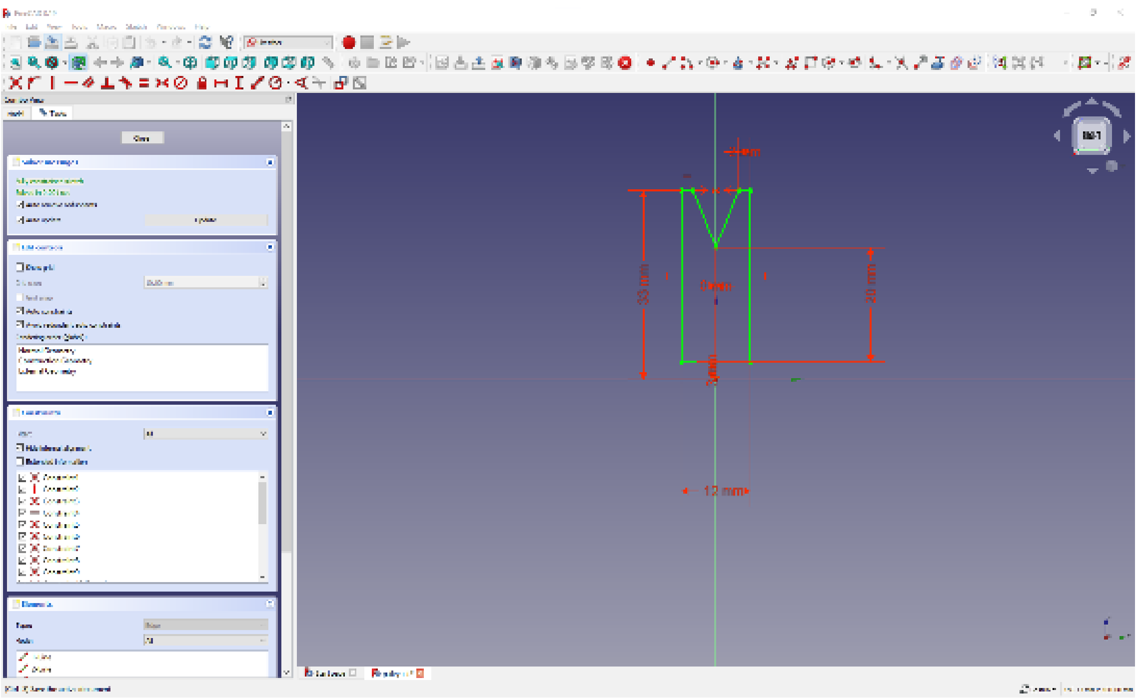
25 mm
10 mm
R3.1 mm
25 mm
R3.1 mm
10 mm



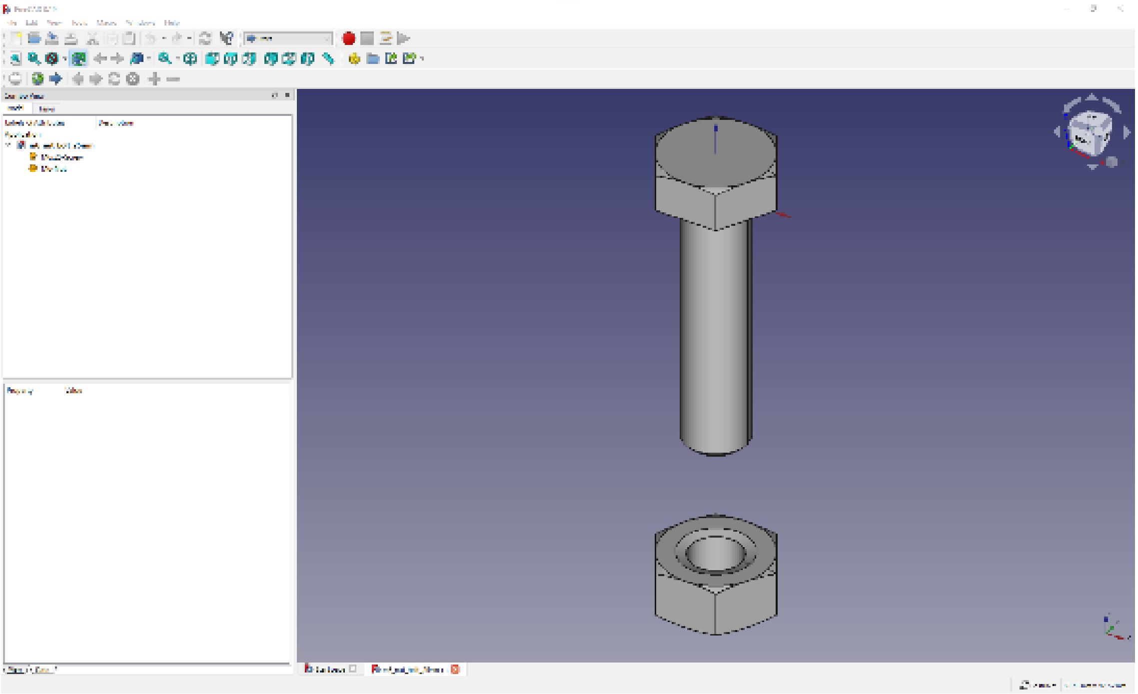
R3.1 mm
17.5 mm

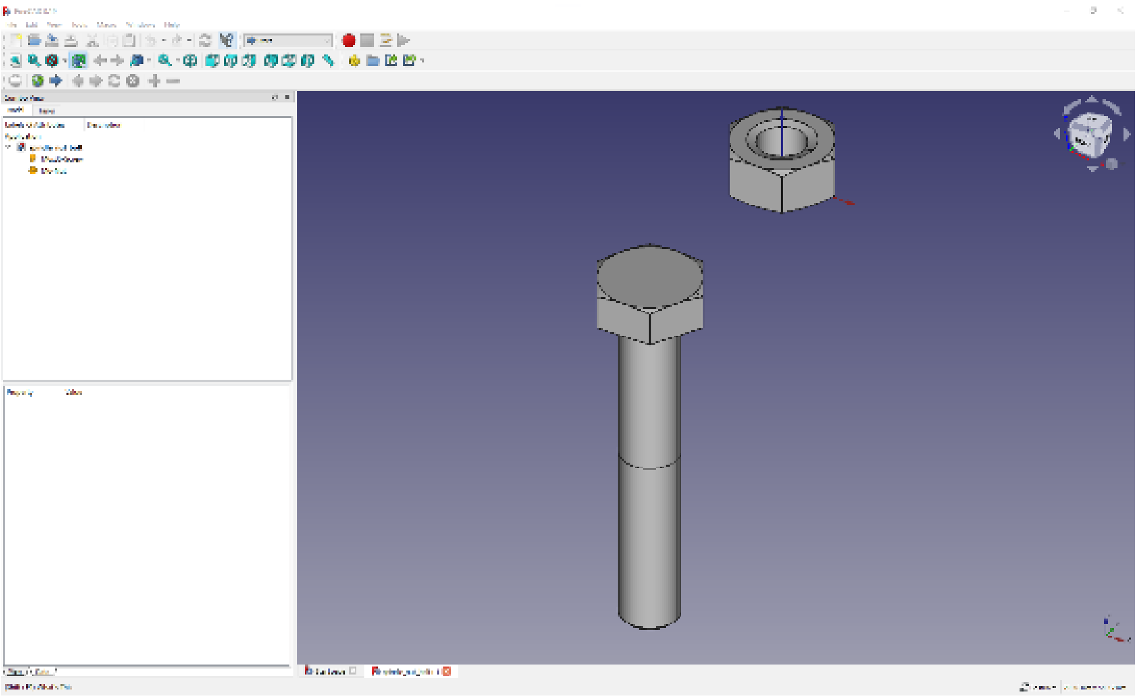
- pulley has a diameter of 66 mm with a centerhole of 6.0 mm





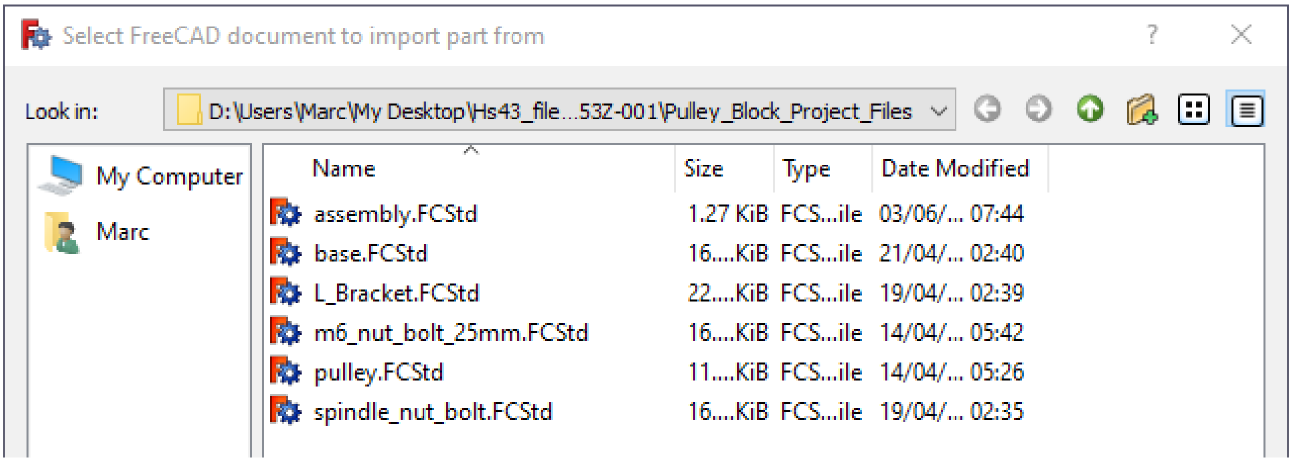
- bolts and nuts are M6 with bolt full length of 45 mm and 30 mm or thread-lenght of 40 mm and 25 mm





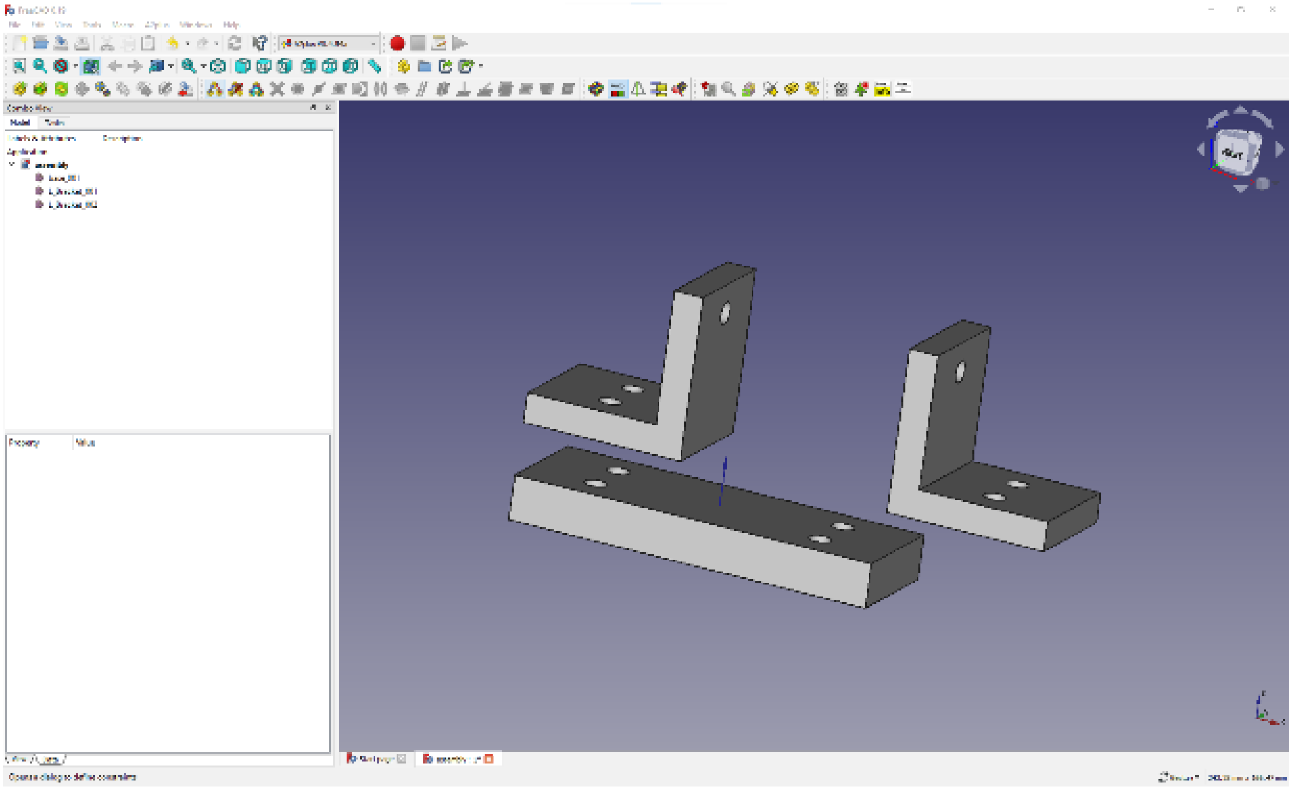
Now that you have all the parts, create a new project for the assembly in FreeCAD. For ease, save this new project in the same folder as all the component files.
Move to the A2plus workbench and start importing components. On the left-hand side of the A2plus toolbar, you should see a yellow and green tool icon, which when you hover over reads ’Add a part from an external file to the assembly.

Click this and navigate to select the ’base’ file in the folder. Select the file and click Open.

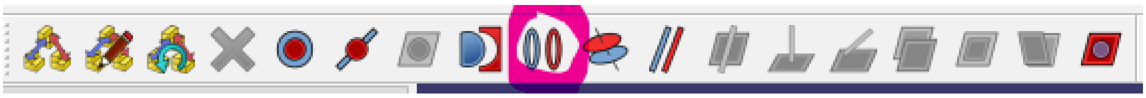


In the preview window, you should now have a copy of the base object inserted.

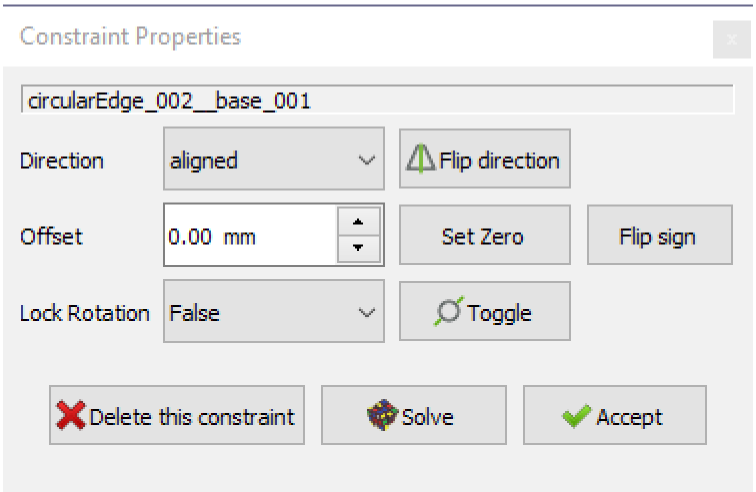
Let's repeat the task with the L-bracket file. The L-bracket will import overlapped with the base – you might notice that until you left-click, you can move it to any position in the preview area. If you need to move it again, double-click on the item in the file tree menu and you can transform it incrementally using the dialog and the axis arrows. Move it to a position free of the base to make things easier to see. Repeat the process and import a second instance of the L-bracket file. The files in the file tree will automatically be given a number suffix, so you have L_Bracket_001 and L_Bracket_002. As an example of one way of working, simply rotate the second bracket so that it faces the correct way relative to the base in terms of how it will sit when assembled.

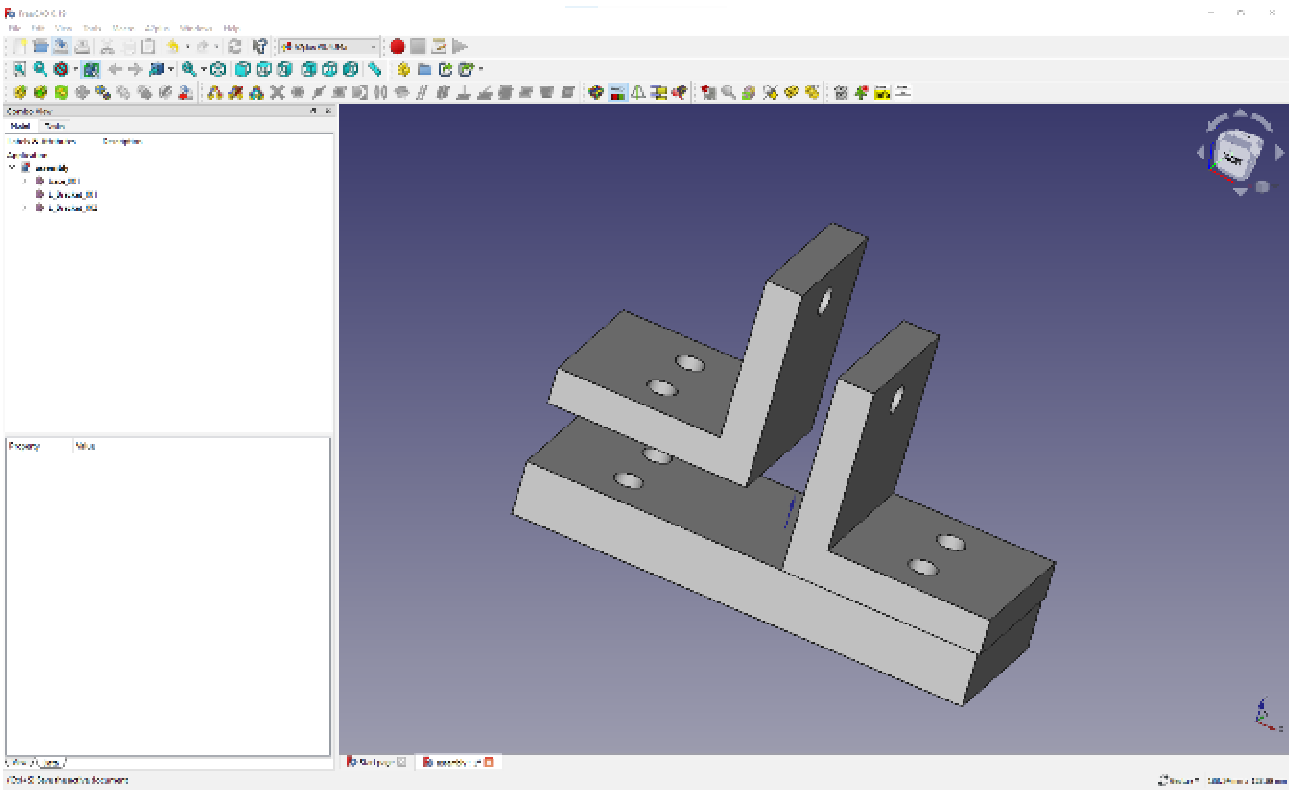


Let’s assemble the L-brackets to the base – to do this, select the bottom edge of one of the holes on the first L-bracket. This edge has a corresponding edge on the upper end of a hole in the base, and when in position, these two edges would sit together (Figure 4). Move the model so that you can see that particular hole edge, and then using the CTRL key, click to select it. You should now have two hole edges selected. When you select items in A2plus like this, any constraint types that could be applied to the items automatically become available on the toolbar.
You should now be able to click the ’Add circular edge constraint’ tool icon.



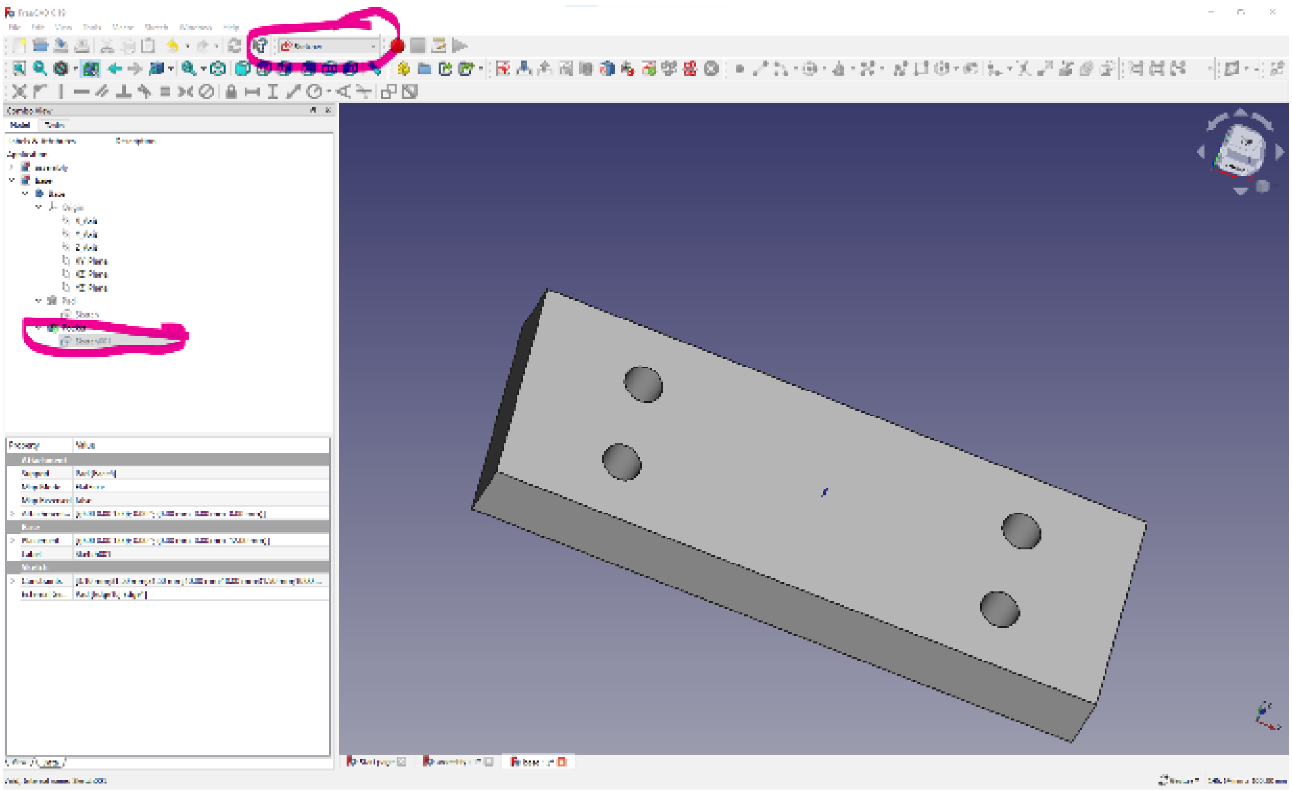
You will see that the L-bracket moves, hopefully into a correct position.
You will also see a dialog box with some options showing the constraints you can adjust. You shouldn’t need to adjust the constraint for this one, but it’s worth noting the ’Flip’ button, which is commonly used if the constrained items end up in an undesired position – often clicking the Flip button repeatedly will toggle through the optional placements until you find the one you require. Click Accept to confirm the constraint.





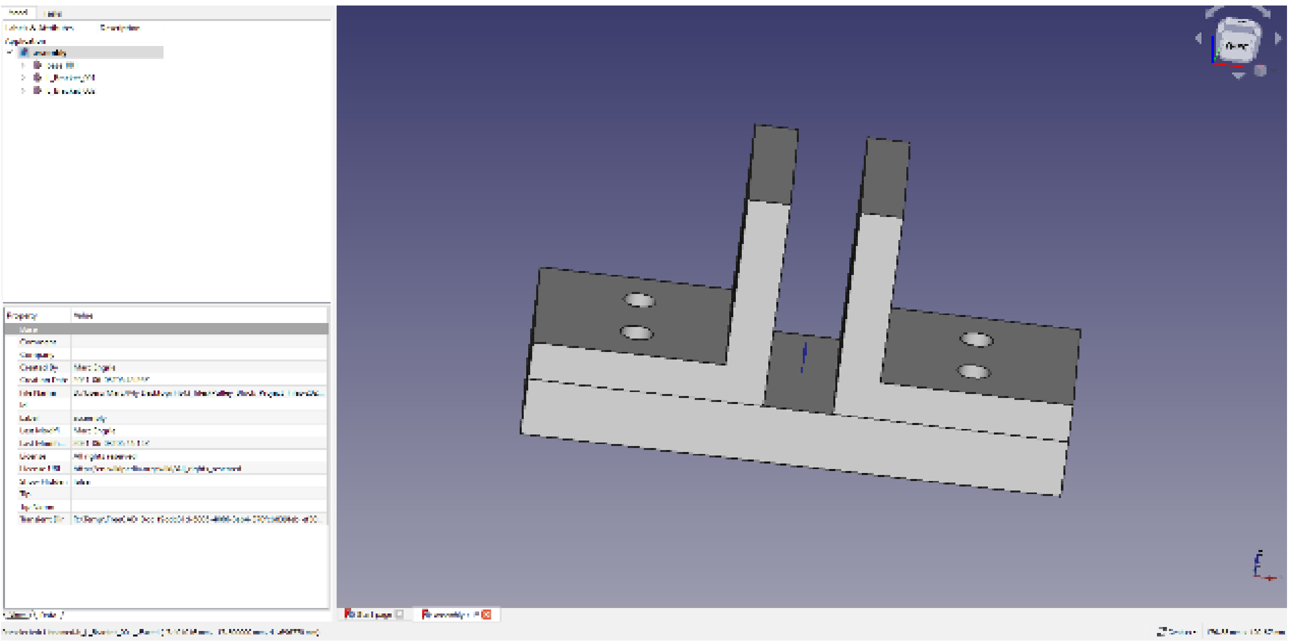
Double Up

Repeat the process for the second L-bracket to end up with the two L-brackets facing each other with the gap in between for the pulley wheel. Noting the error on the size of the base, let's use that to work through how to edit a part from the A2plus workbench.
Highlight the base in the file tree and click the 'Edit an imported part' tool icon (Figure 6). Clicking this should open the base part in a separate project, and the file tree of the part will appear. Even though you are still on the A2plus workbench, you can click the drop-down menu to select the primary rectangular sketch, which is the root of the base object. Double-click this to open the sketch in the Sketcher workbench, and then change the horizontal dimensional constraint to 101 mm.



Close the sketch and then save the base part project by clicking the Save button. Now close that project tab. Notice that the base in your partly assembled project hasn't changed. To update the parts, click the 'Update parts, which have been imported to the assembly' tool icon (a yellow part icon with two curved green arrows over it).

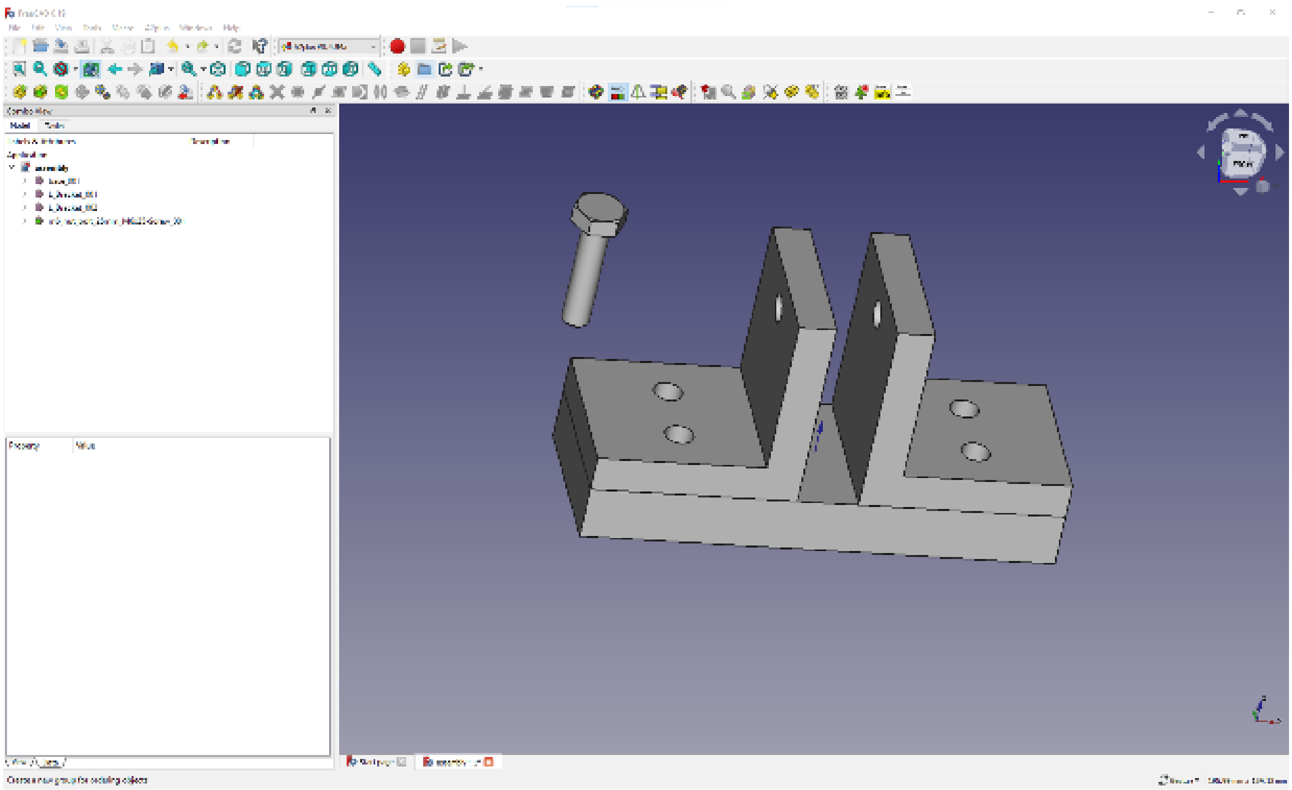
Once clicked, you should see the base now fits, perfectly matching the L-bracket ends.



A point to consider again is that the holes in the base file are constrained to the centre line of the base. This means as the base dimensions are expanded, the holes stay the same distance apart. If you had constrained the holes relative to the edges of the base, then your update to the horizontal length would have moved the holes, and the brackets would still overhang. As you develop parts destined for assemblies, these factors become more important to consider.

Using the project with the M6 nuts and bolts you just made, let's add the four nuts and bolts to connect the brackets to the base. Just as an experiment, try and import the nuts and bolts project using the same technique you did for the base and the brackets. You'll note that even though there are two objects, it will import, but the nut and the bolt appear as a single item in the file tree and cannot be moved independently. The correct way to do this is to use the 'Add a single shape out of an external file' tool icon.

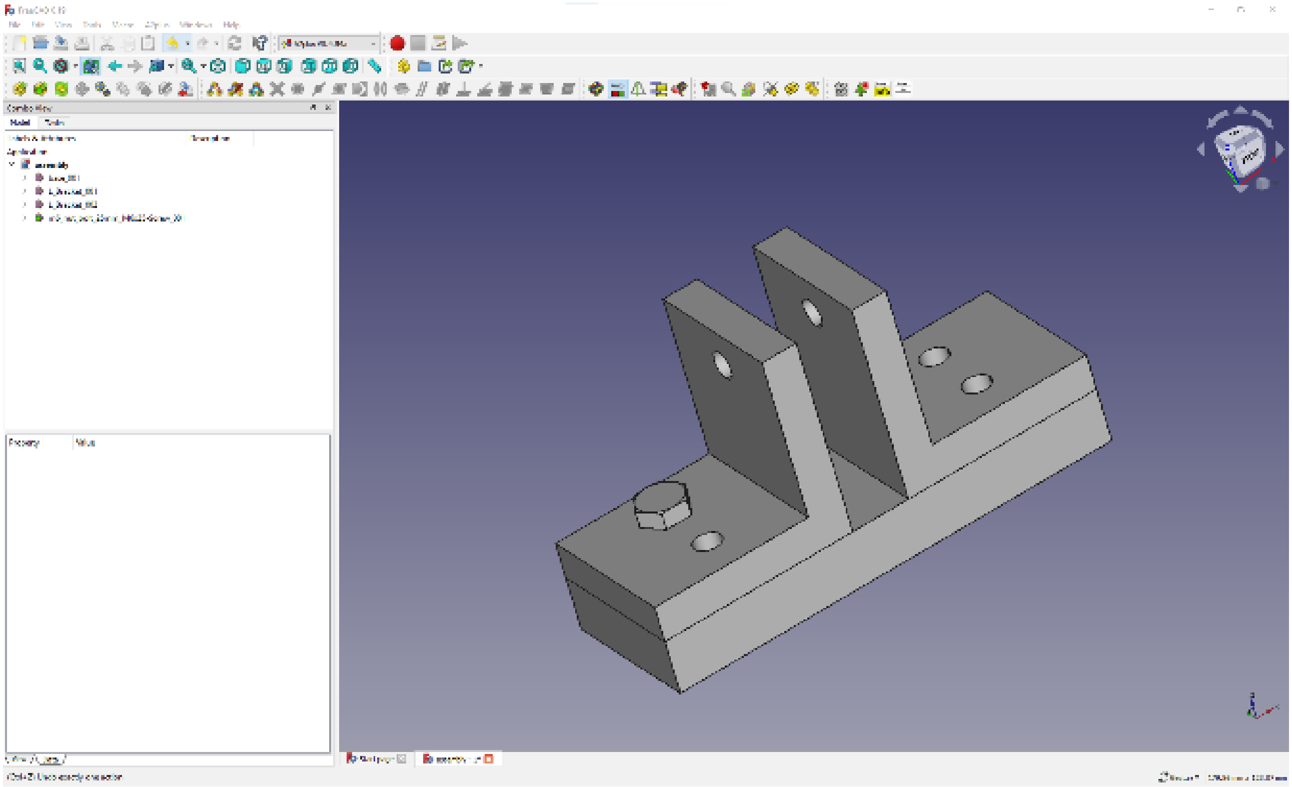
Having deleted your incorrectly imported nut and bolt, click this and then select the file again; this time, a dialog asks us to specify which part to import from within the file. Select the nut or bolt and repeat this for the other part. Once you have the first nut and bolt in place, let's constrain this one before tackling the others.



Let's constrain the bolt section first. Select one of the circle edges at the opposite end of the bolt from the head. Using the CTRL key, select one of the upper edges of one of the holes on the bracket. You want these two circles to align on an axis, but you don't want to constrain the edges together as the bolt end circle obviously needs to be out of the other end of the hole. Clicking the 'Add an axis constraint between two parts' achieves this – the bolt should now align to the hole.

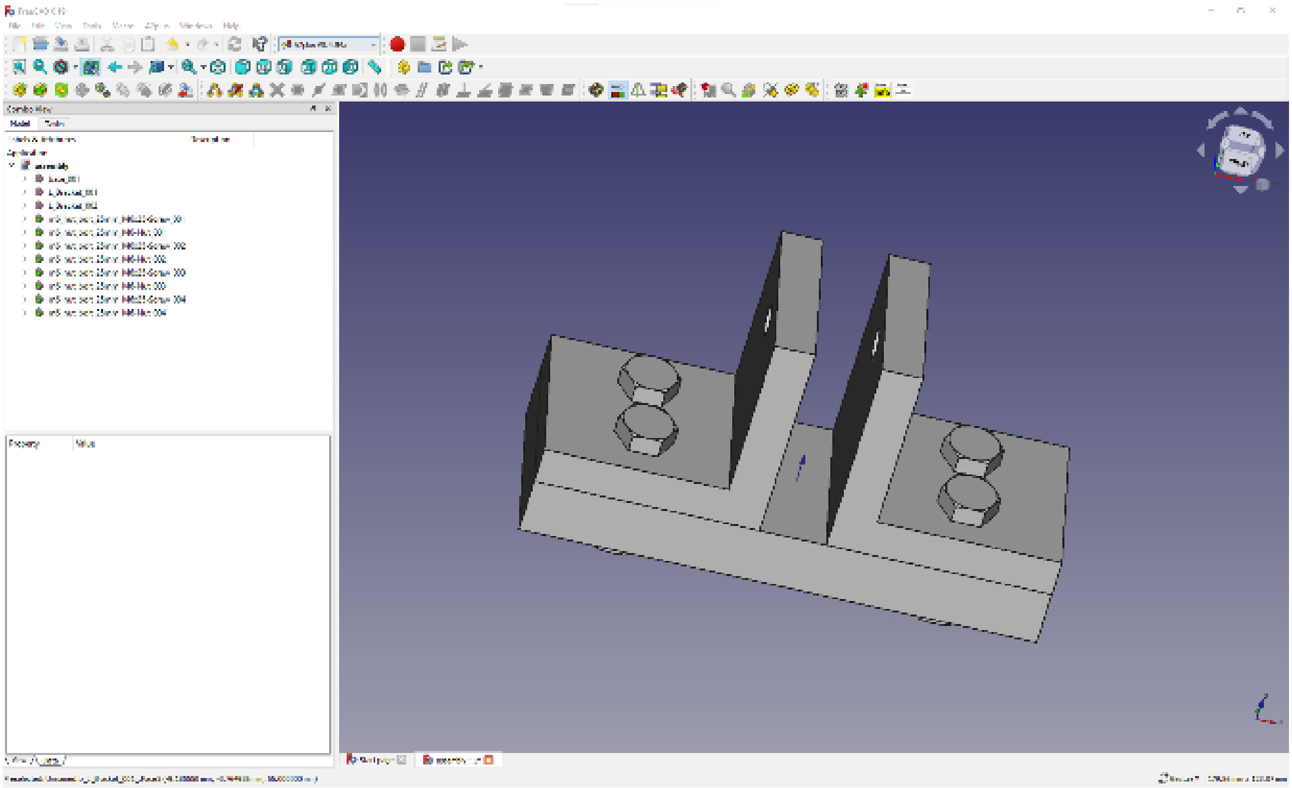
To make the bolt-head sit flush to the surface of the bracket and the bolt to sit all the way through the hole, let's add another constraint. This time, click to select the upper face of the bracket and then select the lower face of the bolt-head. These two faces can then be brought together by clicking the 'Add a plane constraint between two objects' tool.

Now the bolt should be correctly in position. Click Accept in the dialog box to finish the constraint.

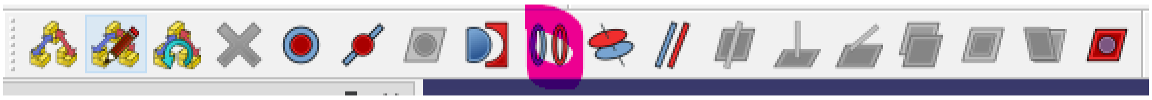


Moving to the nut, if you move the nut closer to the bolt end under the base, you can then select the edge at the base of the bolt before the slight chamfer and the lower similar edge just inside the nut. Again, apply a 'circularEdge' constraint to fit the nut to the bolt.

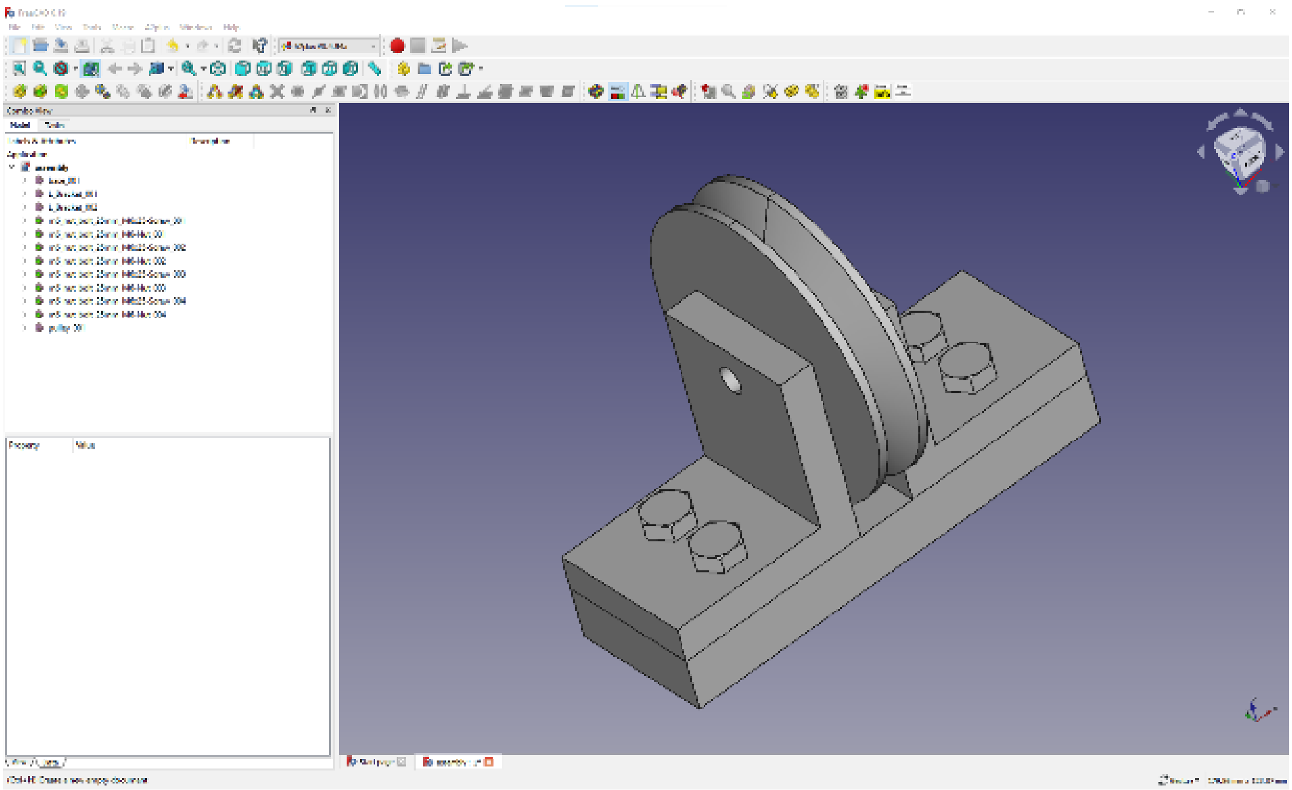
Now that your nut and bolt are constrained, you want to add the other nuts and bolts. Again, as an example for learning, you might think – based on other workbenches – that you can copy and paste the nut and bolt in the file tree system.
Of course you can, but now that you have added constraints to your first nut and bolt, if you copy and paste, that will copy all the constraints applied to that part.
Of course, you could delete these constraints and reapply them in the other positions, but it's easier to either import multiple copies or to copy and paste before adding constraints to an object.
However you choose, create three more nuts and bolts and constrain them into position in the assembly.



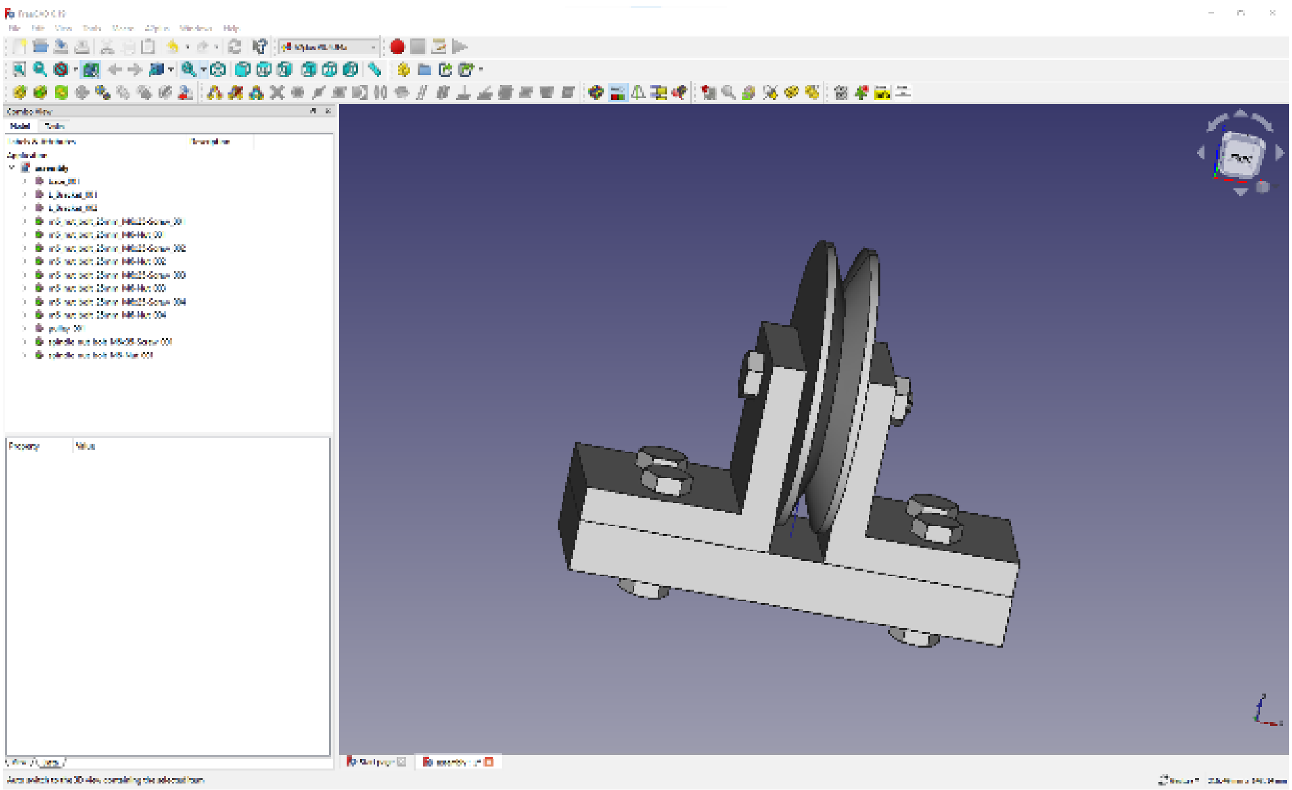
Pulling Together
Let’s add the pulley wheel. Import the pulley wheel, but let’s use constraints to position it rather than double-clicking to use the transform tools. Select one of the edges of the hole through the pulley and then click one of the holes on the inside of the L-brackets in the gap for the pulley wheel. Even if the pulley wheel is at 90 degrees to the brackets, clicking the ’Add a circularEdge constraint’ should bring it into the correct orientation.



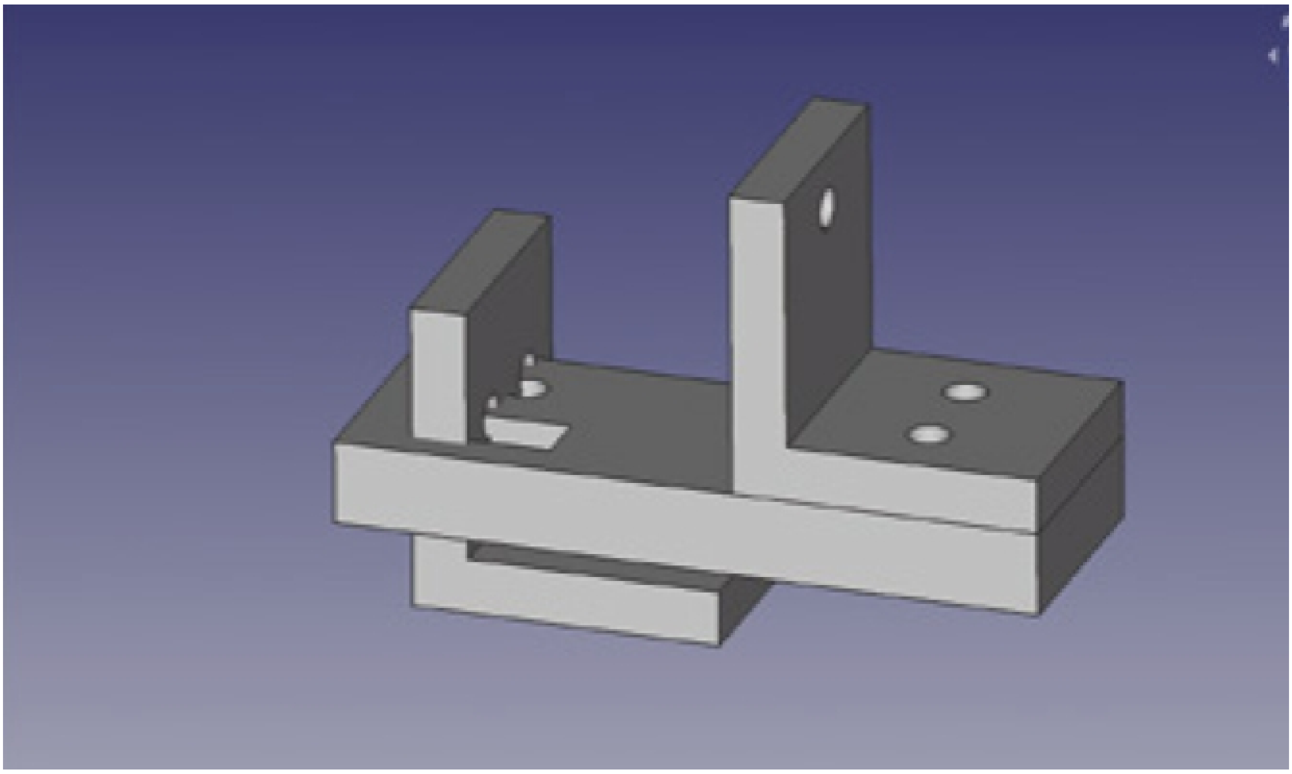
You may need to flip the direction in the constraint dialog to get the pulley largely between the brackets. Before you close the constraint dialog, note that the pulley is 12 mm wide and the gap between the brackets is 13 mm – this is because one would probably want the pulley wheel to have some clearance. Currently, the pulley is positioned touching the bracket of which you selected the hole edge. Setting the ’offset’ to 0.5 mm in the constraint dialog moves the pulley 0.5 mm off the bracket and positions it equally with clearance between the two brackets. Finally, there is a ’Lock Rotation’ value in the constraint dialog. Setting this as ’false’ means that your pulley wheel part can be rotated around this circular constraint, emulating its real-world movement. If you wanted to lock it at a certain point in rotation, set the part in position and then toggle this value to ’true’.



To finish your assembly, import the spindle bolt file and constrain it through the centre of the brackets and the pulley, then fit the nut to the other end. You are sure by this point that you can work out how to constrain this one into position without instruction.



Keeping Backups
If you hang out in CAD circles for long enough, regardless of what CAD environment you use, you may come across a term for a problem. The ’topological naming problem’, or ’toponaming’ for short, is a common issue you can easily demonstrate in your project. Objects and parts – for example, the base of your pulley block – are all made out of edges and faces.
These edges and faces are automatically given an annotation. Sometimes you see these names when you select things like ’edge 12’ or ’face 3’. When you apply constraints in the A2plus workbench, you are essentially telling FreeCAD to ’attach face X to face Y’ or ’make edge 2 parallel to edge 5’, etc. When you edited your base part to make the brackets fit, you only changed a dimension, and, as such, none of the names of the features changed. As an experiment, edit the base and create a small rectangular pocket over one of the holes – to any depth – and then save the file.
As you did before, update the parts in the assembly. When you do so, you’ll see that the model is now a little broken, as in the image below. This is because the names of the edges and faces have changed, but the constraints don’t know this. It’s definitely worth having a backup of a saved assembly before making any major edits to parts within it. You might find that sometimes these issues are easily resolved by removing constraints and reapplying them.



Quick Tip
You used the Part Design workbench and Sketcher workbench tools to create the different parts for your assembly. You covered these approaches in the first two parts of this series.

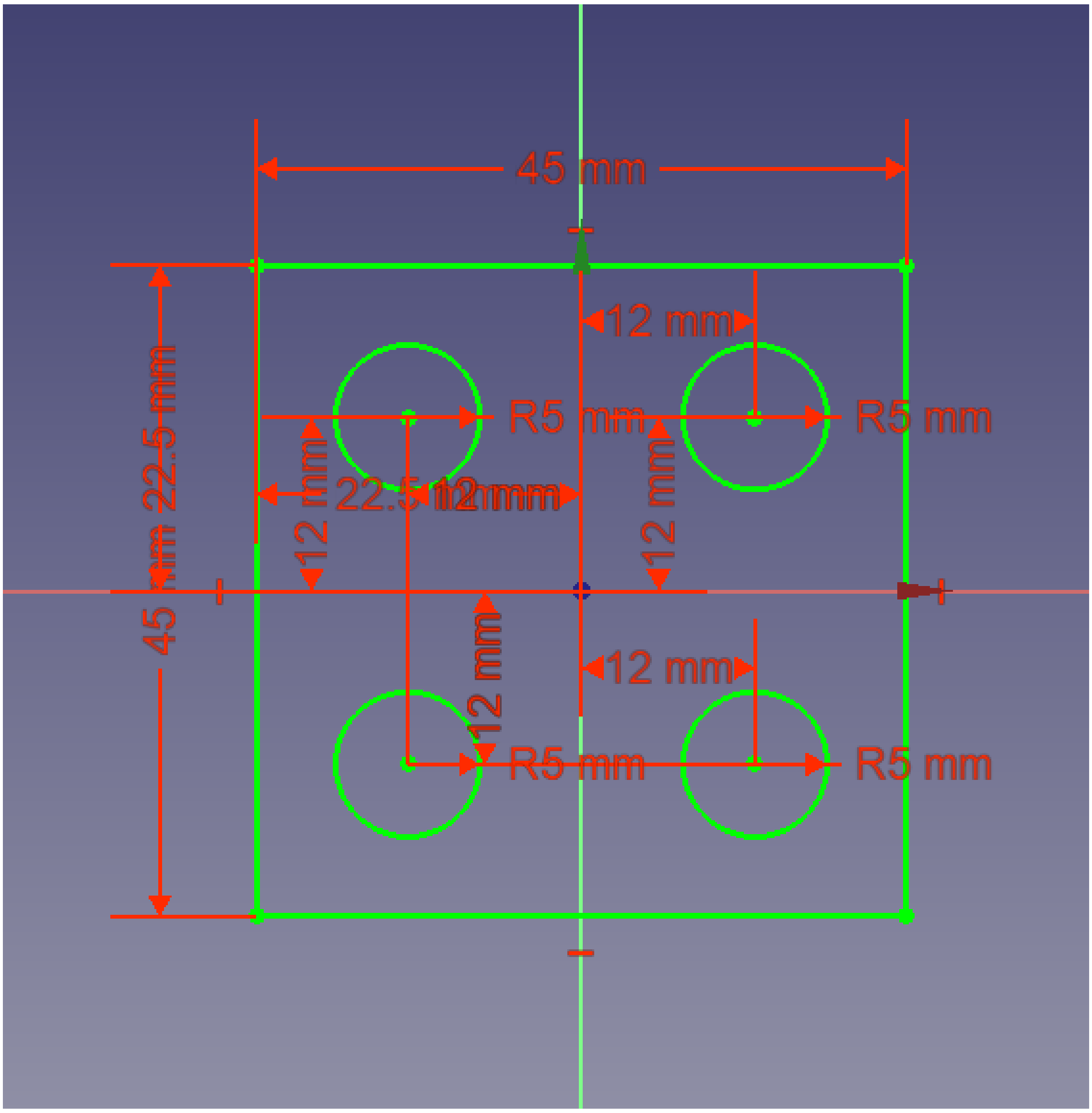
At the risk of sounding obvious, you can use a component part numerous times in an assembly, so don’t model things more than you need to!

Similar to Sketcher and other parts of FreeCAD, there are numerous ways to achieve similar results using different assembly constraints.

Making multiples of things

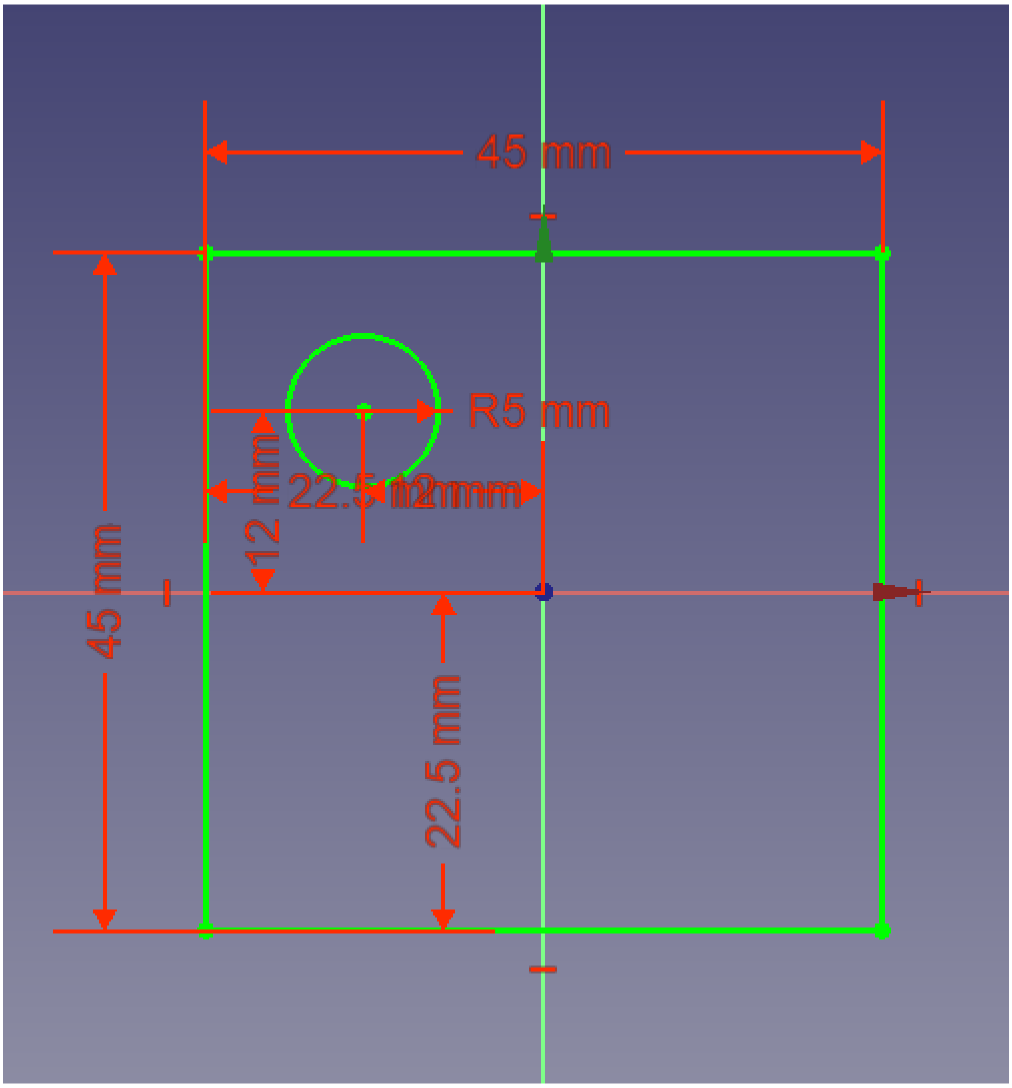
Often when you are designing something, you would like to make multiples of some aspect of the design and you want to do it as quickly and as efficiently as possible. A common example might be placing circles to become pockets, holes, or pads in a sketch. As a first example, let’s go straight to the Sketcher Workbench and create a new sketch in any plane.

You can see the desired sketch item, a small square with a hole in each corner.



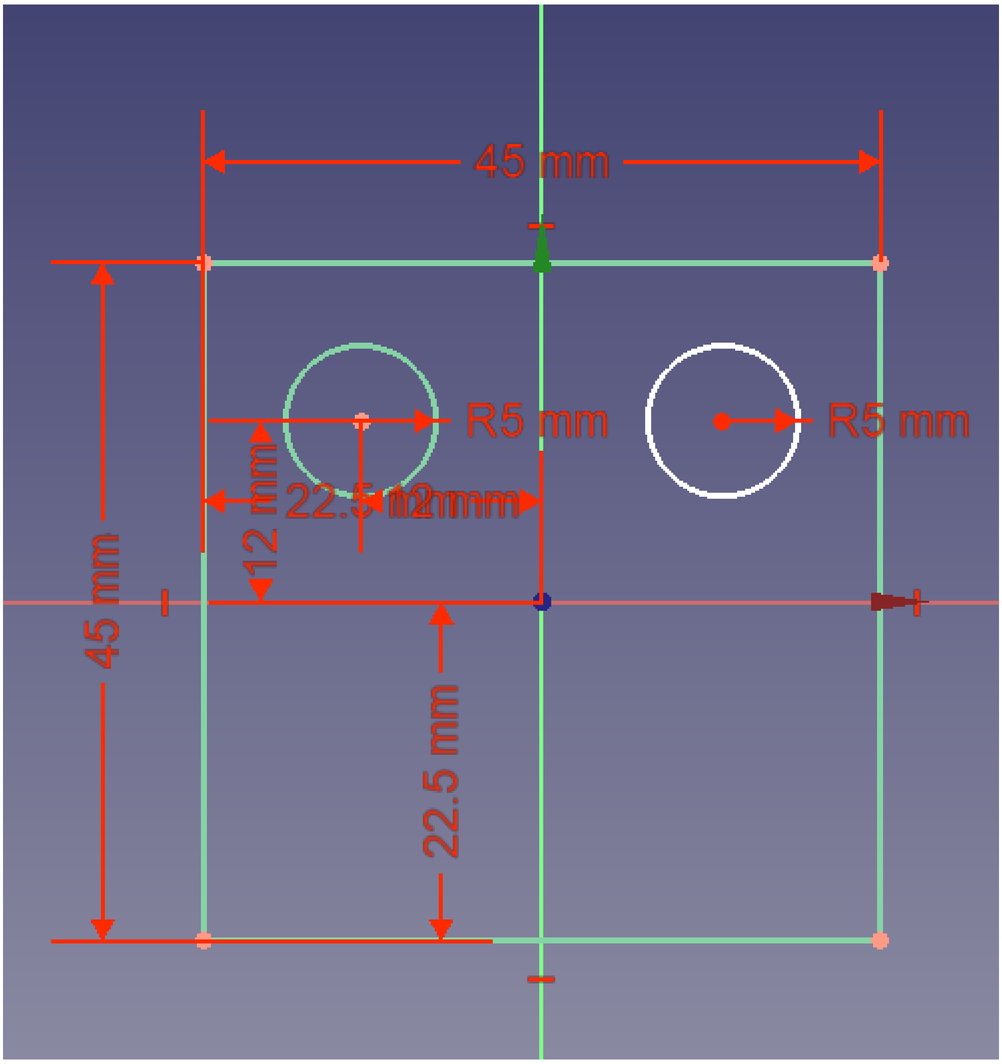
To create this sketch, you drew four separate circles and constrained each of them dimensionally and positionally. It’s a fine approach, but it’s quicker to use some of the cloning and array tools the Sketcher Workbench offers. In Sketcher these tools can create matching objects and carry over some of, or manipulate, the existing constraints to save us some time.

Starting over, if you draw one circle for example in the upper left-hand corner of the square, you can constrain the radius and then the position by setting a vertical and horizontal distance constraint between the centre point of the circle and the origin point of the plane.



Next, select the circle and the Y axis datum line. You can now click the 'Create symmetric geometry with respect to the last selected line or point' tool icon.

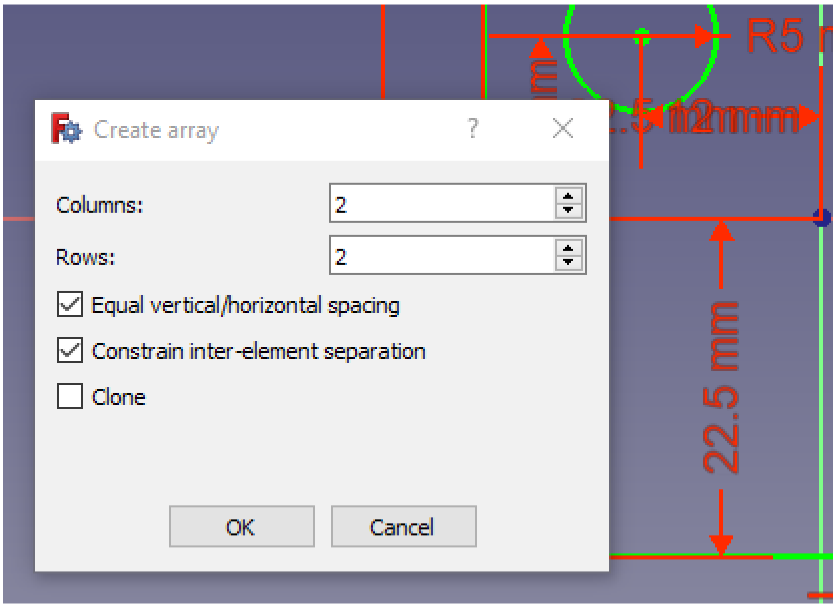
This should now create a second circle, with the same radius constrained and in a mirrored position in the upper right-hand corner of your square. It doesn't have the positional constraints, but it is in the same mirrored position as the original, so if you add a vertical and horizontal constraint without changing the default position then they will remain correct.



Of course, you can then select the two circles and the X axis datum line and repeat the process to create two more circles in the lower left and right corners.

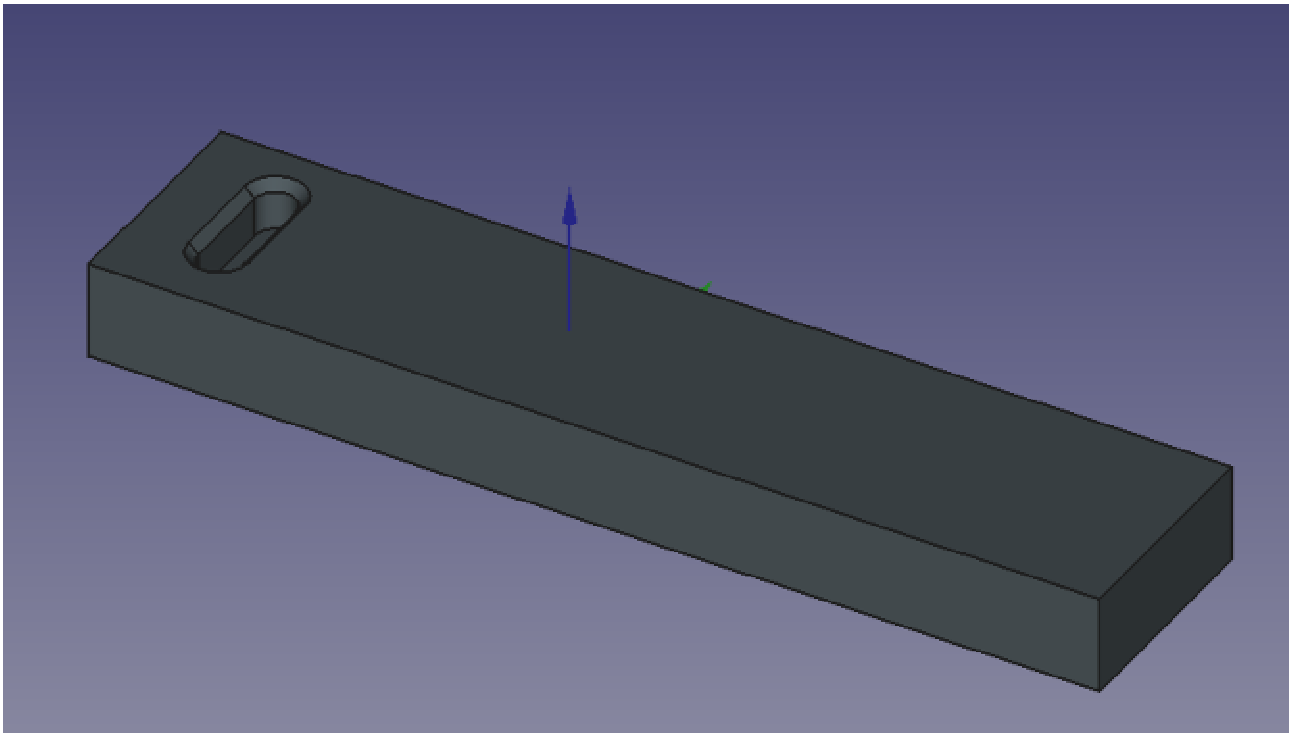
Another approach is to use an array tool called 'Create a rectangular array pattern'.

Starting again with just your upper left constrained circle highlighted, click this tool and you'll see a dialog box.



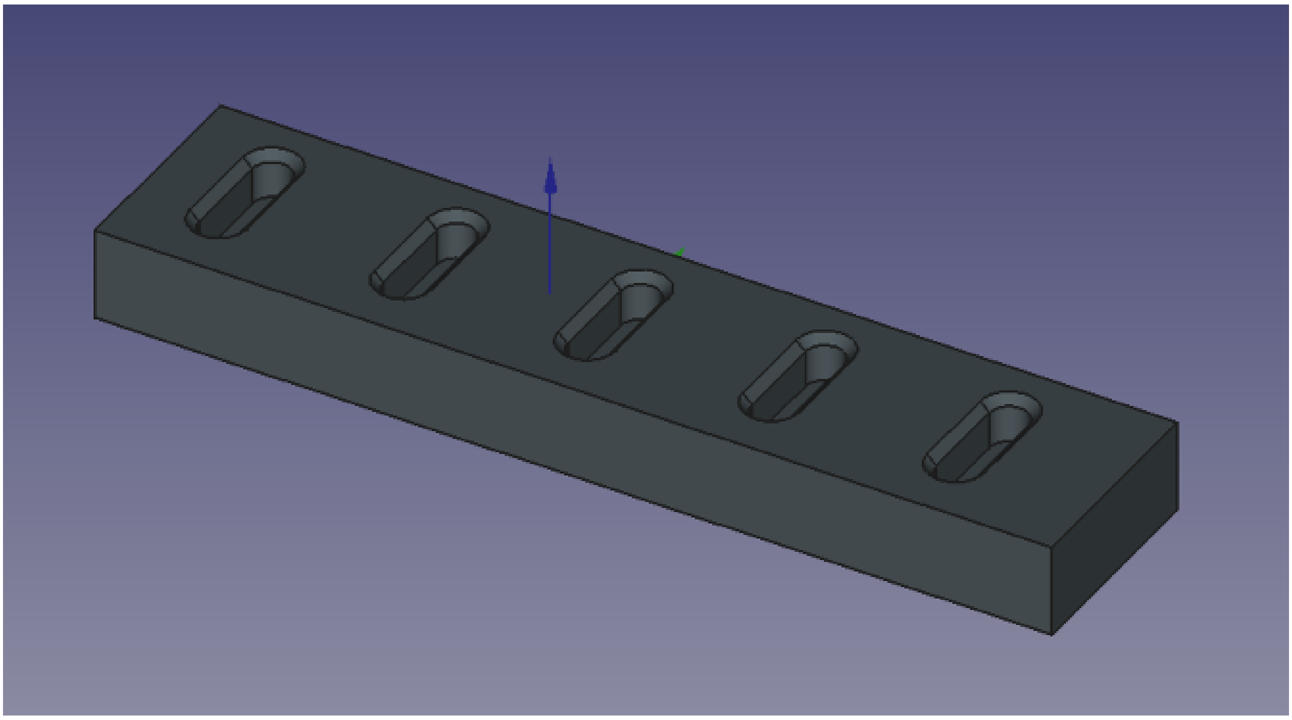
In the dialog, set the column and row number each to '2' and then tick the 'Equal vertical/horizontal spacing' checkbox as well as the 'Constrain inter-element separation' box. Clicking OK, if you now move your mouse around the sketch area, you will see a line protruding from your original circle; this represents the angle at which your array will be created and the distance from the original circle. Left-click to place the rectangular array of circles into the sketch.
Don't worry if you set it at a slight angle: click on the generated angle constraint and set it to zero degrees. Finally, edit one of the distance constraints it has added; as your original circle is 12 mm from the datum point vertically and horizontally, you set the constraint distance to 24 mm, which makes your square array of circles fully constrained and accurately placed.

Design Parts With Part Design
For your next examples, let's move to the Part Design Workbench and create a body and then create a sketch in the XY plane. In the sketch, draw a simple rectangle and don't worry about constraining it. Close the sketch and pad the sketch to any thickness. Next, select the upper face of the padded rectangle in the preview window and click to create a new sketch on that surface. Select the 'Create a slot in the sketch' tool and draw a small slot at one end of the rectangular surface. Close the sketch and then perform a pocket to cut the slot into or through the pad. Just to add to the example, select an edge of the slot and apply a fillet, which should auto-complete around the entire pocket edge.



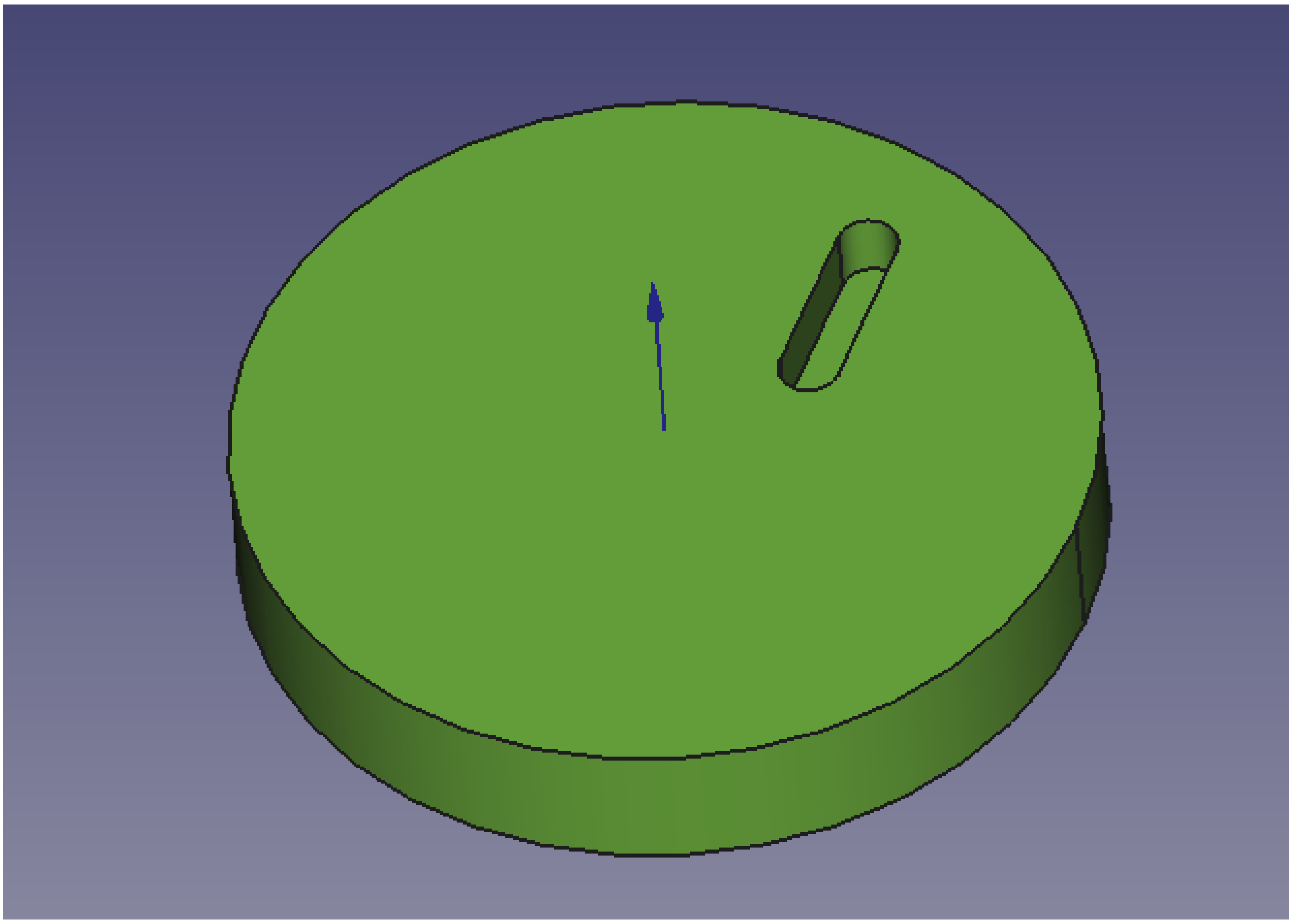
The Part Design Workbench has some specific array tools, and you are going to try the 'Create a linear pattern feature'. With nothing selected in the file tree, click the linear pattern tool.

You should now have a dialog opened in the Tasks tab of the Combo View window. At the top of this dialog, you should see a panel titled 'Select feature' and all the available features should be listed; in this case, that is the pad, the pocket, and the fillet. For now, highlight just the pocket item in the list and click OK. You may see a duplicate pocket in the other end of your rectangle now, but don't worry if you don't. You should see that the dialog box in the Tasks tab has now changed; scroll down to the bottom of this new panel. You should see two adjustable parameters: 'Length' and 'Occurrences'. The value of Length represents the length of the array and isn't linked to the length of any attached object. At the default length of 100 mm with two occurrences, this might place the second occurrence of the pocket outside of the rectangular pad. Increase the number of occurrences to 5 and adjust the length of the array so that your array of pockets fits in the rectangle. You might also discover that if the length is shorter than the multiple widths of the pocket, it will overlap the pockets in the array.
Before you close this example, you'll note that your fillet hasn't been replicated in the array. To add the fillet, click the 'Add feature' button, then either select part of the original fillet by clicking it in the preview window. Alternatively, you can swap to the Model tab and select the fillet in the file tree. Now, when you click OK on the linear pattern dialog in the Tasks tab, you will see the array with the fillets added to each instance of the pocket.



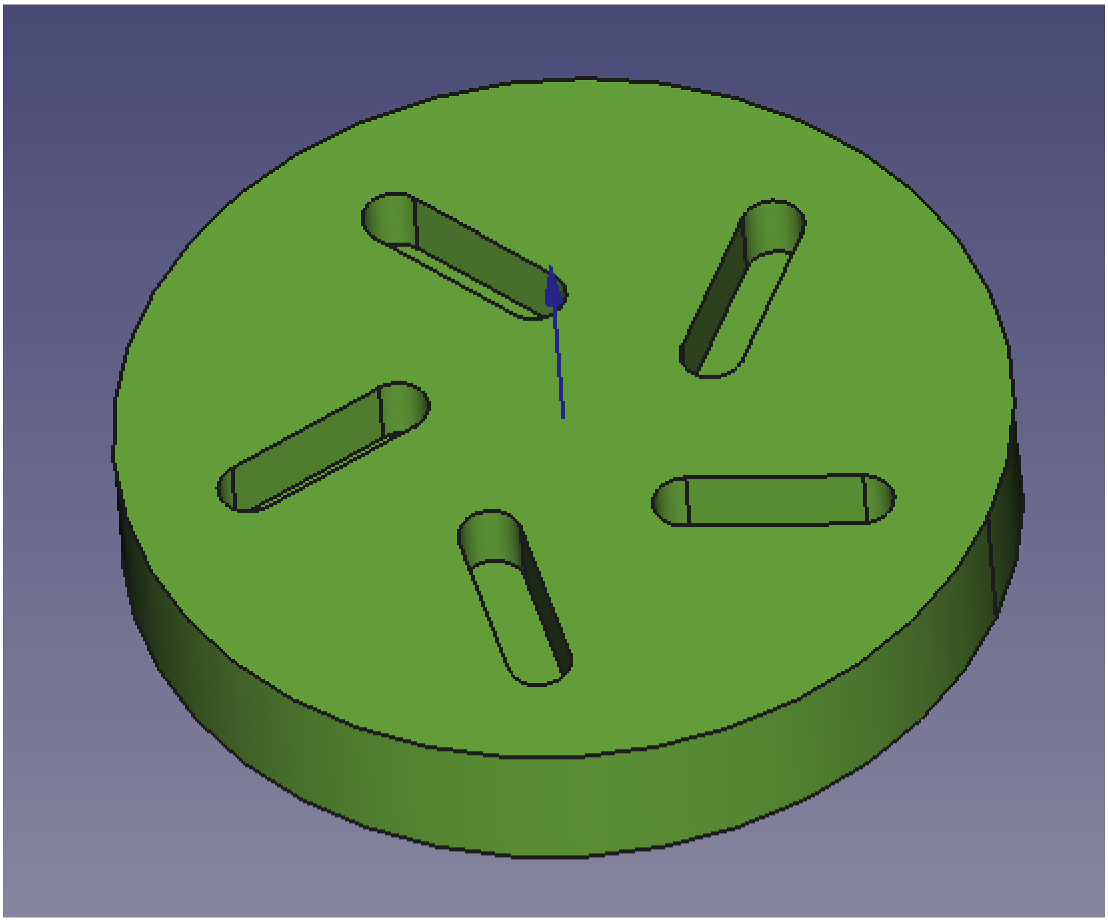
Removing a feature is slightly counter-intuitive in that you can’t select the feature to remove in the list in the ‘Add/remove feature’ box in the dialog. Similar to adding a feature, you click the ‘Remove feature’ button and then highlight the feature in the preview or in the file tree view.

Round And Round
Sometimes you might want to make an array of objects radially. As an example, start a new body and in a sketch on the XY plane, draw a circle with 30 mm radius; constrain the centre of the circle to the datum point. Close the sketch and pad the circle. Similar to your earlier example, create a sketch on the top surface of your cylinder and draw a small slot to create a pocket. First, leave the slot without constraints and position it somewhere in the upper left quarter of your cylinder face and perform the pocket.

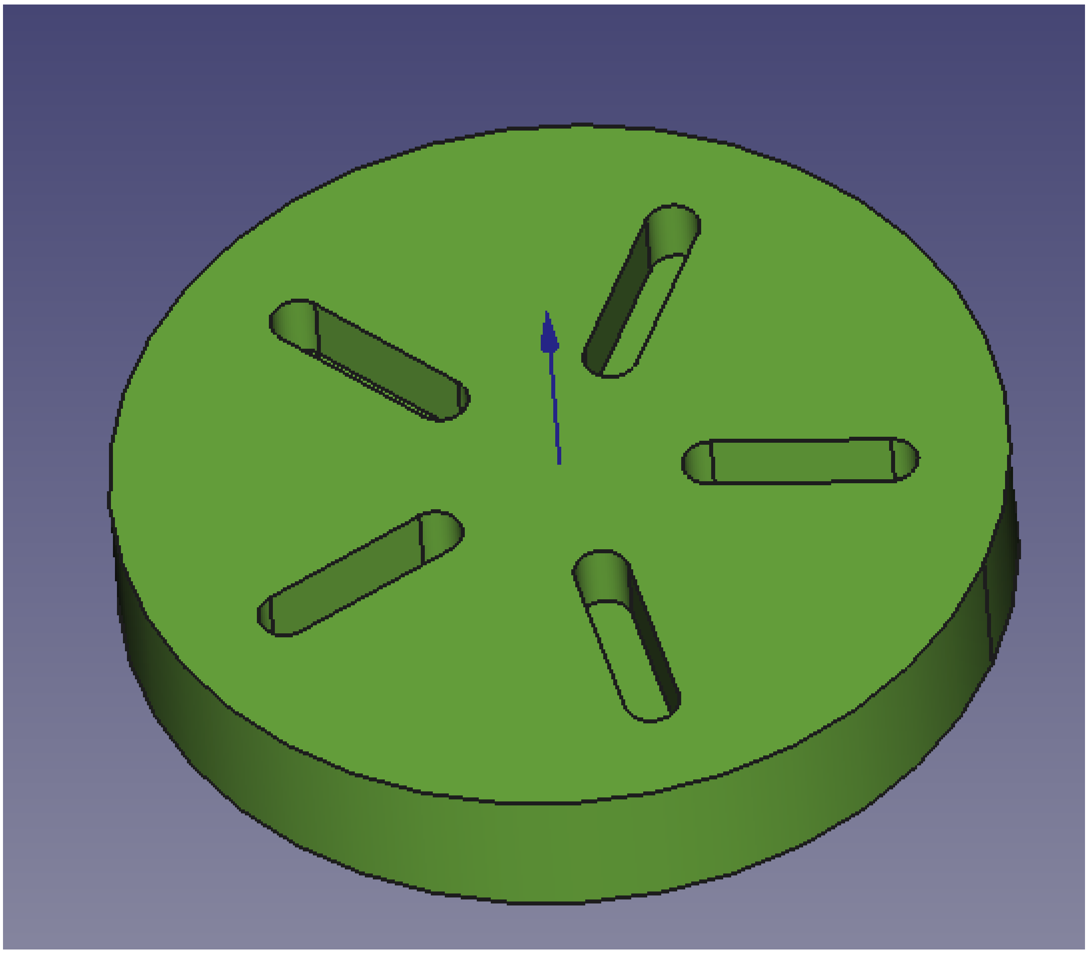


Again, with nothing highlighted, click the ‘Create a polar pattern feature’ tool icon.

The resulting dialog box is similar to your earlier linear pattern tool. Select the pocket that you made. Scrolling down, leave the axis and angle settings as the default and set the occurrences to 5. If you have the 'Update view' box ticked, you should see that the pocket is replicated and rotated around the datum point. As your original slot was off axis, the array positioned the repeated slots relative to this original position.



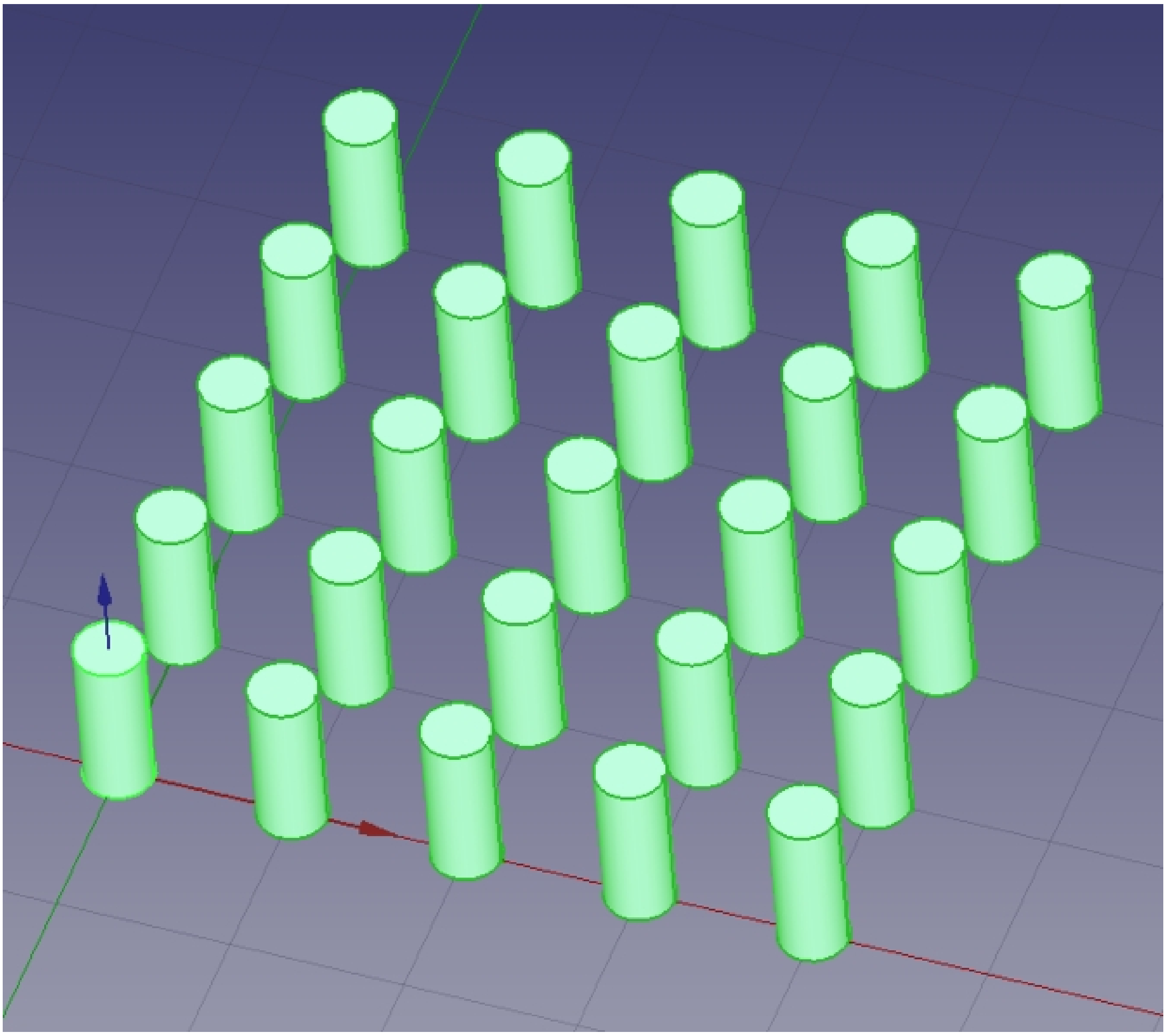
You can of course go all the way back to the root sketch of the pocket and adjustments will be recomputed to the pattern array. Close the polar pattern dialog and open the slot sketch in the file tree. In the slot sketch, constrain the slot so that the centre of the slot is positioned on the Y axis datum line.



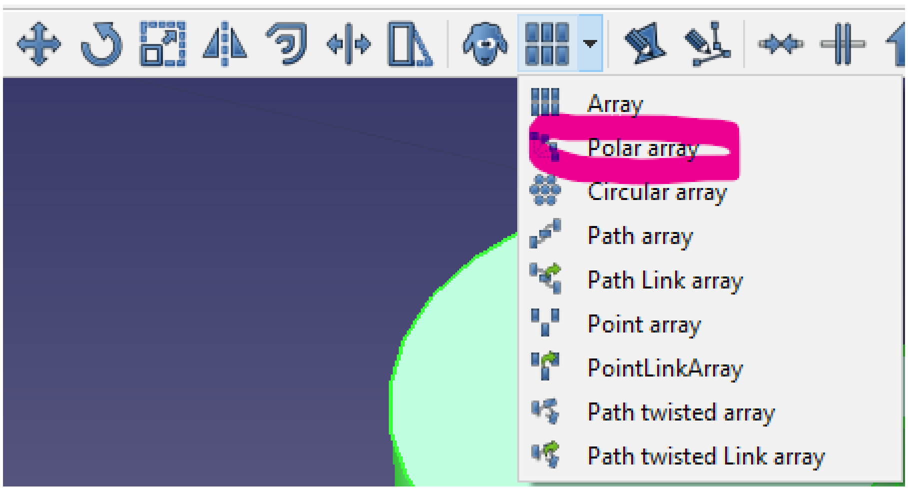
On Repeat
Many times you will want to create more complex arrays, sometimes with more complex geometries or sometimes not in a single continuous body, which means you would be working outside of the Part Design Workbench. The built-in Draft Workbench has some array tools that can be extremely useful for this, but it’s a useful Workbench for much more as well.

As a simple example, let’s start a new empty project and move to the Part Workbench. Click the ‘Create a cylinder’ tool to create the default 10 mm tall cylinder with 2 mm radius. Next, use the drop-down menu to move over to the Draft Workbench. Let’s highlight the cylinder in the file tree and then click the array tool icon, which resembles a collection of six blue rectangles.

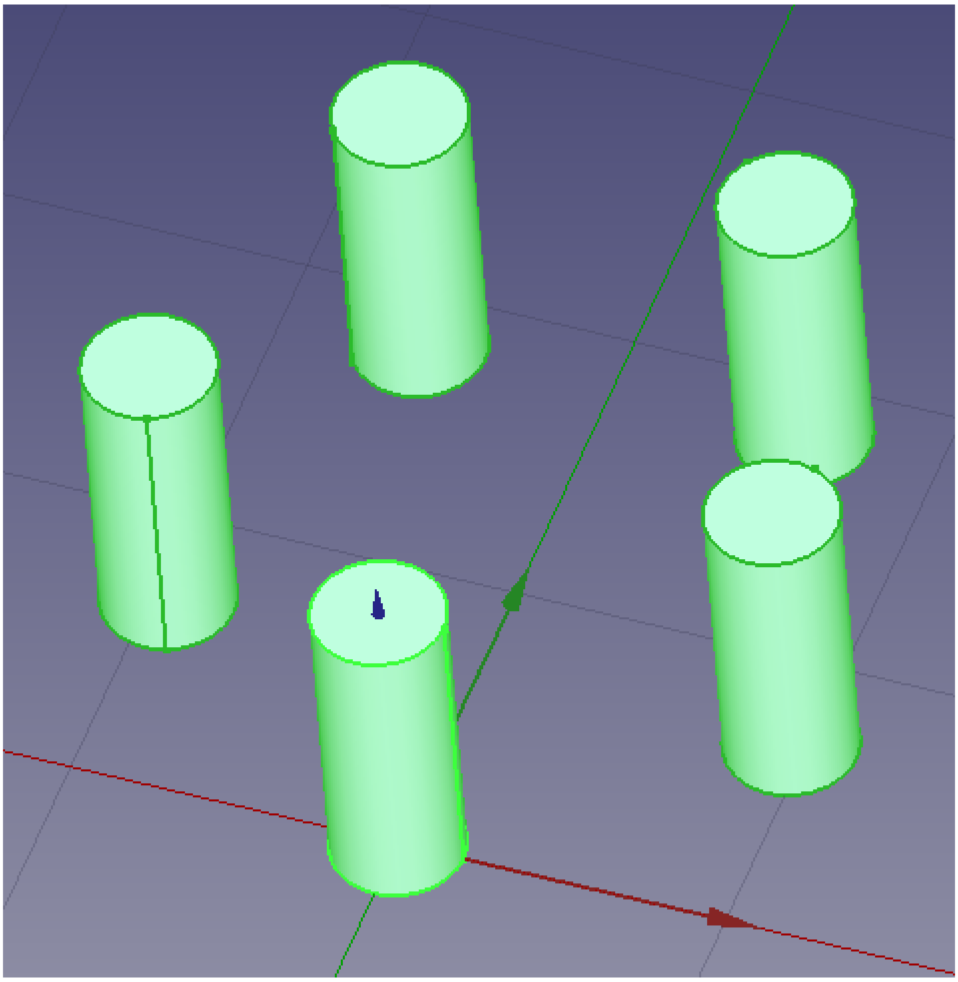
In the dialog box, the first items you can adjust are the ‘Number of elements’; edit the X and Y values to 5 and leave the Z at 1. This is then aiming to create a rectangular array of 5×5 of the cylinder object. Next, you can edit the spacings of the array. It’s slightly counter-intuitive in that the X, Y, and Z elements each have an X, Y, and Z input box, but if you play with the values and apply the array, you’ll soon see what effect these have. For now, in the ‘X intervals’ section, set X to 10 mm. In the ‘Y Intervals’ section, set Y to 10 mm and leave the Z, as you only have one layer in your array. Clicking OK, you should now see an array of your cylinders spaced at the intervals you set.



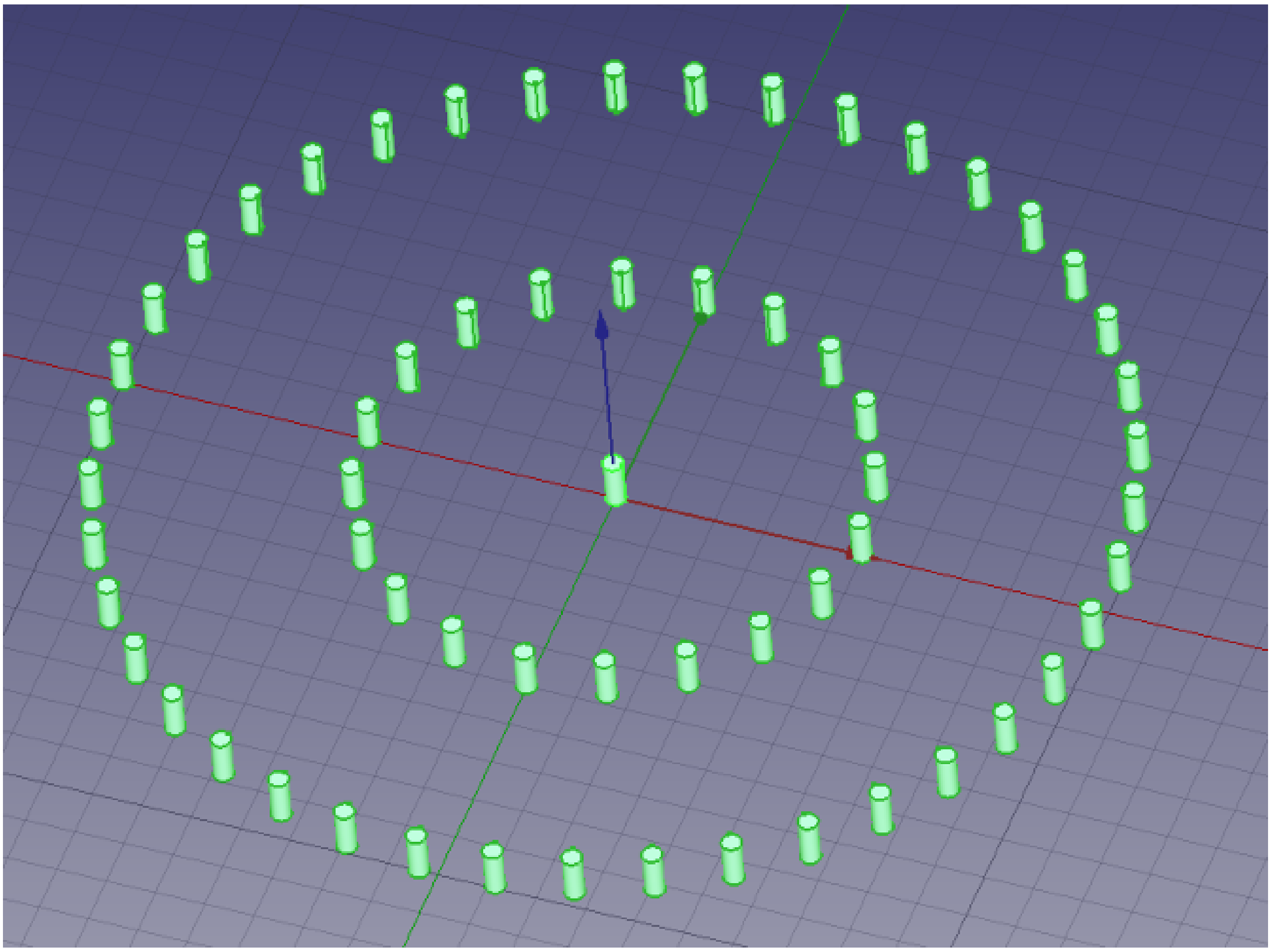
Highlight the array item in the file tree and delete it. You should now see your cylinder item and if you highlight it and press the SPACE bar, we'll make it visible again. Next, let's try the 'Polar array' tool from the Array tools drop-down menu.



In the dialog, leave the polar angle set to 360 degrees to create a full circle array and set the number of elements to 5. You need to add a point which is the centre of rotation; if you left the centre of rotation set at the 0,0,0 point, you would simply create five cylinders placed on top of each other. Change the Y co-ordinate to 10 mm and then click the OK button. You should now see a circular array of five cylinders appear.



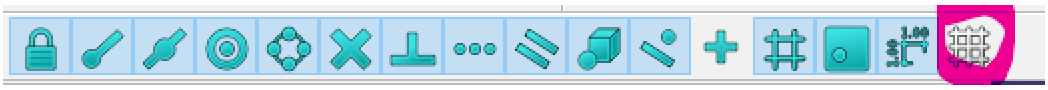
Delete the polar array and let's try the circular array tool. Highlight your test cylinder and click the 'Circular array' tool icon. This tool works differently in that the number of copied objects in the array is dependent on the size and spacing of the array rather than a directly input value. In the dialog box, the 'Radial distance' is the radius of each circle in the created array; you set this to 50 mm for a test. If you don't change the centre of rotation and leave it at the datum or zero point, then the first circle in your array will be made with a 50 mm radius from this point. The second circular layer will be made at a further 50 mm from the first circle so therefore will have a 100 mm radius, etc. Next, the 'Tangential difference' is the spaced distance between instances of the array object in the same circular layer of the array. You initially set this to 15 mm, but play with this value to see how it affects your test arrays. The 'Number of circular' is the number of concentric circles that are added, spaced as you said at the radial distance; you set this to 3 for your test. Finally, there is the Symmetry input. It's difficult to explain the effect, but this value controls the number of symmetric planes an array has – the best thing is to start with it set to 1 and then change the values and see what effect it has on your array! With the above settings, you should see an array similar to this.



We’ve seen that the Draft Workbench array tools can be used to create arrays of copies of a part, but they can also be applied directly to sketches and even be applied to bodies created on the Part Design Workbench. The caveat with arrays of a Part Design body is that the array will usually create separate objects that cannot be considered part of the original body. When this occurs, the array is listed outside of the active body in the file tree and the original body part visibility is toggled off.

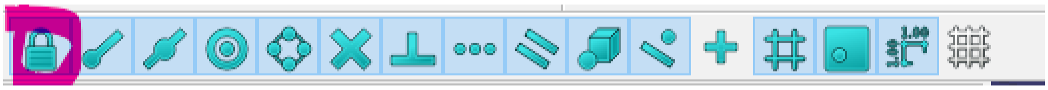
Curved Line
As a final example exploring arrays, let's create a complex-looking decorative array using other Draft Workbench tools to create the underlying geometries.

Create a new project and move to the Draft Workbench. You should see a grid on the XY plane. If you don't, find the 'Toggle Draft Grid' icon, which looks like a mesh and is at the end of the snap toolbar icons; toggle this button until you see a draft grid.

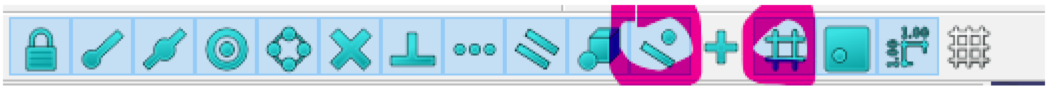


You may notice that the Draft Workbench has a collection of drawing tools that look similar to the Sketcher drawing tools. They do work in very similar ways, but in the Draft Workbench, wires and edges are drawn without constraints and use either snapping to the grid or other objects, or inputting positional coordinates.

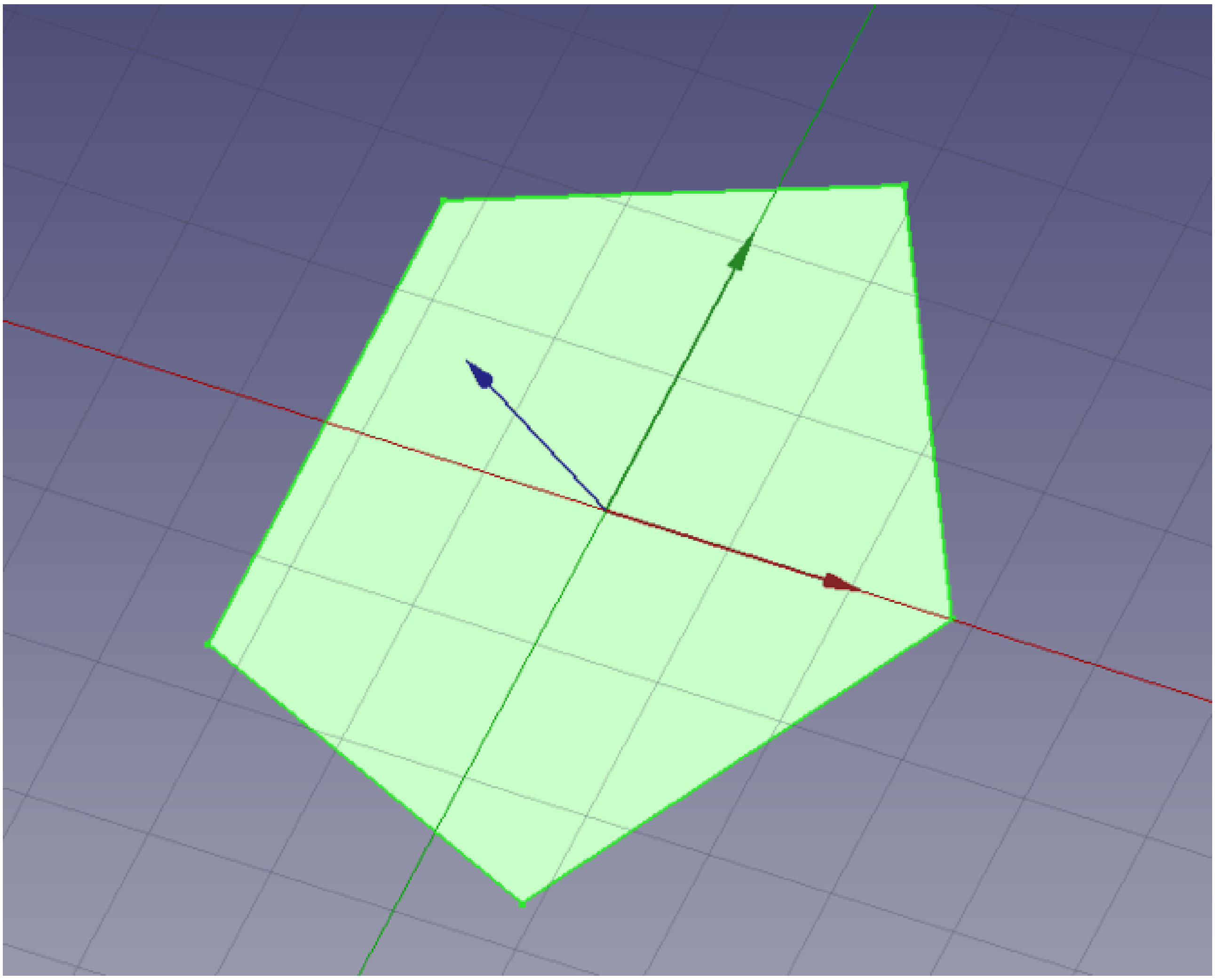
As such, the Draft Workbench has a lot of options for different snapping styles. For your example, you need to enable snapping by clicking the 'Snap Lock On/Off' padlock icon.



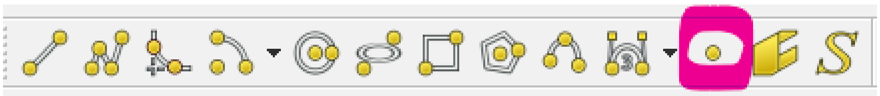
This should now activate the other snap icons. You want 'Snap grid' and 'Snap near' to both be on.



First, let's draw a polygon on the Draft grid so that it sits on the XY plane. Click the 'Create a regular polygon' tool icon. In the properties dialog you can set the number of sides you want for your polygon. Set it to 5. Then left-click to draw a polygon around the datum or zero point. To help you draw it accurately, zoom in towards the datum point and you should see the cursor lock to the grid before you click to draw. Adjust the polygon placing circle to roughly 6 mm in diameter. If you create the default triangle, you can highlight the Polygon object in the file tree and adjust the number of sides in its properties table.

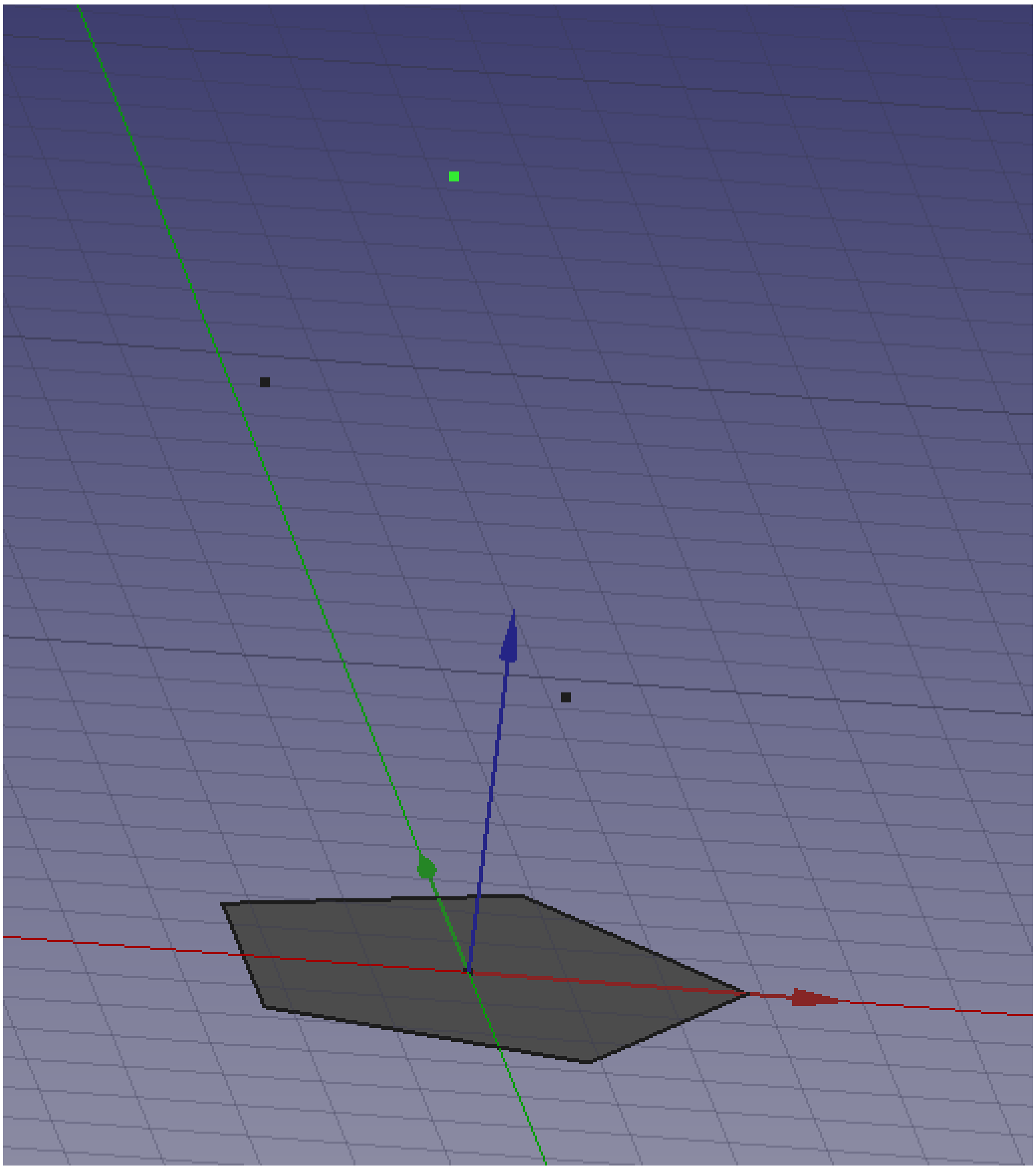


Next, let’s create some points which you will use to create a curved line in three axes. The first point of the line is going to be the centre of your polygon, or the zero point. Click the ‘Create a point object’ and click to place a point at the datum.

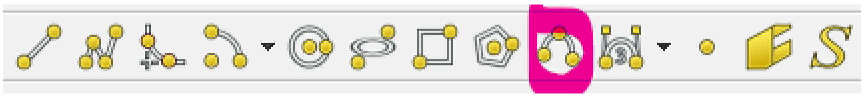


Next, let’s add a second point and then raise it on the Z axis. You are aiming to create a curve, so place this second point on the XY grid at a random point close to the zero but not on it. You went with an X co-ordinate of 6 and a Y co-ordinate of -2. Having created this second point, you highlighted it in the file tree and, in the dialog, adjusted the Z axis position to 30 mm. You may need to adjust the zoom, but you should see this point now rise up above the plane.

You added two more points above: the next had the co-ordinates X = -28, Y = 0, Z = 60, and the final point was at X = -10, Y= 9, Z = 80.

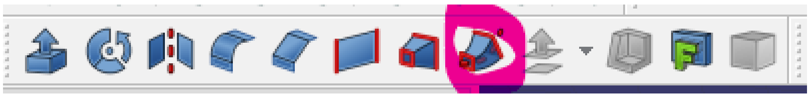


Creating a curve between the points you just made is easy with the 'Create a multiple-point B-spline' tool.

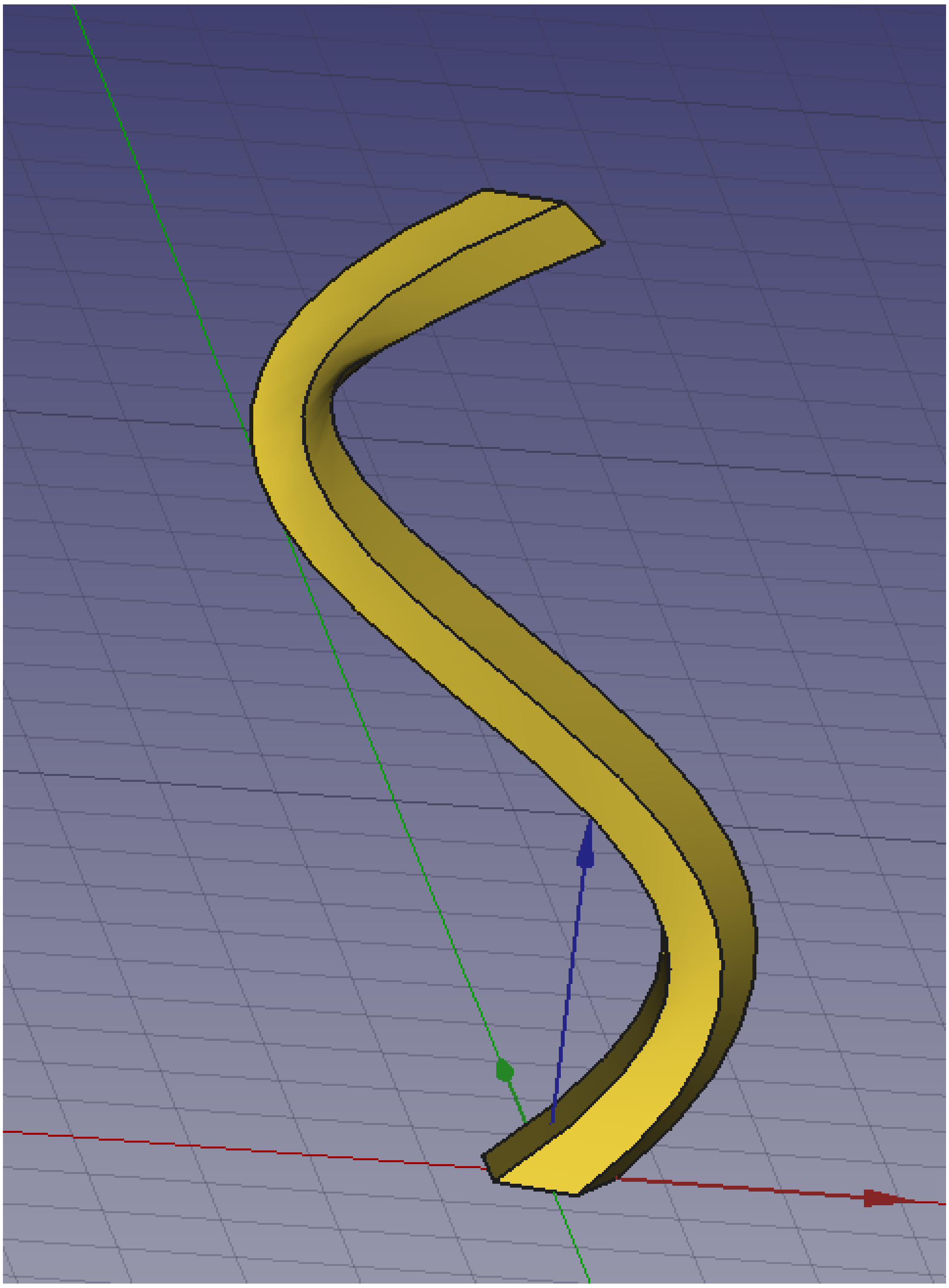


Select this tool and hover near the upmost point you just made. When you are hovering, if you are too far away from the raised point, you'll see that next to the pointer position there is an icon that looks like the Draft grid icon; this indicates that if you click, you'd be snapping to the grid. If you are close enough to the raised point, this icon disappears, and if you left-click, you will attach the B-spline to the point. Left-click on the upper point and then move to the next lower point and left-click again; continue to do this until you reach the last point at the zero position. Left-click to attach the last point and then click the 'Close' button in the properties dialog.

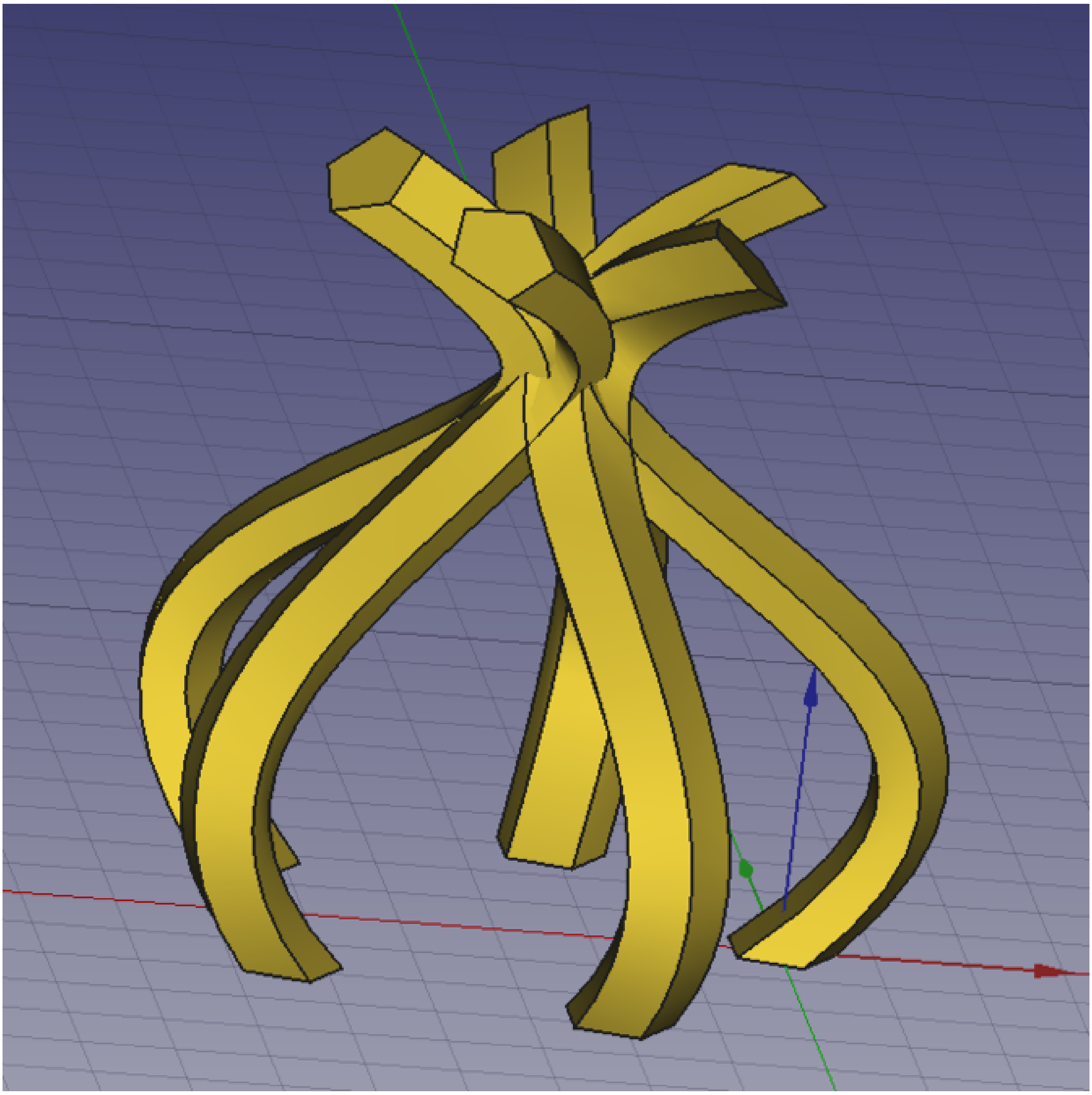
Just for a moment, you are now going to switch to the Part Workbench to use the sweep utility. On the Part Workbench, click the ‘Utility to sweep’ tool icon.



In the dialog, select the Polygon item you drew from the left-hand list, and click the right-facing arrow to add it to the right-hand column. Next, click the ‘Sweep Path’ button, then left-click on the B-spline curve you drew in the preview window and click ‘Done’. Check the ‘Create solid’ box and click OK. You should now have a curvy swept hexagonal object in the preview.



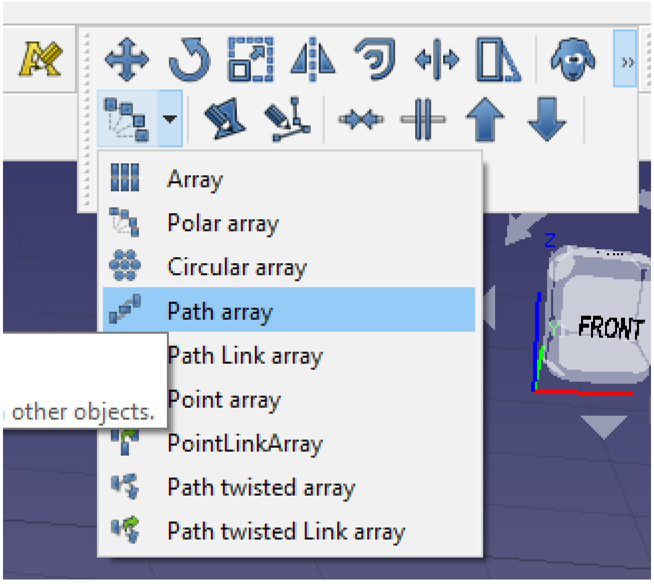
Moving back to the Draft Workbench, select the swept object in the file tree and then click the polar array tool icon. You put your centre of rotation at around -30 mm on the X axis to create your array of five of your swept items.



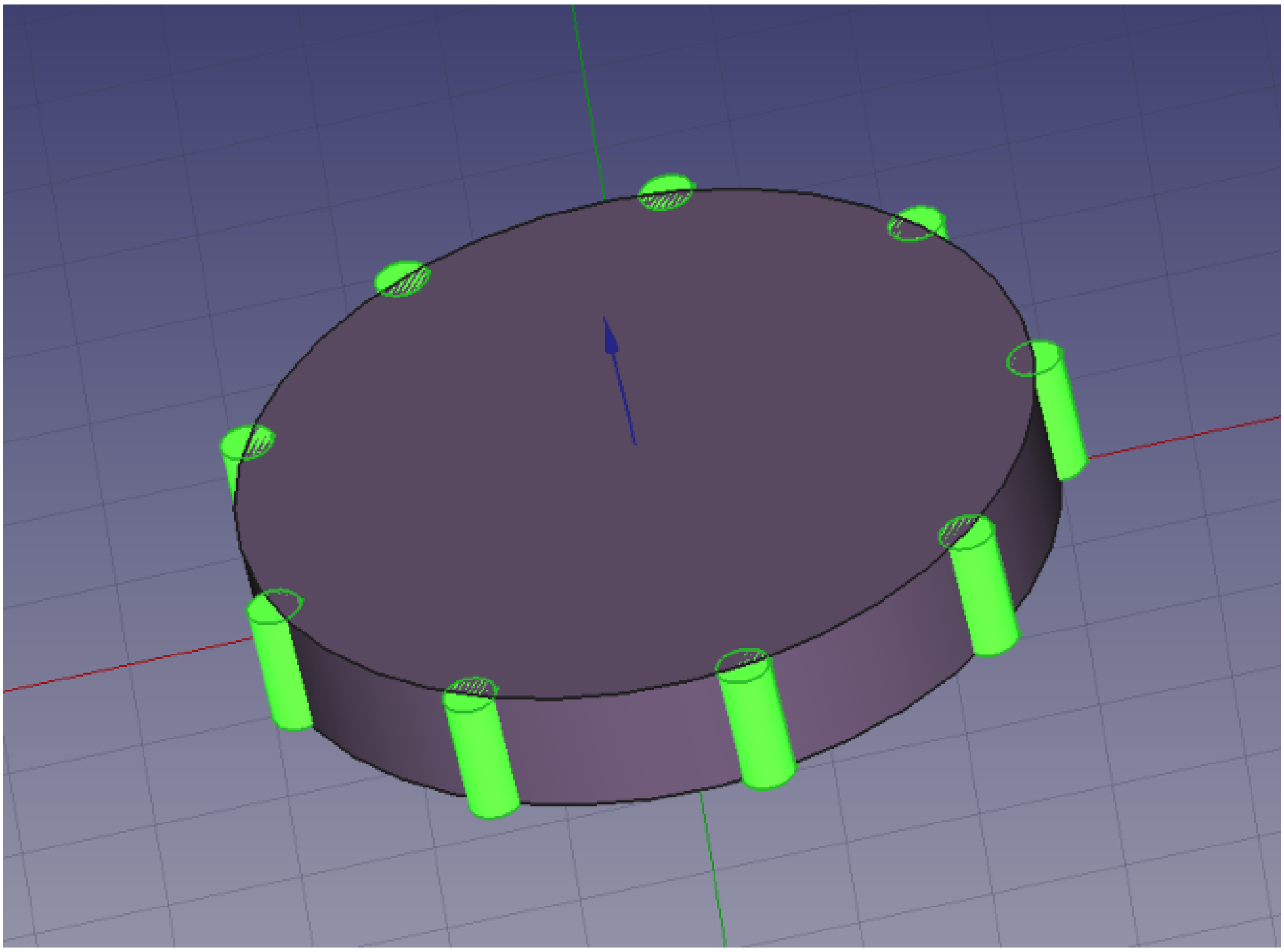
Follow The Path

Sometimes you will want to create an array of objects, but constrain them onto another object or path. This can be achieved using the path array tool. In a new project, once again go to the Part Workbench and create a first cylinder and set the radius to 30 mm and leave it at 10 mm tall.

Then create a second cylinder and leave it with the default dimensions of 10 mm tall and 2 mm radius. Move back to the Draft Workbench and first select the object of which you wish to create an array. Let's select your second, smaller cylinder. Next, you select the path you want the array of objects to be connected to. Holding CTRL/CMD, click the lower outer edge of the larger cylinder. Then click the 'Path array' tool.



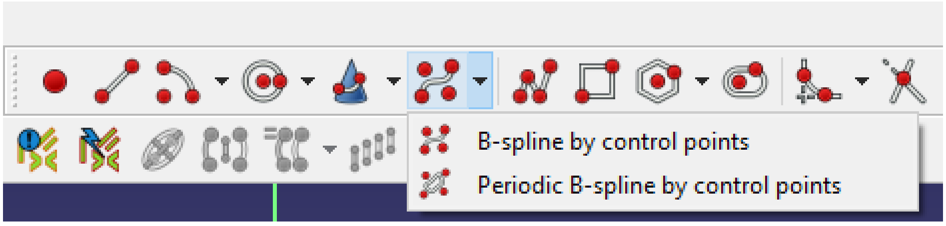
This applies an array without a pop-up dialog box, but the resulting array has some editable parameters in the Combo View window. You should see the array has been created with four instances of the smaller cylinder placed equidistantly around the larger cylinder edge. If you highlight the array object in the file tree and adjust the 'Count' parameter, it will add or remove instances from the array and redistribute the count to maintain equal distances between each instance.



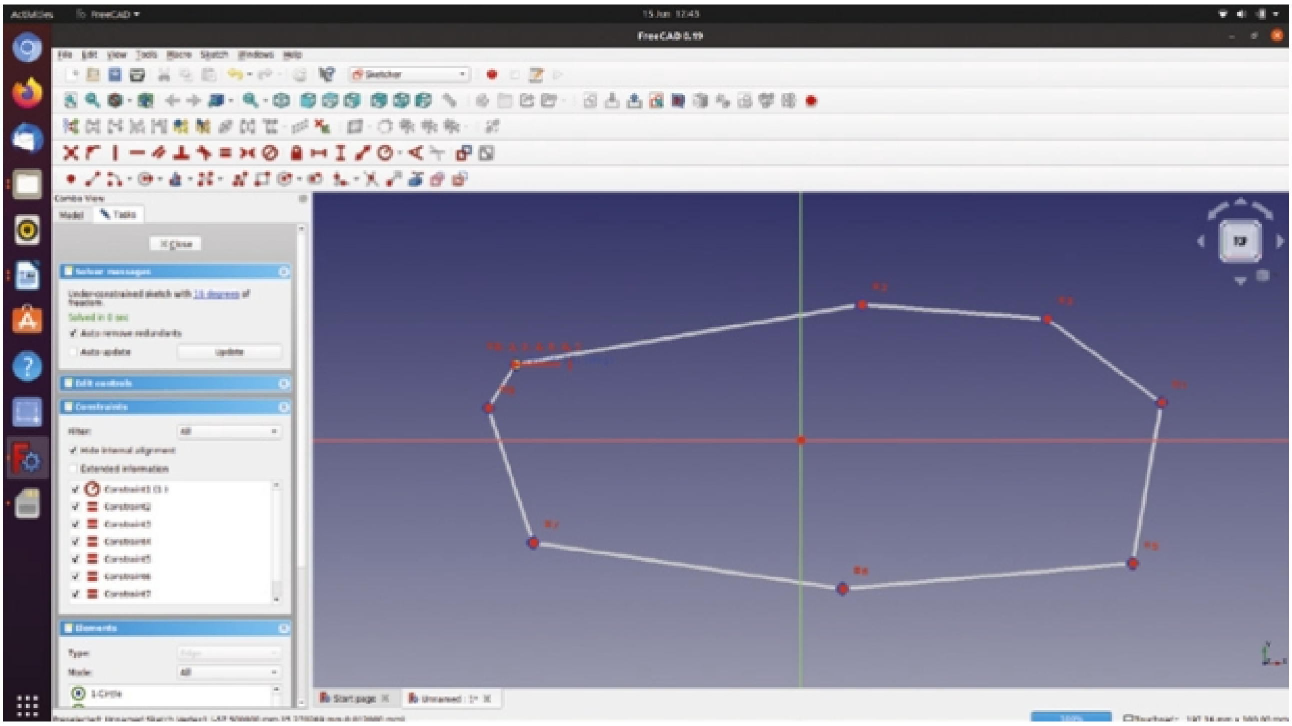
Quick Tip
Press SHIFT+V or SHIFT+H to quickly add a horizontal or vertical distance constraint between two selected points or a selected edge.

Surfaces and projection - Turn lines and sketches into solid objects

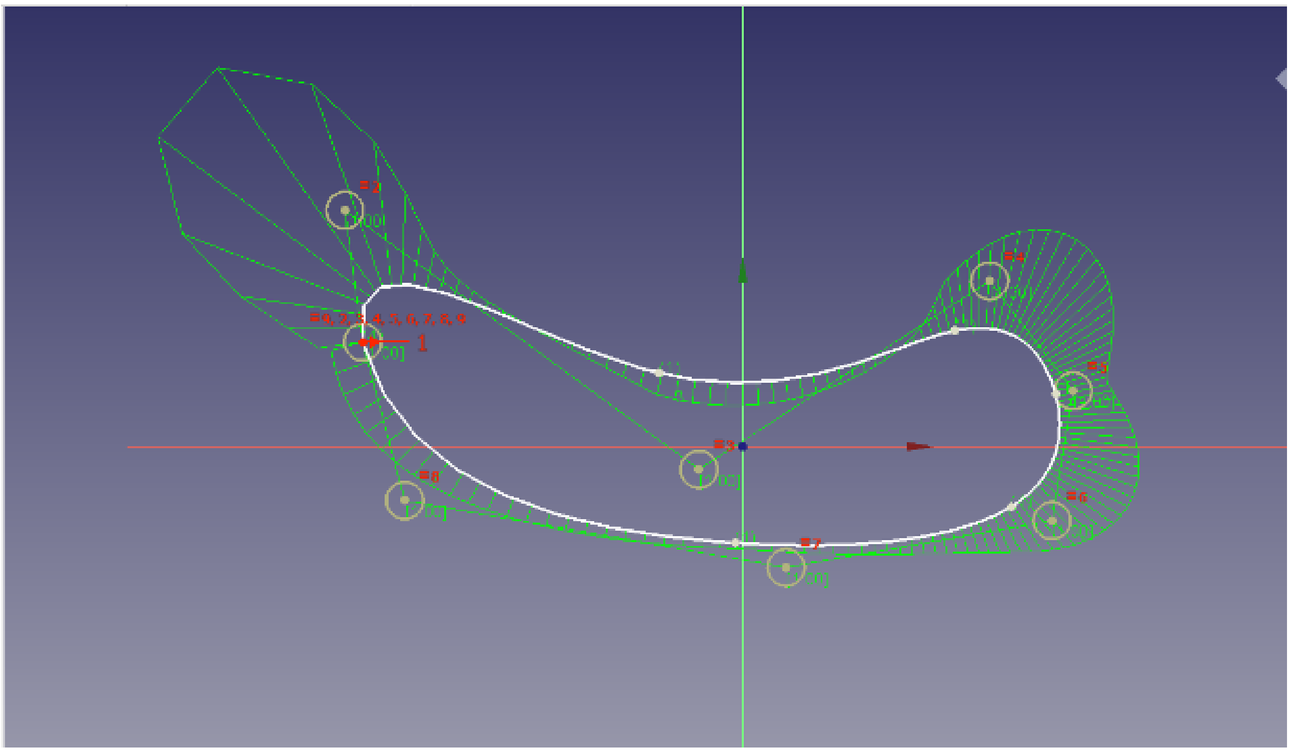
You are going to start with a sketch, but you are going to explore creating surfaces from sketches or wires to build an object. To begin, let's go to the Sketcher Workbench and create a new sketch on the XY plane.
Select the 'B-spline by control points' option from the 'Create a B-spline in the sketch' drop-down in the drawing tools area.



Draw a collection of B-splines around the origin point, making sure that the final B-spline ends at the first B-spline point.

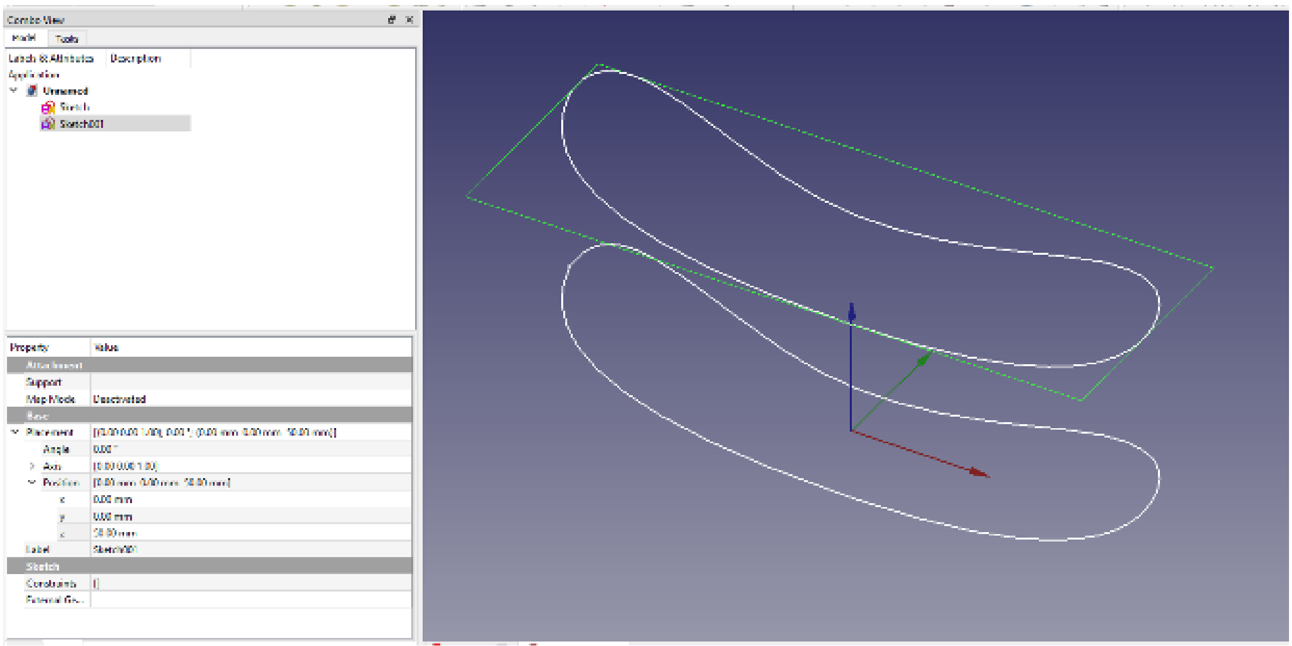


You aren't aiming for anything in particular, just an interesting curvy outline that has some inner and outer swooping curves. Play around moving the B-spline control points until you get a shape you like that doesn't have too tight a set of curves.

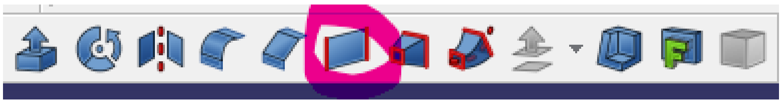


You don’t need to constrain the B-spline that you have just created, so go ahead and close the sketch.

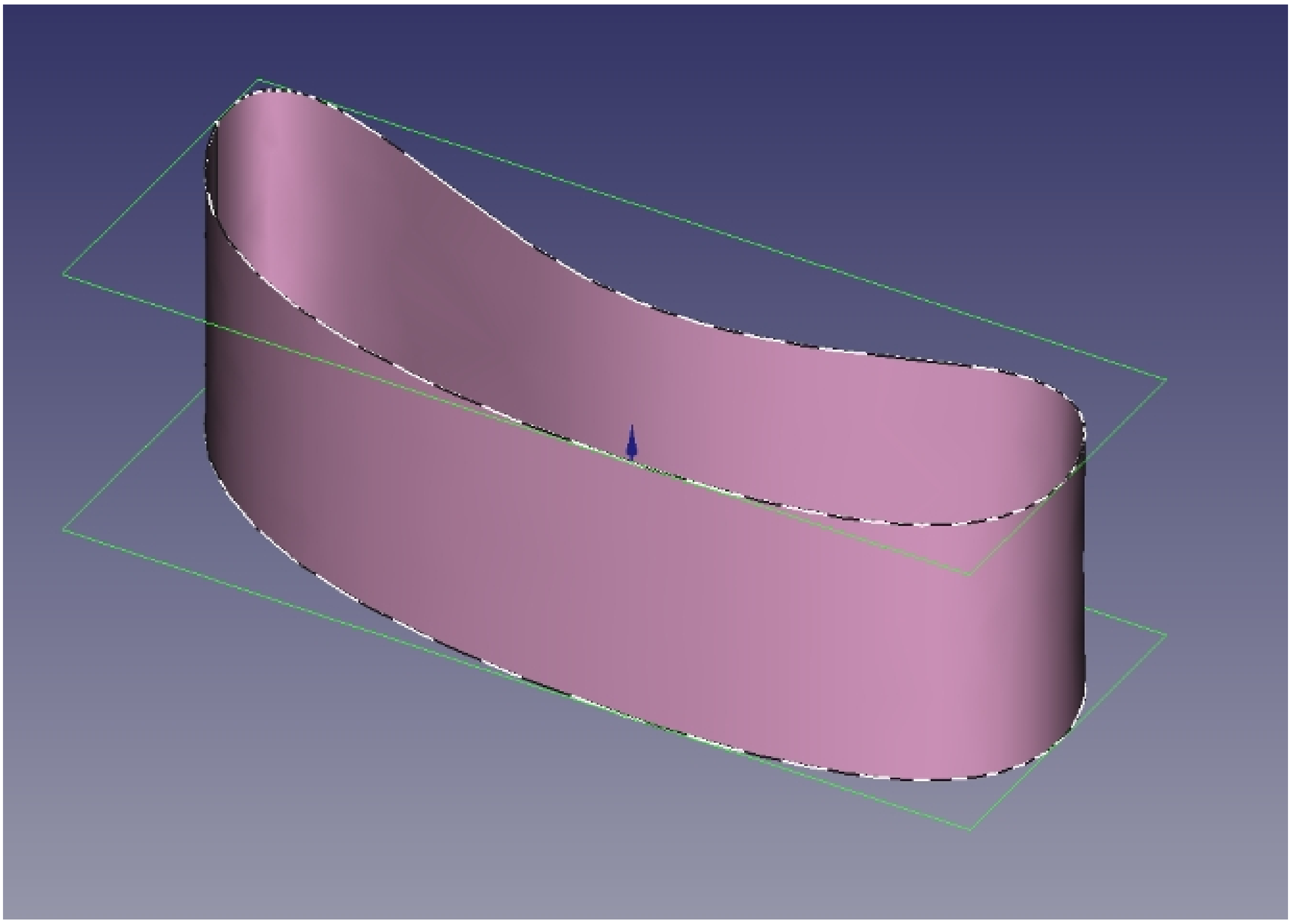
Moving back to the Part Workbench, select the sketch in the file tree, and copy and paste the sketch to create a second identical sketch that will appear as ‘Sketch001’ in the file tree. Select this second sketch in the file tree and, in the sketch dialog, increase the Z axis value in the Placement > Position drop-down to move the second sketch directly above the first. You moved your copied sketch upwards by around 50 mm.



You should now see, in the preview view, two sketches perfectly aligned above one another. Select both the sketches in the file tree and let's create your first surface by clicking the 'Create a ruled surface' tool icon.

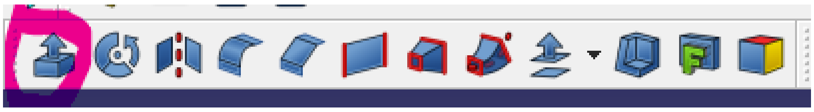


Hopefully, you now see that a surface has been created between your two sketches, similar to this.



You'll notice that, as it stands, your surface doesn't have any thickness and your object isn't a solid part with a top and a bottom. If you wanted to create a solid shape, of course, you would have used a single sketch and the pad or extrude tools. Instead, let's aim to make your curved wall into a sort of tray with a base, and turn your ruled surface into a wall with some thickness, but open at the top.

First, let's make the ruled surface you created, have some thickness.
But, just for fun, let's look at the wrong way to do this using the extrude tool. Doing this incorrect experiment makes us think about the projection of axes through workpieces, which you will return to as a subject later. Select your ruled surface and click the 'Extrude' tool in the dialog.



select the Y axis as a custom direction and set to extrude it a couple of mm. Apply the extrusion. Inspecting the results, you'll see that it works for some parts of the surface but not for others, as the extrusion is projected across the ruled surface in the Y axis. Applying an extrusion like this also leads to some parts of the extrusion being added inside the boundary of the ruled surface, but other parts extruded outside. Select the extrusion in the file tree and delete it, and toggle the ruled surface so that it is visible.

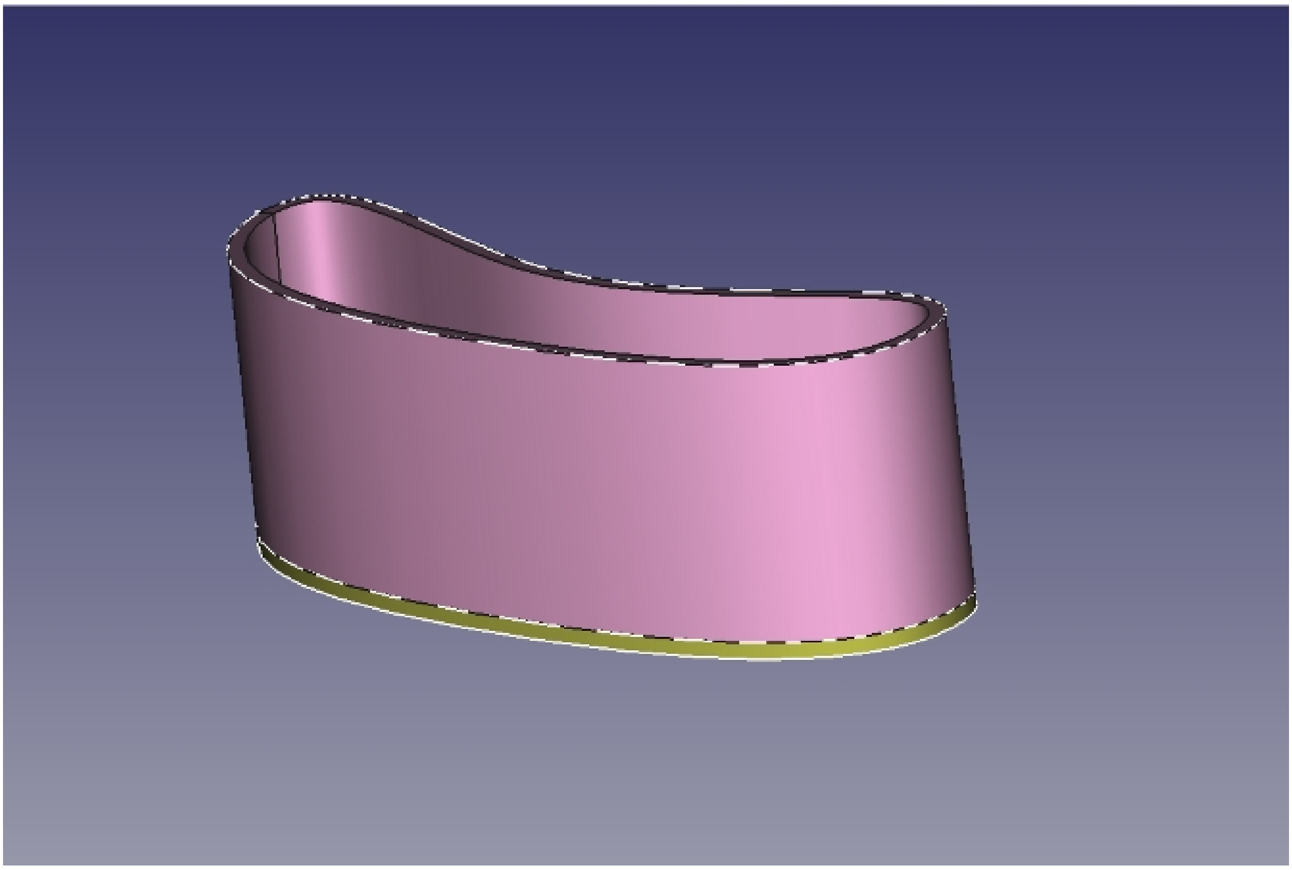
A simple way to solve this is to use the "Utility to offset in 3D" tool.

Highlight your ruled surface and then click this tool. In the dialog set your offset amount to 3mm and leave the mode setting as "skin" and the join type as "Arc". Click to check the "update view" box in the lower left-hand area of the dialog. You should see that you have created an internal second surface running parallel to your original ruled surface at a distance of 3mm. If you wanted to create this offset on the outside of your ruled surface then adding a negative number orientates the new surface correctly.

You'll note that the 2 walls are separate and unfilled – click the "fill offset" option and the two ruled surfaces become a filled object.

Click OK to apply the offset. One of the best parts of using FreeCAD is the ability to apply multiple operations or tasks to a single piece of geometry.

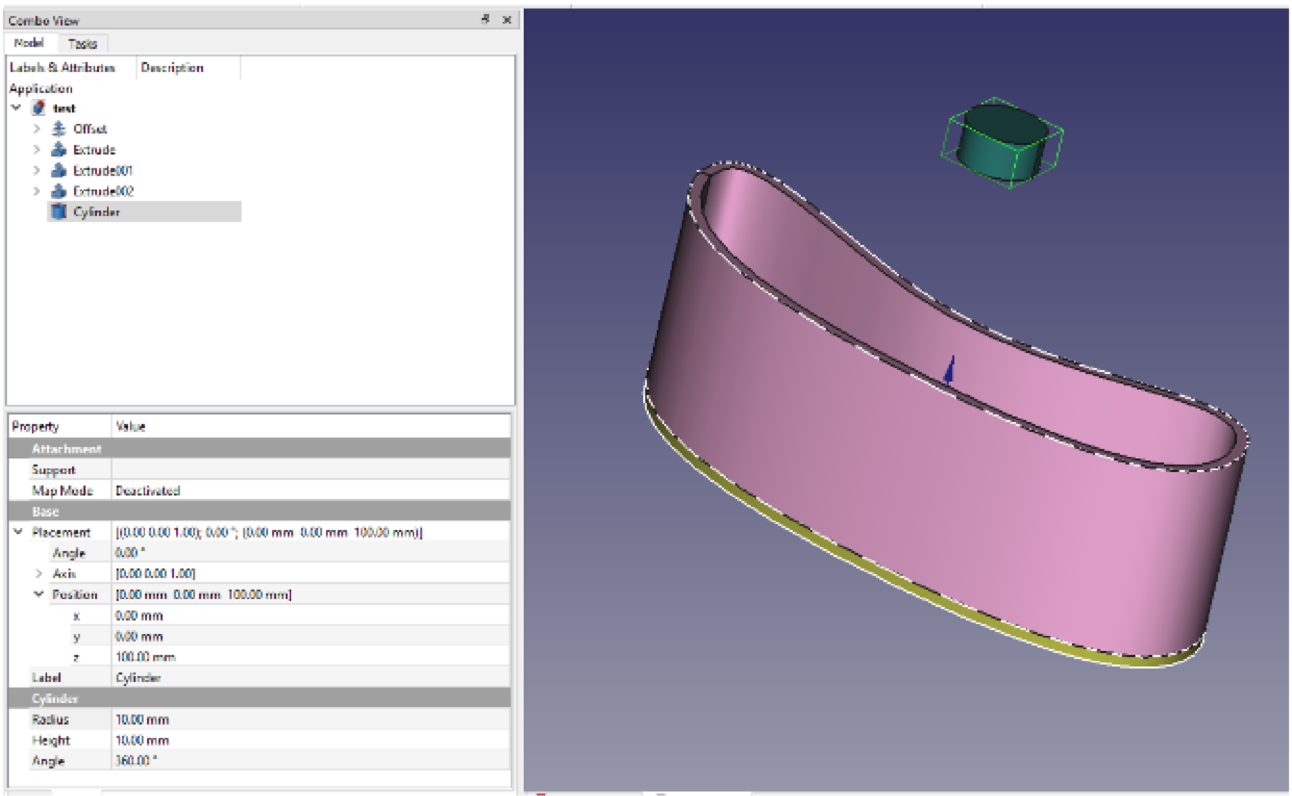
To add a base to your curvy wall, to make it into your curvy tray, you can use the drop-downs in the file tree to go all the way back and select your original sketch. You can then click the Extrude tool and add -3 mm to the Z axis custom direction and set the length option to 0. Clicking Apply, you should have added a base underneath the offset wall to complete your tray.



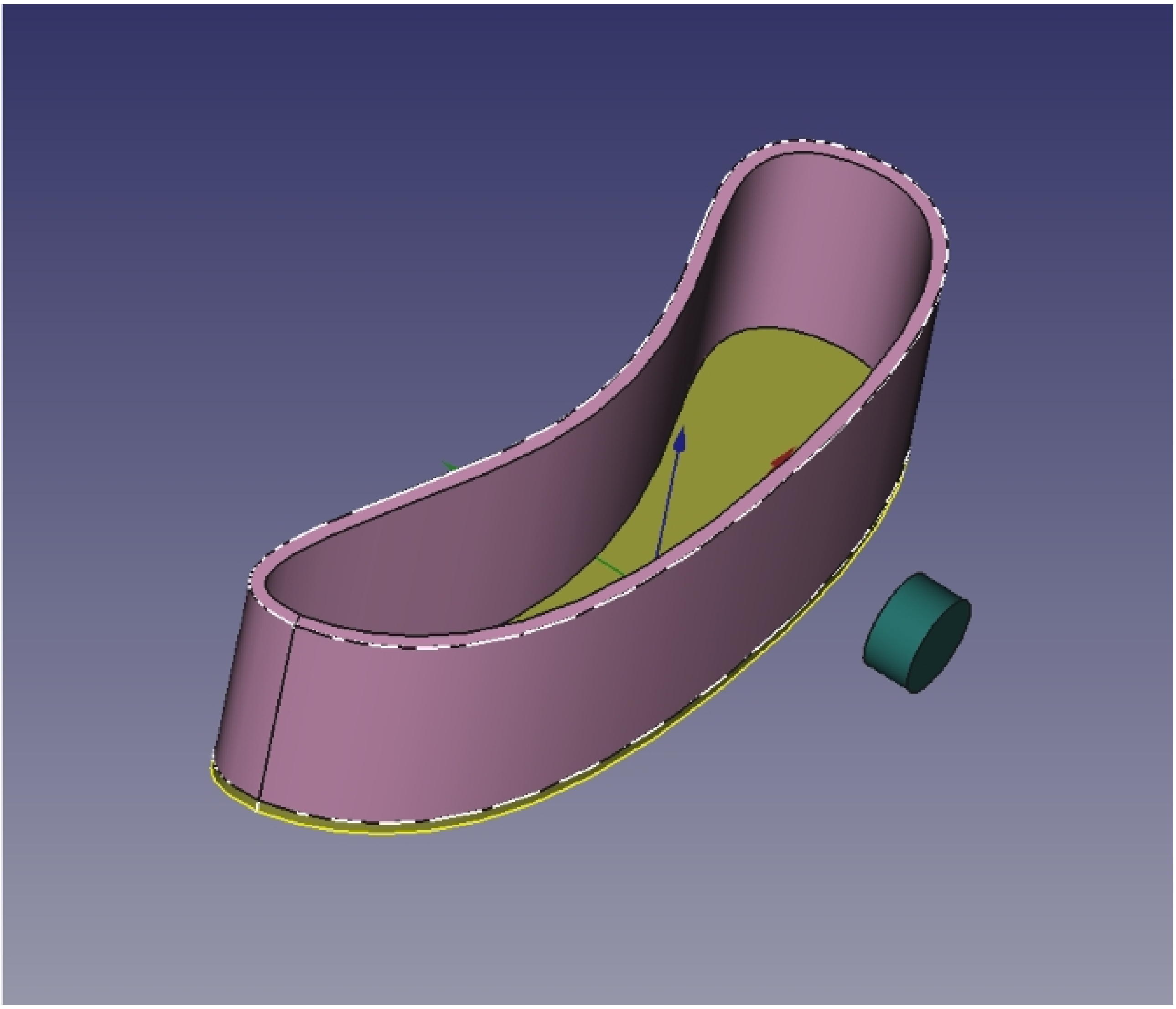
Previously, when you have worked with solid objects, you have often created sketches either attached directly to a flat face of an object or drawn on a plane, like XY or XZ, and then extruded or pocketed through an existing solid. With a curved object, such as the wall of your tray, drawing directly onto the surface is not really possible. However, you might often want to add design aspects to a curved surface. One approach to this is to use ‘projection’.

Imagine placing your hand in front of a display projector and seeing the shadow of your hand on the projection surface on the wall or whiteboard. You can do this in FreeCAD, except the projector is referred to as the camera view and is simply the current view in the preview window. The equivalent of your hand is the object you want to add to a surface, and your ruled surface is the object onto which you want to project. Whilst this might sound complex, it’s actually pretty straightforward to do.

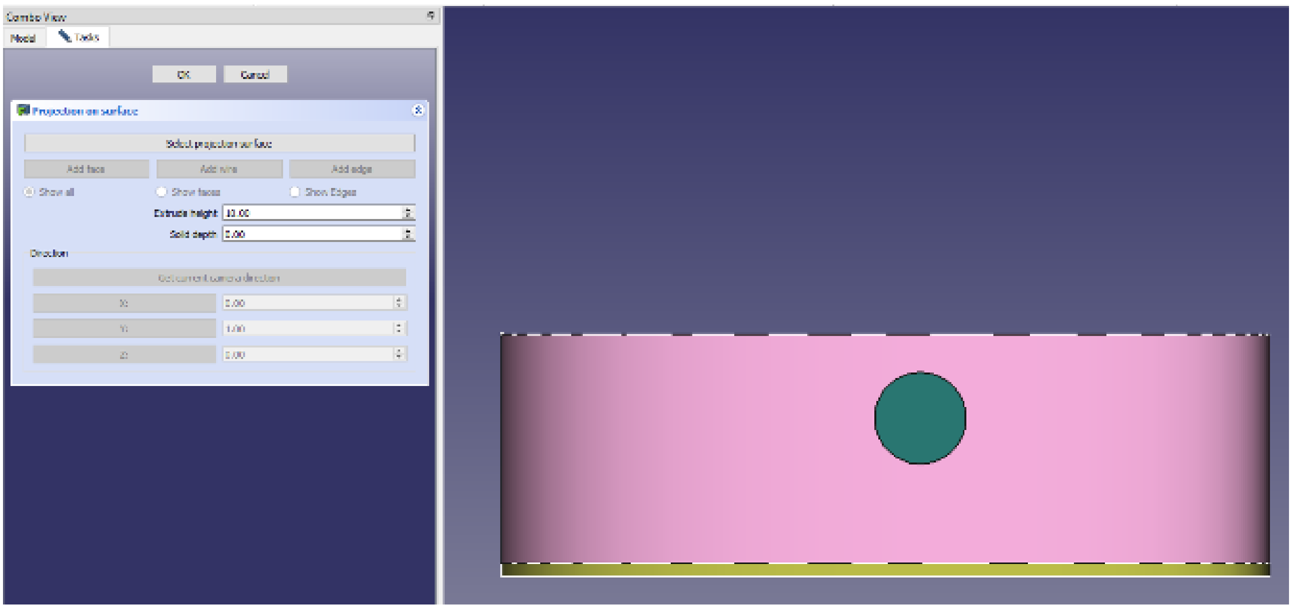
Being in Parts workbench, create a cylinder with raduis 10 mm, height 10 mm and postion it outside your curvy tray (Z = 100 mm)



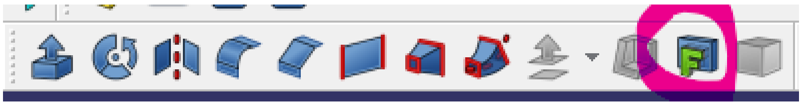
First, you need to choose which surface area of your object you want to project the logo onto. You can project parts onto any surface, but obviously if the surface is very folded, the effect isn’t going to work as well; therefore you chose a larger, smoother face.
You then need to orient that chosen face to be pointing at us when viewed in the preview, remembering that the preview window is the projection viewpoint. With that established, you then need to move your object you want to project onto the curved surface into a position in line with the projected surface and the preview window. You can do this via the usual right-clicking the object in the file tree, selecting Transform, and moving and rotating the object until you are happy with the position.
It’s not too critical how far away from the surface the object is, but it’s a good idea to leave a reasonable gap so you can see the effects of the projection. In Figure 7 overleaf, you have rotated the preview view to show how your projection was set up.



With your preview view and your projection object correctly set,

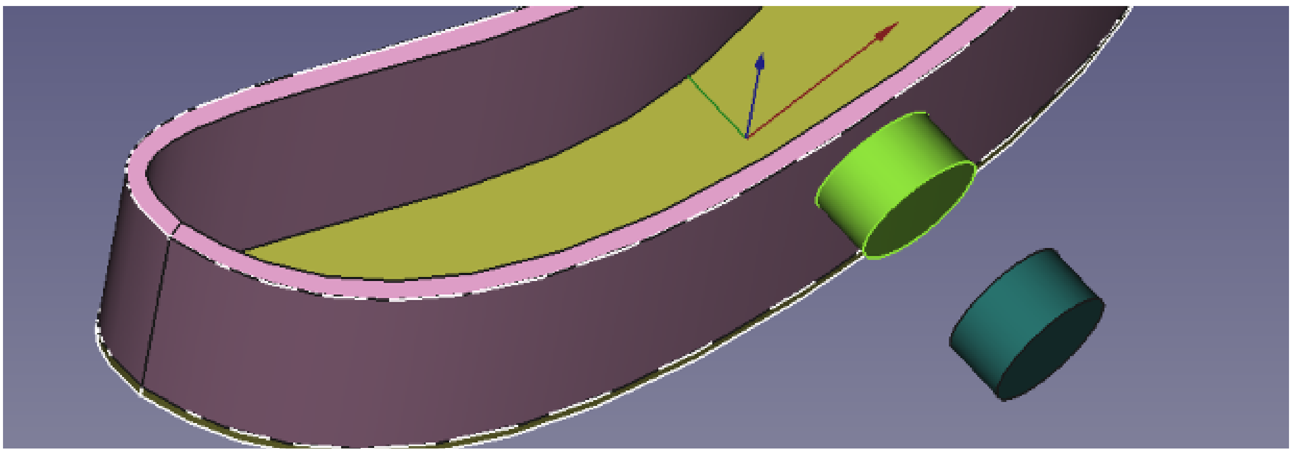


click the 'Project edges, wires, or faces of one object onto a face of another object' tool.



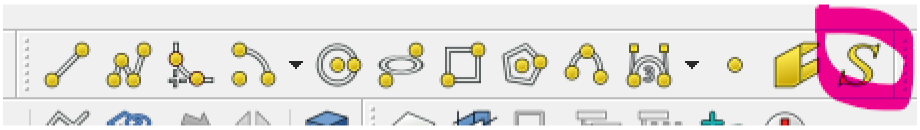
In the dialog you will see a long button that says 'Select projection surface' – click this button and then select the surface onto which you want to project. Next, click the 'Add face' button and select the face of the object you wish to project that is visible in the preview view. It may take a few seconds, but the selected face should turn a purple colour.
You'll need to change the extrude height to a value that works for the shape of your projection surface. In your example, the extrusion height was 10 mm – this gave enough height for the whole to be projected to the lowest parts and highest parts of the curve of the ruled surface. Experiment with values until you get a projection that works for you.

Finally, click 'OK' and you should now see your object projected onto the curved surface of your tray part. Of course, now you can remove or toggle the visibility of the part you used to make the projection.

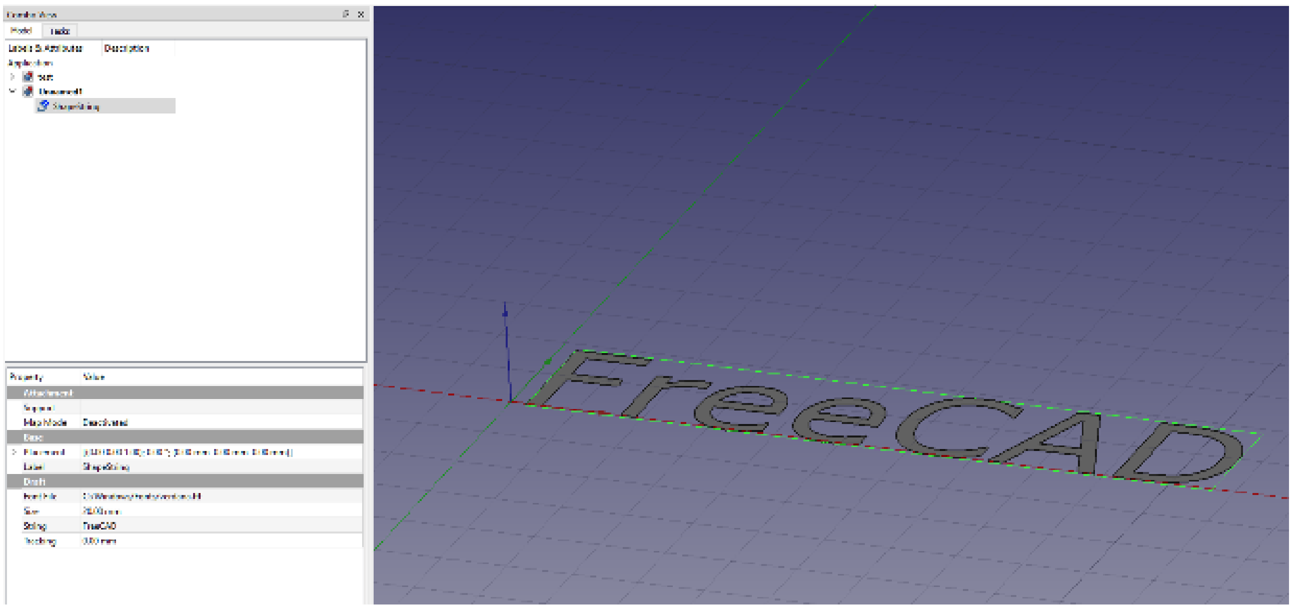


A common use for projection is to create 3D text effects. Let's first look at how to generate 3D text in FreeCAD, and then let's use some text and project it onto your curved surface again. Then you can combine projecting onto a curved surface with some Boolean part operations to create some interesting effects.

For simplicity, let's just open a new project to look at the text tool you are going to use. In the new project, open the Draft Workbench and then click the 'Create a shape from a text string' tool icon, which looks like a yellow 'S'.

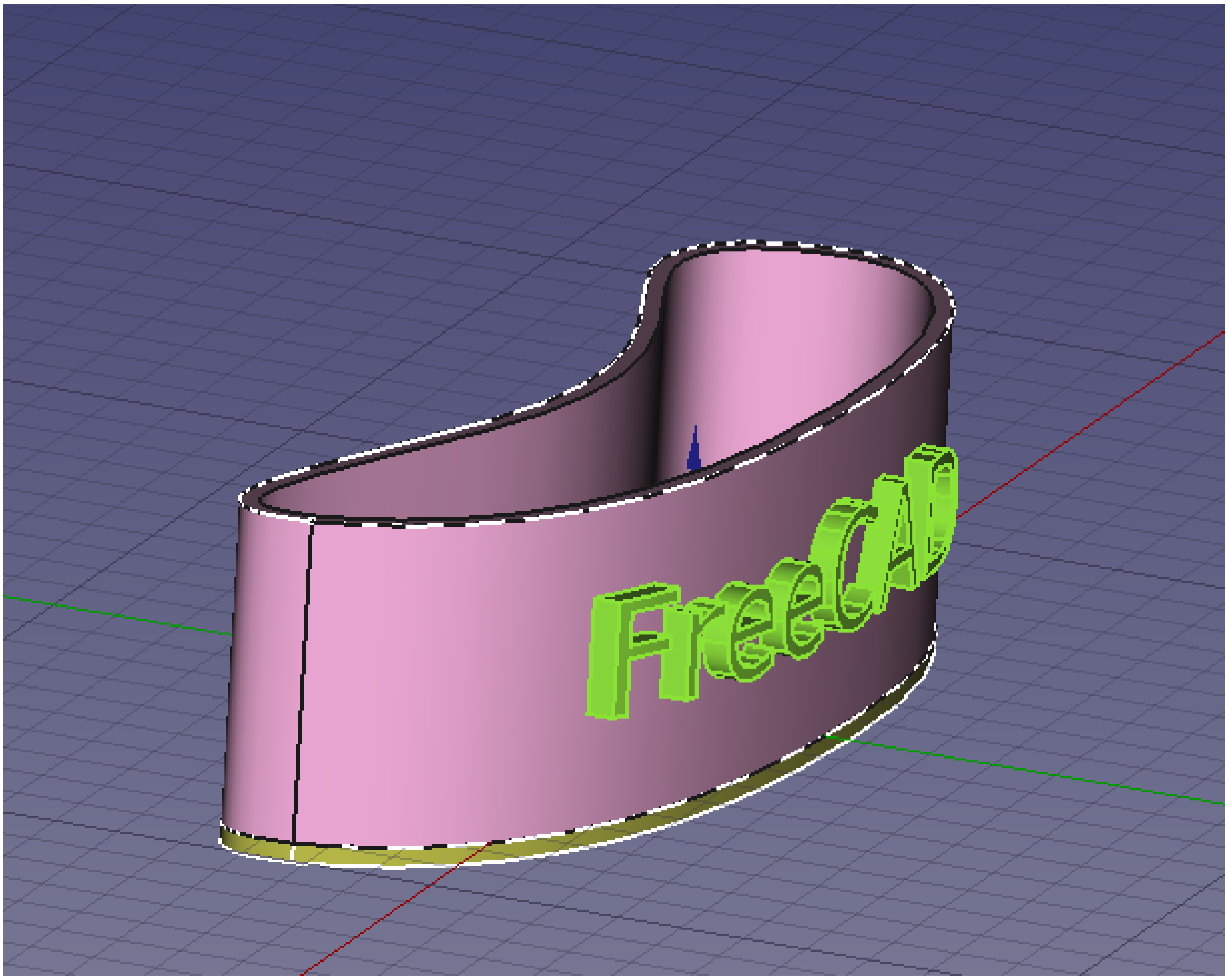


In the dialog, you can see some positional co-ordinate input boxes which select the point at which your text will be created from. If you move your pointer over the grid in the preview window, you will see that these co-ordinates react to where your pointer is. For this example, let's leave them set at 0,0,0, which you can either type in or click the 'Reset point' button.
You need to point the ShapeString tool at your font installation folder. Clicking the ellipsis at the side of the 'Font file' input box should open a file browser. On Windows 10, the location is usually C:\Windows\Fonts. Navigate to your font folder and select a font.
In the 'String' input box, type the text you want to create, and then set the height of the text, noting that the text height is not in the usual 'point' font units found in word processing packages, but rather, in millimetres. Clicking OK, you should now see the text string you input created in the draft preview window. Notice that it creates a wire/edge, but also creates a surface face.
You should also see that you have a 'Shapestring' object in the file tree.

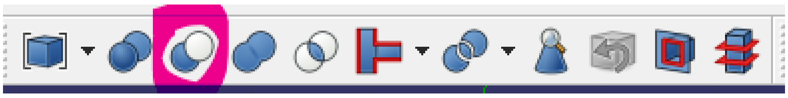


Now that you have a ShapeString object, you can move back to the Part Workbench and perform the usual sort of operations upon it. Whilst not totally necessary, if you extrude the ShapeString on the Part Workbench a little, say 3 mm, it means that you can use the simple 'Transform' operation you used earlier to move the object into your projection position.

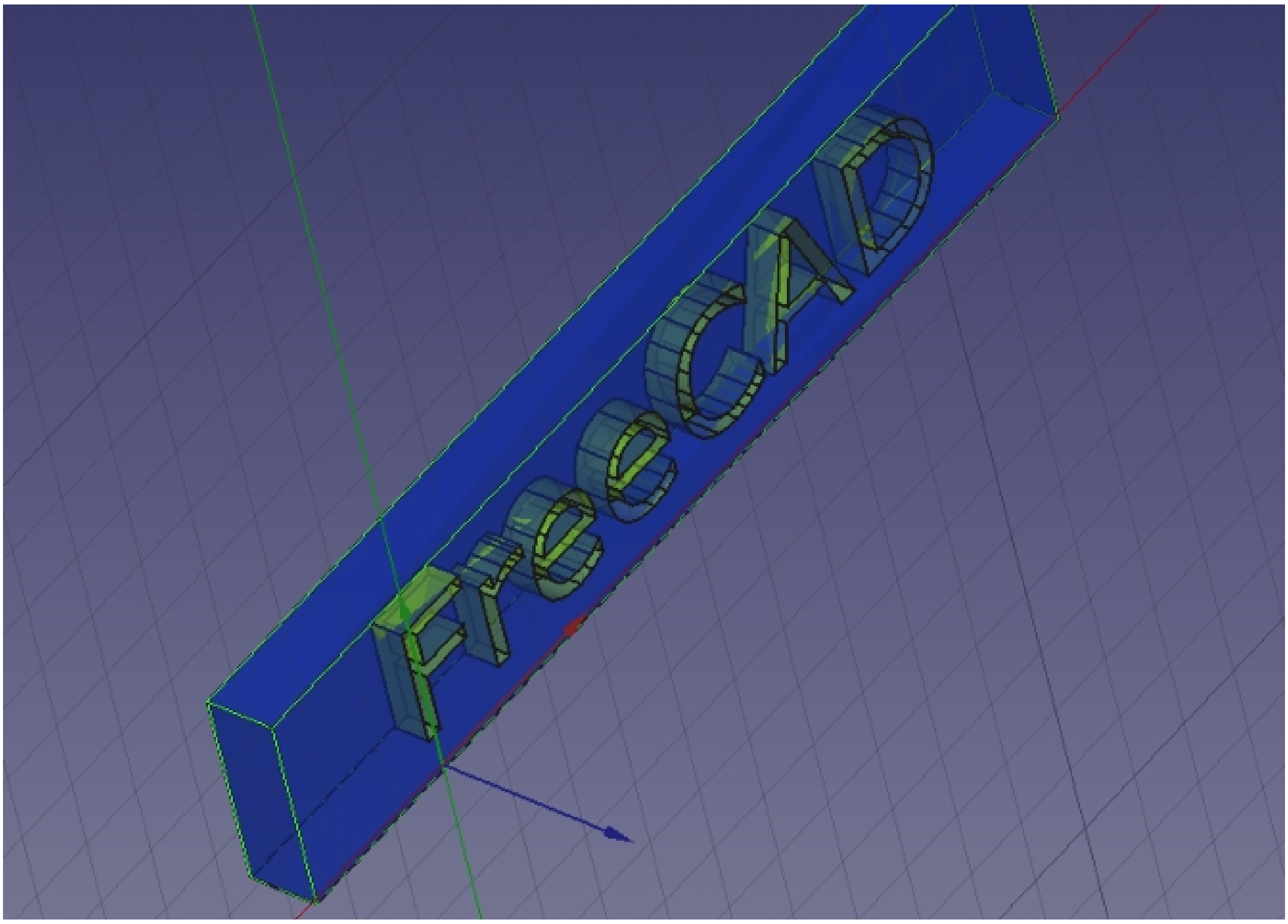
Go back to your curvy tray project and remove the cylinder projection.
(Re)create your text in this project the same way you just did.
You can now use the text in the same way as you used your small logo or part in the earlier projection example. You can see that you have again projected the text onto the curved side of your curvy tray example.



You may have noticed when you set up a projection, as well as a height input box, there is also a ‘Depth’ box. Of course, the depth option allows you to project through the selected surface to a set depth. To see how this works, a useful experiment is to project into a part that is partially transparent.
In a new project, create some example text using a ShapeString.
Now return to the Part Workbench. Click the ‘Create a cube solid’ to create a cube. Resize the cube so that it is larger than your text ShapeString. Then right-click and select Transform. Move the cube item down underneath the ShapeString, ready for projection.
Click the Projection tool and select the surface of the cube item you just made. Then select the faces of the ShapeString in the usual way. Add 2 mm of height, but add 5 mm of depth to the projection, and then project onto the cube item. Select the cube item in the file tree, right-click, and select Appearance. In the appearance dialog, slide the ‘Transparency’ slider to the right to set the transparency to around 40%.
Closing the Appearance dialog, you should now be able to see that the cube is partially transparent, and you can see the projection is inside the cube to a depth of 5 mm. Finally, you can select the cube and then the projection object, and click the ‘Make a cut of two shapes’ tool to remove the projection from the cube object.



Now go back and delete the cut as well as the projection. Recreate the projection but now with a height of 5 mm and a depth of 5 mm. Make the cut again as well. Set also transparency on the cut to see the cut-out of the text. This is a useful effect if you are trying to make a text-based sign object for 3D printing



The surface tools and projection approaches can create very complex effects and designs, but you mentioned earlier that they have their limits. How, for example, could you create a pattern or text that flows completely around a cylinder or other curved shapes?

Surface Workbench

Using the ruled surface tool on the Part Workbench is useful. In some ways, it is the first step towards thinking about 3D modelling in terms of edges and surfaces – rather than creating a 3D object via extrusion or padding, and then cutting and adding parts to and from it. Surfaces in FreeCAD are given their own Workbench, and it's certainly worth exploring in its own right. The ruled surface tool you use in this article has some limitations – one main one being that, if you have a closed path like the base of your curvy tray, the ruled surface tool can't be used to create a surface filling the path. This is why, in the article, you create a small extrusion from the original sketch. If you are interested, create a sketch similar to the first B-spline sketch you created in the main tutorial, and then move to the Surface Workbench. Select the first tool icon, 'Creates a surface from a series of picked boundary edges', and in the dialog, click 'Add edge', and then select the sketch edge in the preview window. You should see a surface created filling the sketch area.You hope to revisit the Surface Workbench and other surface tools in future articles, as it has a plethora of tools that enable complex operations in some ways similar to the lofting and sweeping operations you looked at in issue 40, but it's even more versatile.

Vector Graphics

For your example of projecting a part onto a curved surface, you have used cylinder. However you can also import as an SVG and then extruded into a part. The SVG file is a single path outline.

The ability to import an SVG is a really useful feature occasionally, and works well. Simply go to File > Import and navigate to your SVG. When selected, you get a dialog asking if you want to import as a 'Drawing' or 'SVG as Geometry'. You need to select the latter, and then the file is imported and appears in the file tree as an object labelled 'Path'. On the Part Workbench, if you select the imported SVG, click the Extrude tool. You can extrude in the usual way.

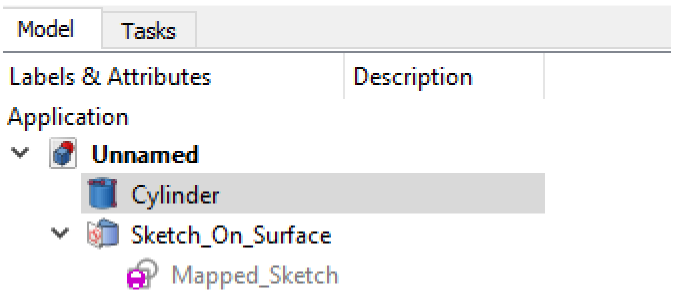
There are also ways to import an SVG and convert the SVG into a sketch so that you can constrain its contents, making it usable in other Workbenches. A useful video tutorial on this, is available here: https://www.youtube.com/watch?v=6LedIN5S2so.

Design your own ring

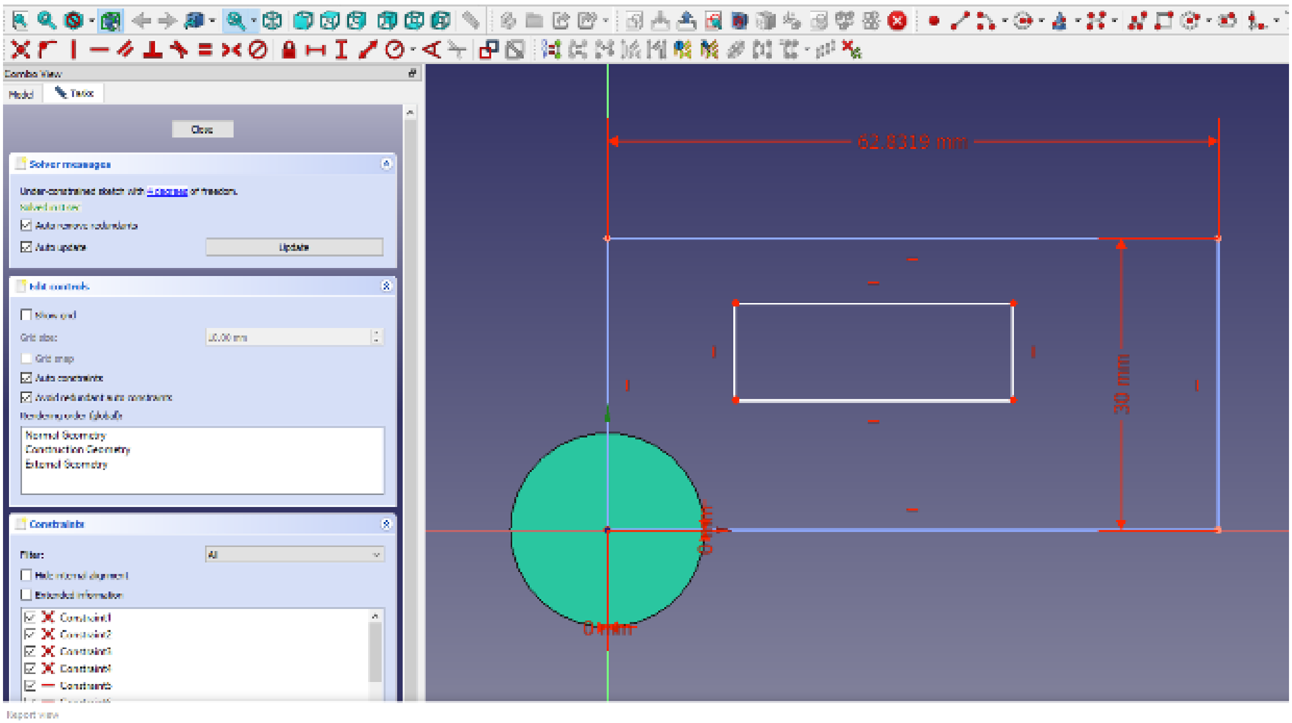
You looked at projection as a tool to add geometries or objects onto curved surfaces. In this chapter, we'll discuss the limitations of projection and look at how you can map sketches onto the surfaces of curved objects.
Along the way, you will make a geeky maker ring to 3D-print, and you will also explore some more complex uses of the Curves Workbench.
You projected objects onto curved surfaces, and this works well in many instances. It has some limitations, though. The major one is that by design, projection works with a fixed projection view and as such, it can't be used to wrap a design around another design. Let's start by looking at how you can wrap a sketch around a cylindrical object.

Open FreeCAD and use the Addon Manager in 'Tools'-list to add the "Curves" Workbench. When the list populates, scroll to the "Curves" Workbench and click to install. Once downloaded and installed, you will be prompted to restart FreeCAD. Once you have restarted FreeCAD, open a new project.
For your first example, let's start in the Part Workbench. Click the 'Create a cylinder' tool icon and then highlight the 'cylinder' object in the file tree.
In its dialog box, change the radius to 10 mm and the height to 30 mm. Next, let's switch over to the newly installed Curves Workbench. In the preview window, select the surface that forms the outer wall of your cylinder, and then click the 'Map a sketch on a surface' tool icon.

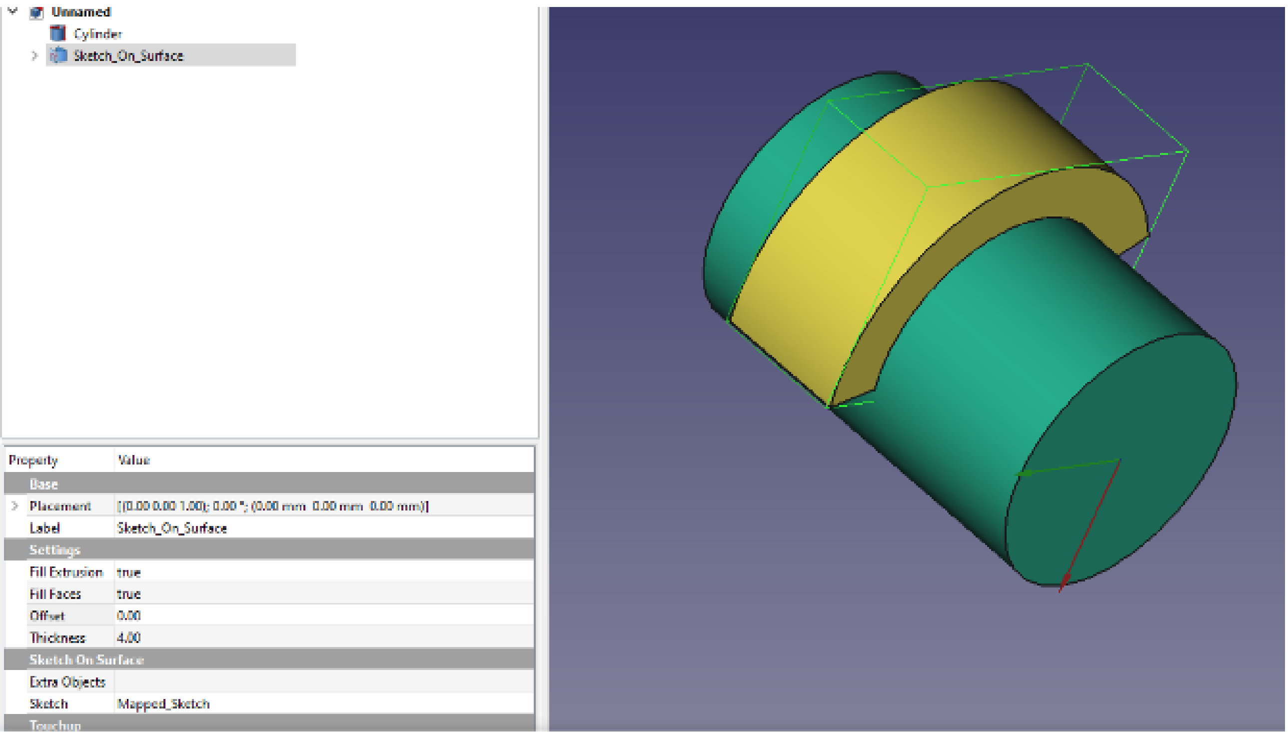
You'll notice in the file tree view that a new 'SketchOnSurface' object appears and that this is a drop-down menu item. If you click it to open the drop-down menu, it should contain only one item: mapped sketch.



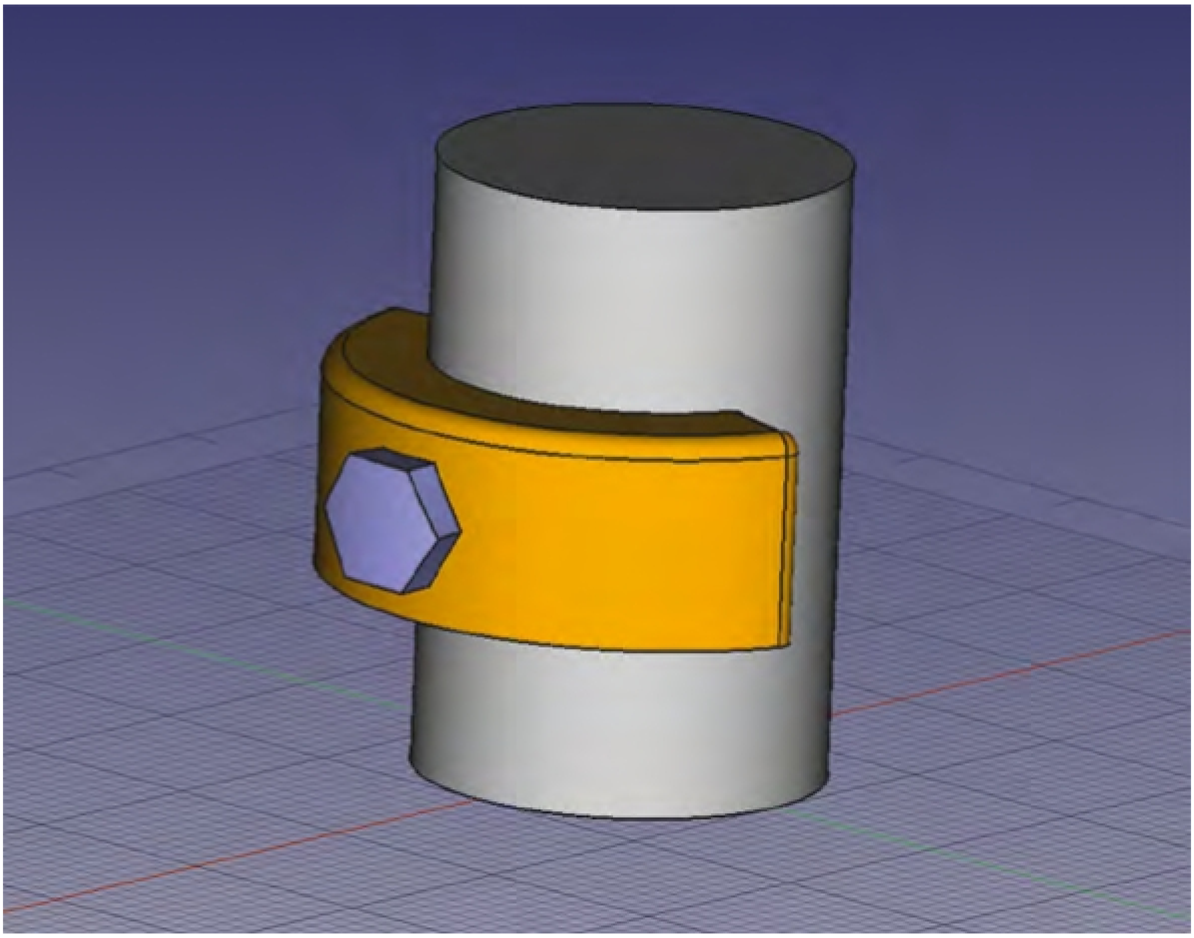
If you double-click on this item, it will open the Sketcher Workbench. In the sketch, there will be a blue rectangle. When geometry lines in sketches in the Sketcher Workbench are blue, it means that they are construction geometry that acts as a guide but doesn't appear in the sketch as part of the object. They have many uses, but in this case, the 'Map a sketch on a surface' tool has created a construction geometry that matches the dimensions of the cylinder's face and acts as a containing area for your mapped sketch. For a first simple example, let's draw a rectangle somewhere inside the blue construction rectangle.



Having drawn a rectangle inside the construction geometry, close the sketch and you should automatically return to the Curves Workbench. You should now see the rectangle you drew on the surface of the cylinder in relative position when the blue construction geometry is wrapped around the cylinder. If you don't see your mapped rectangle, rotate the cylinder as it may be positioned out of view. You can now close the drop-down mapped sketch on the file tree and when you select the parent SketchOnSurface item, the small mapped rectangle should be green and selected in the preview window. Selecting this, you can perform some operations on this sketch like any other. With the sketch highlighted in the dialog, set the 'Fill faces' option to 'true' and then increase the 'thickness' a few millimetres. You'll see the sketch is extruded out from the face of your cylinder.

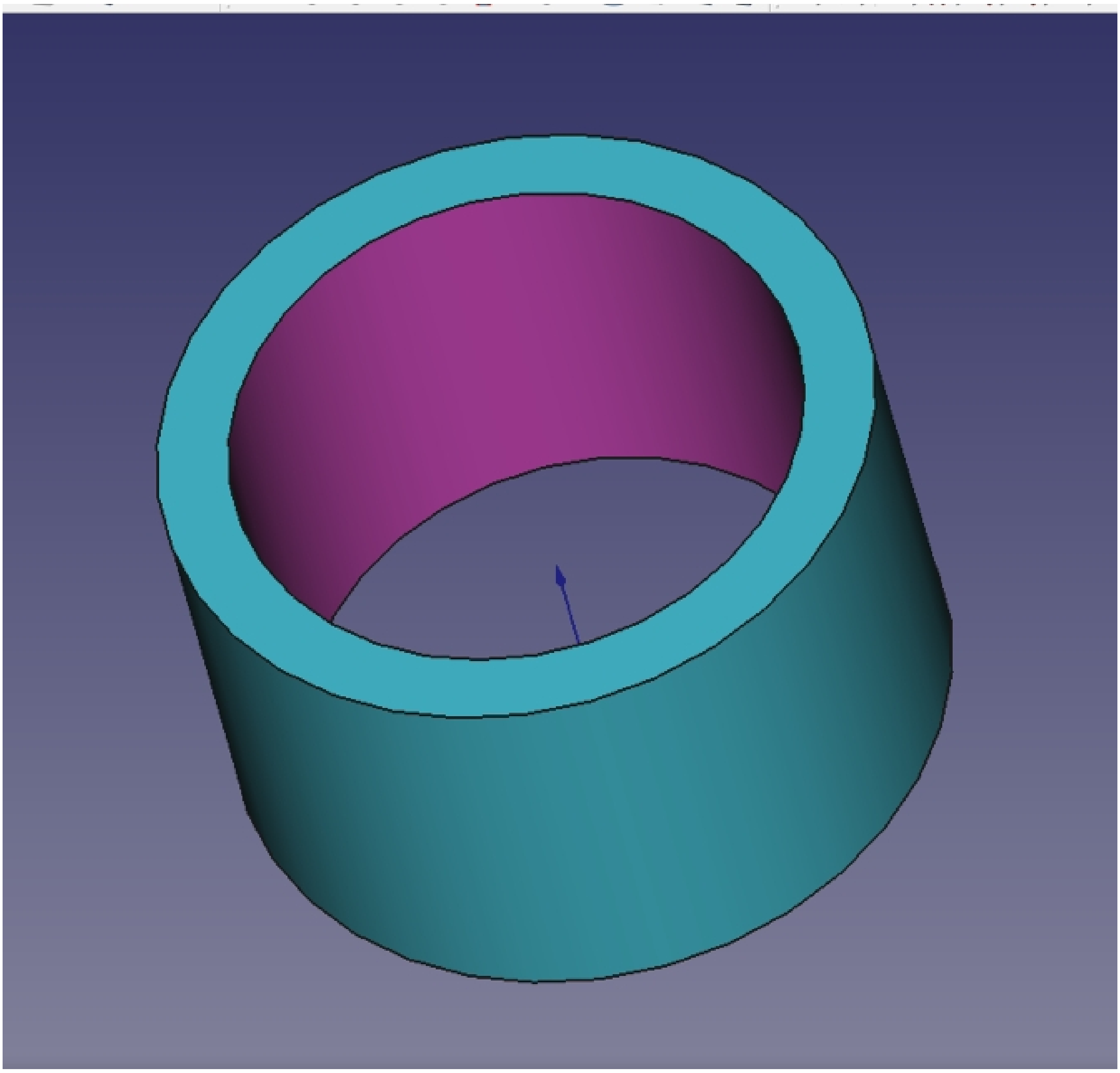


It’s perhaps worth pointing out that the resulting objects made using mapped sketches work in the same way as other objects with regard to the Part Workbench and other tools. You can add some fillets to the edges of the part and use the appearance menu to set the object’s colour. You can also go one step further in that you have clicked the surface of your mapped object and created another SketchOnSurface object to create the small extruded hexagon.

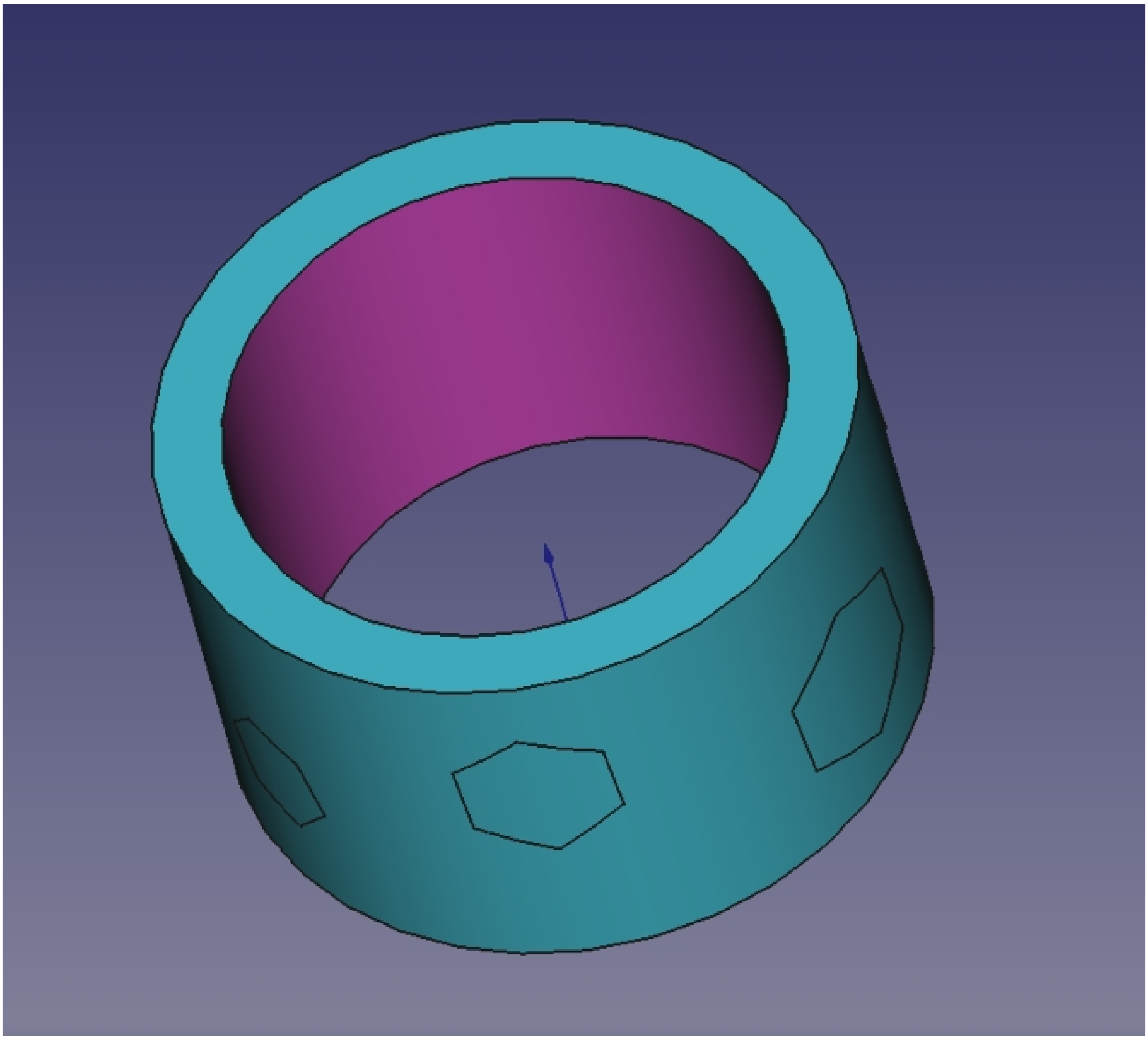


As well as adding more mapped sketches to build up complex geometries, you can use Part Workbench tools such as the Boolean operation type tools to make cut-out parts using the mapped sketch. As an example, you can make a fun mini project to make and 3D-print.

Let’s make a useful (?) 3D-printed ring. Start in a new project and once again with the Part Workbench. Create a cylinder that is just a fraction larger than the diameter of one of your fingers and around 12 mm tall.
Create another cylinder that is slightly larger in diameter but the same height – the difference between the first and second cylinder diameter will be the thickness of the ring. If you don’t move either cylinder, they should be aligned and concentric around the origin point. Select the second larger cylinder first and then select the first cylinder and click the ’Make a cut of two shapes’ tool icon from the Boolean tools selection. You should now have a basic ring shape.

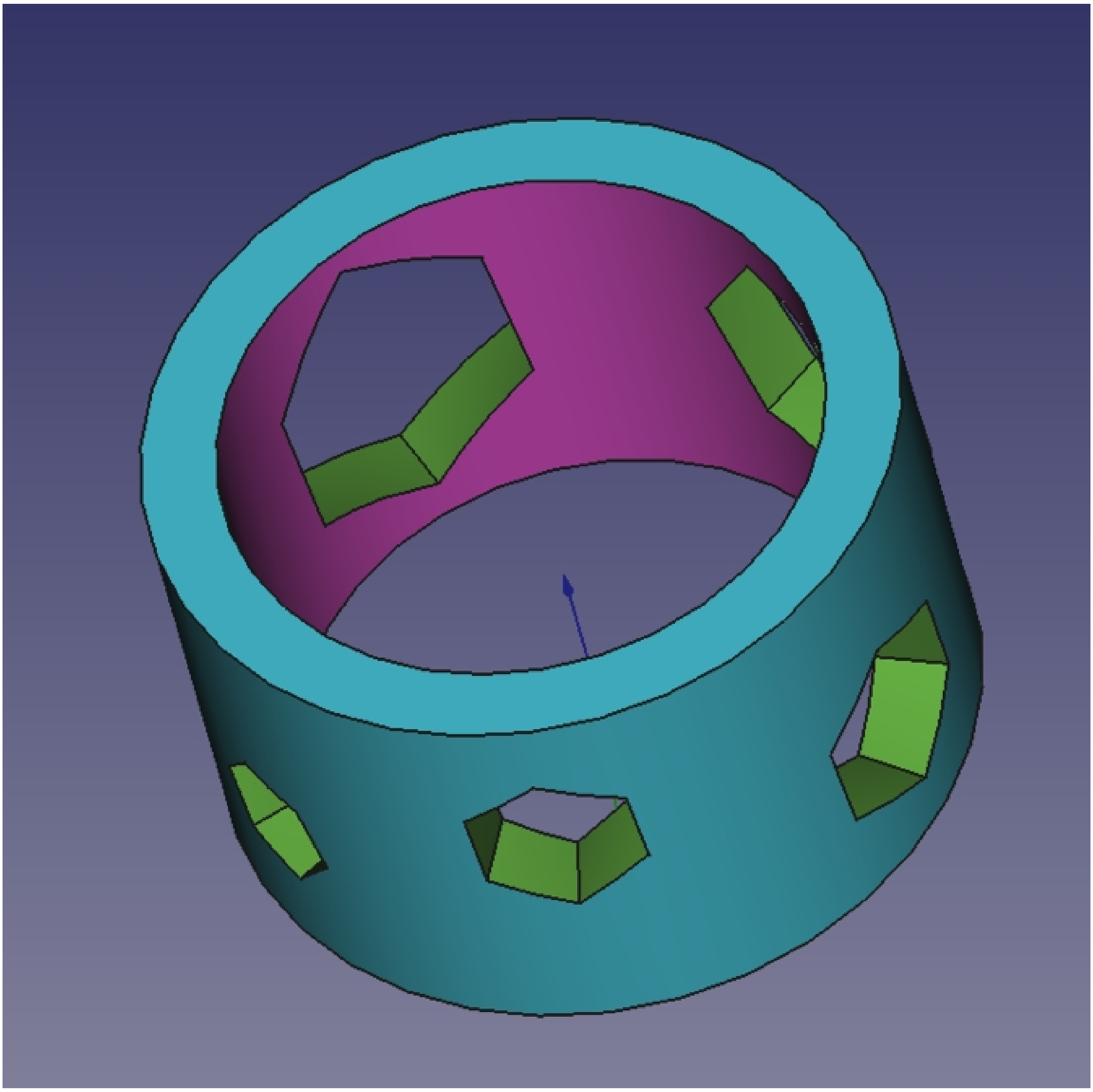


Moving to the Curves Workbench, you will map a sketch to the outside surface of the ring as you did earlier with your cylinder. Click the outer surface and then click the 'Map a sketch on a surface' tool icon once again. Click to drop-down and double-click the mapped sketch, which will open in the Sketcher Workbench.
The idea for this maker ring is that you will cut out some hexagonal shapes that not only look decorative but will serve as an emergency small spanner tool. As such, inside your construction geometry, you added some hexagons sized to common small metric nut sizes. Go for M2, M2.5, M3, M4, and M5. You don't need to fully constrain the sketch, but you can opt to align one line of the hexagon with a vertical constraint so that you could then constrain the nut size by selecting two opposite points and applying a horizontal dimension. You added roughly 0.1 mm to the M-series given nut sizes to make them an easy fit – you can see the sizes you used in the constraint values.



If you preferred your hexagon spanners positioned randomly, you could constrain them to be on the centre line and more equally spaced if preferred.
Once you are happy with your hexagons, close the sketch and you should see the mapped sketch around your ring. Select the SketchOnSurface object and, in the dialog, set 'fill faces' to 'true' – but this time, add a negative value equal to the thickness of your ring design in the 'thickness' input. Your ring is 2 mm thick, so you went with -2 mm. This should now have extruded the mapped sketch through the ring.

Now that you have all the objects created, it's a case of moving to the Part Workbench and selecting the ring first, then the SketchOnSurface object, and then creating a cut of the two parts to create your ring.

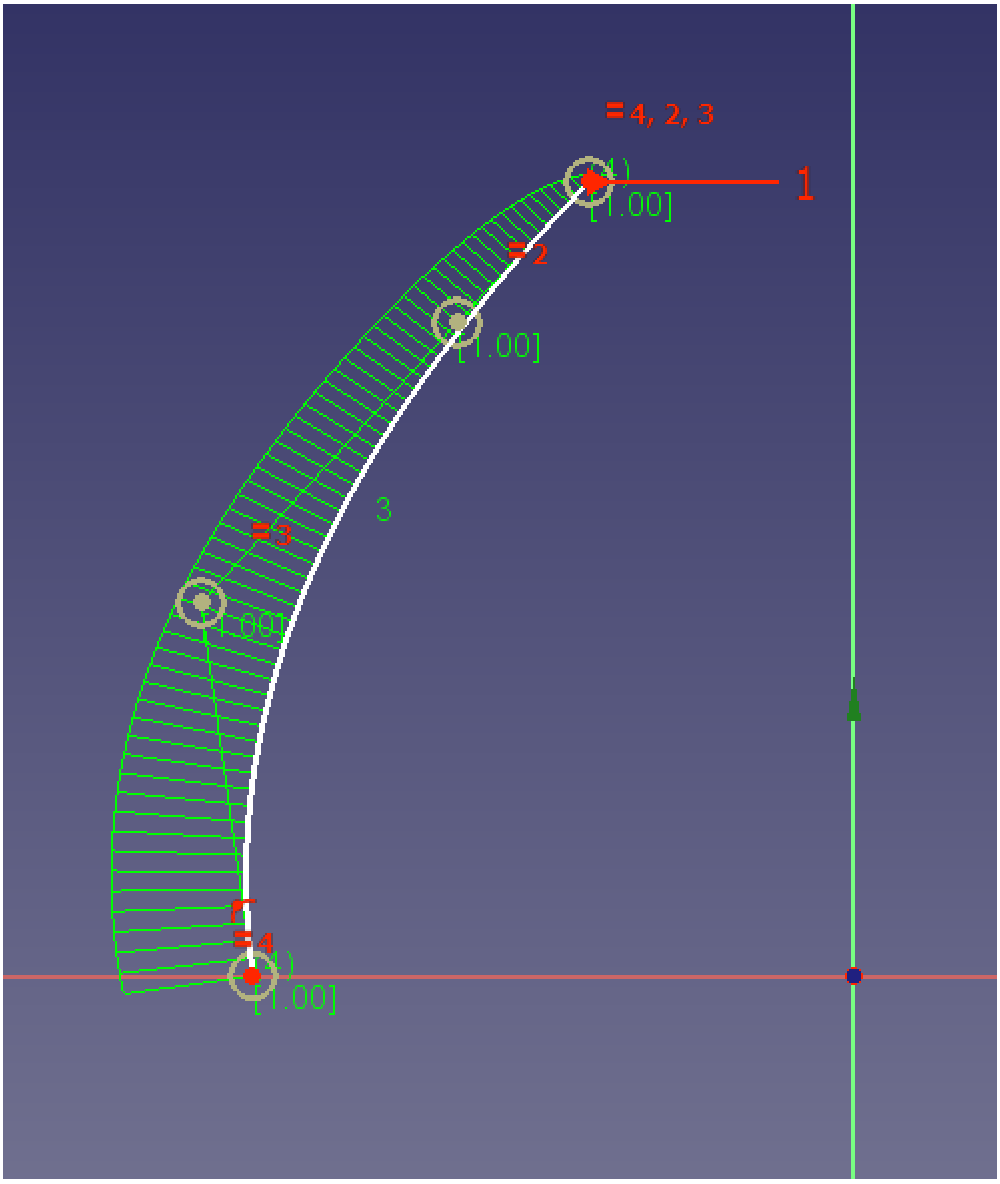


You can then use File > Export to save your ring as an STL file suitable for slicing and printing.

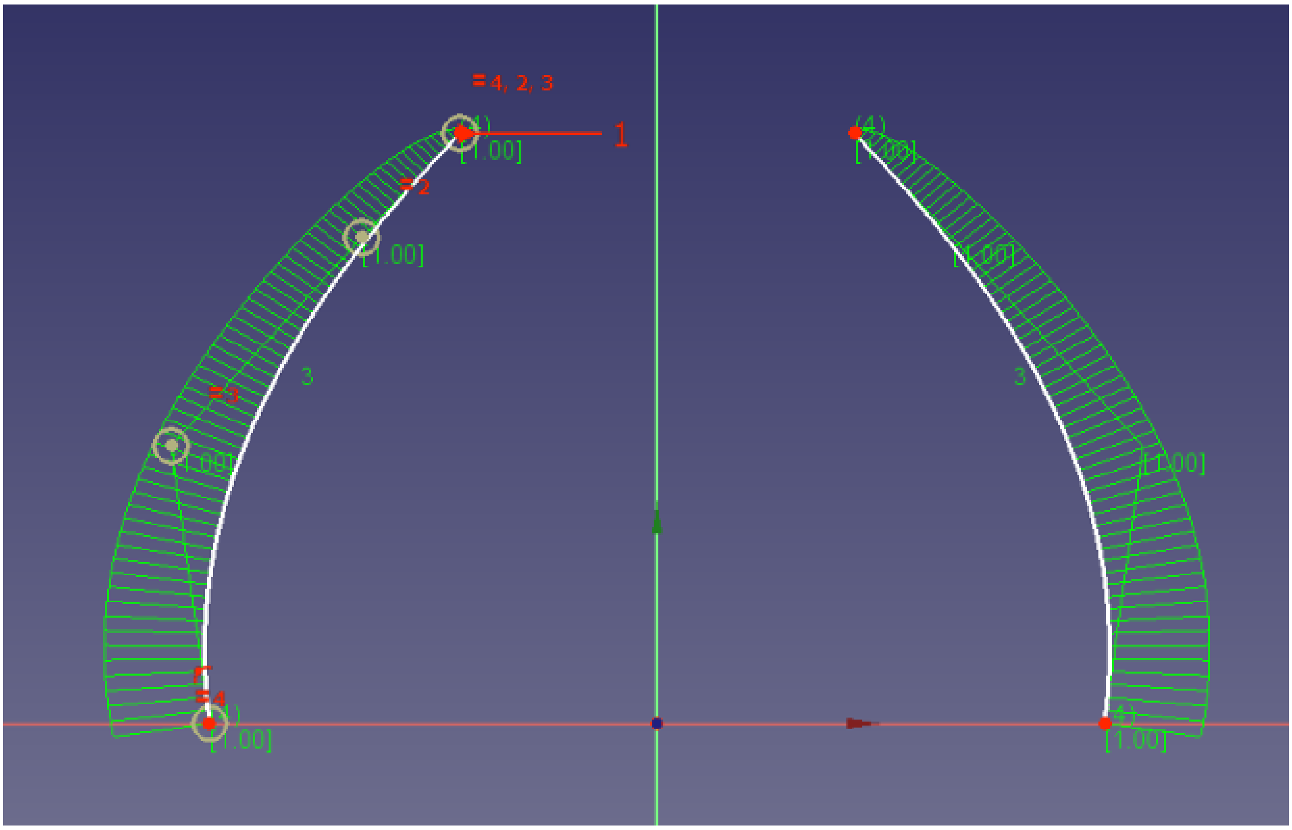
Using the sketch mapping tools on the Curves Workbench is just one example of what can be done with this workbench. Almost every tool on the Curves Workbench could have a tutorial written about it, but let's look at a couple of tools to make a complex curved surface that curves in all axes.

You are aiming to make an object that looks like an aeroplane cockpit canopy cover as it's a challenging shape to model. Again, you are not particular about the dimensions of the object and, as such, you are not worried about constraining aspects of the design.

To begin, create a new project, move to the Part Design Workbench and create a body. Create a sketch in the XY plane. In this sketch, let's create a B-spline. This B-spline is going to be the lower side edge profile of your canopy cover. Click the B-spline tool and start the B-spline attached to the X axis to the left of the origin point. Add some points and then finish the B-spline to make it look similar to the this.

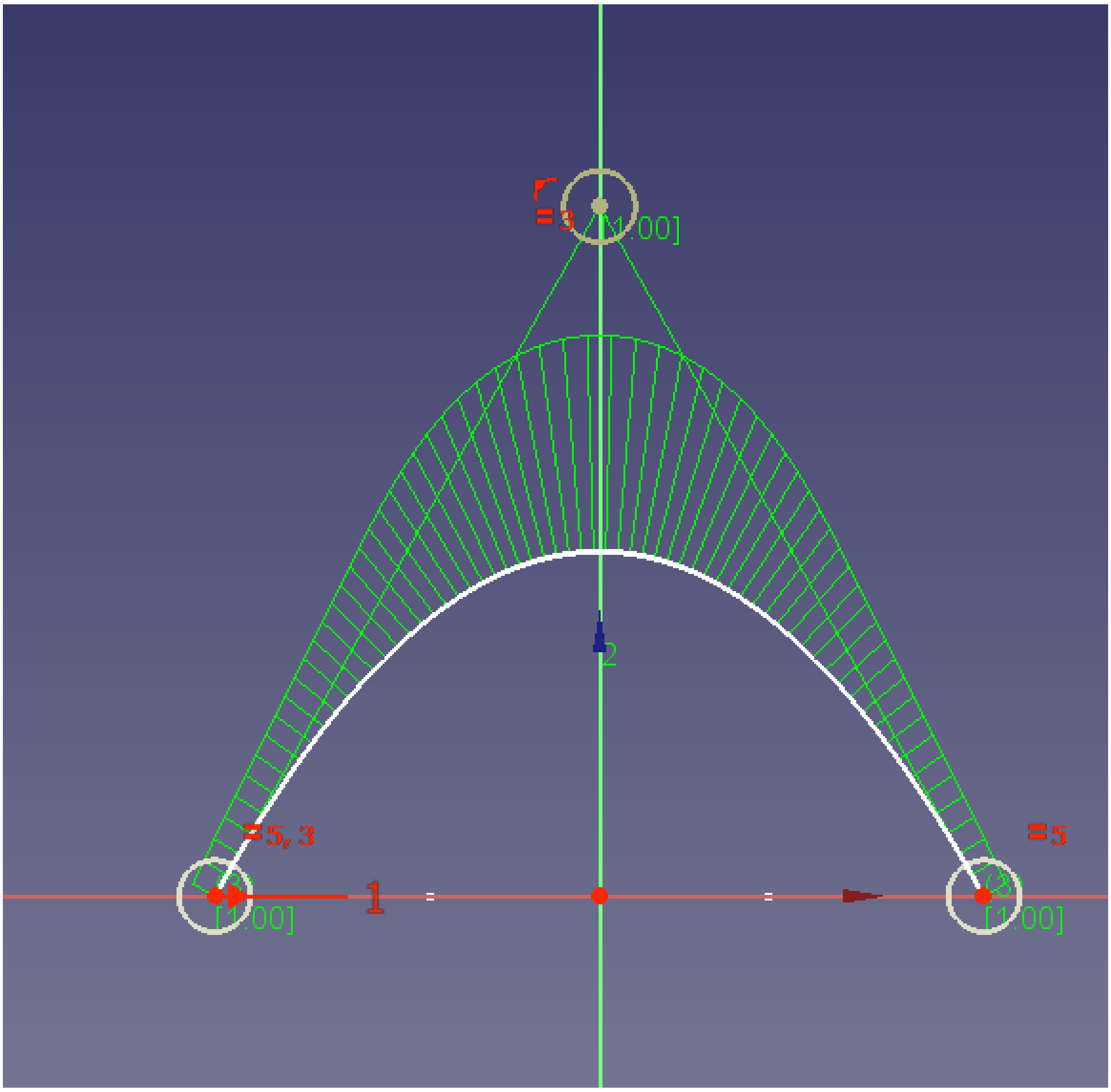


Before you close the sketch, select the B-spline that you just made and click the 'Create a symmetric geometry with respect to the last selected line or point' tool to add a mirrored B-spline on the right-hand side of the Y axis line.



Create another sketch in the active body, but this time in the XZ plane. You need to import the endpoints of the mirrored B-splines that sit on the XY plane (the starting end of the B-splines in the previous sketch). In the new XZ plane sketch, click the 'Create an edge linked to an external geometry' tool and zoom in to select the endpoints of the two B-splines from the first sketch. Now that you have those points available, draw another B-spline that begins with the point constrained to the left-hand B-spline from the first sketch. Place the second B-spline point on the Z axis line above the origin and attach the final B-spline point to the imported edge of the mirrored B-spline.

When you finish this B-spline, you should have a symmetrical curve that will form the rear edge of your canopy design.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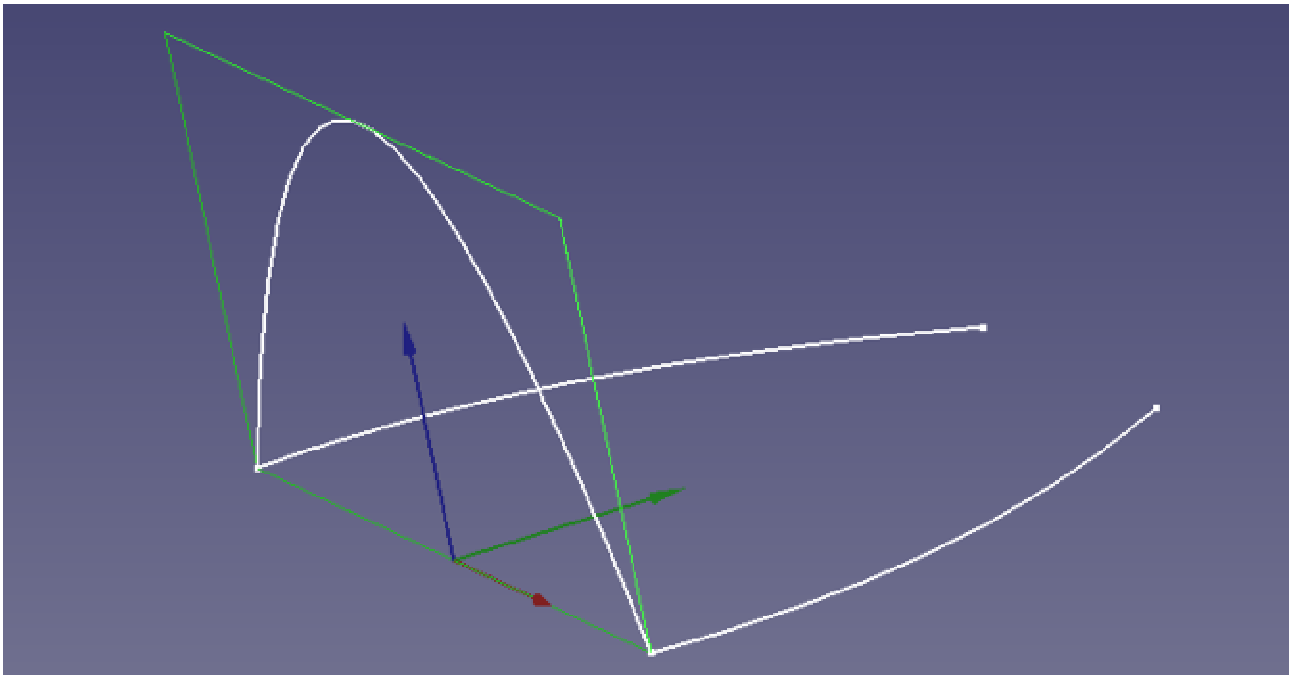
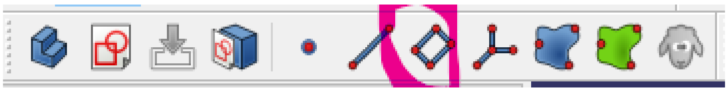
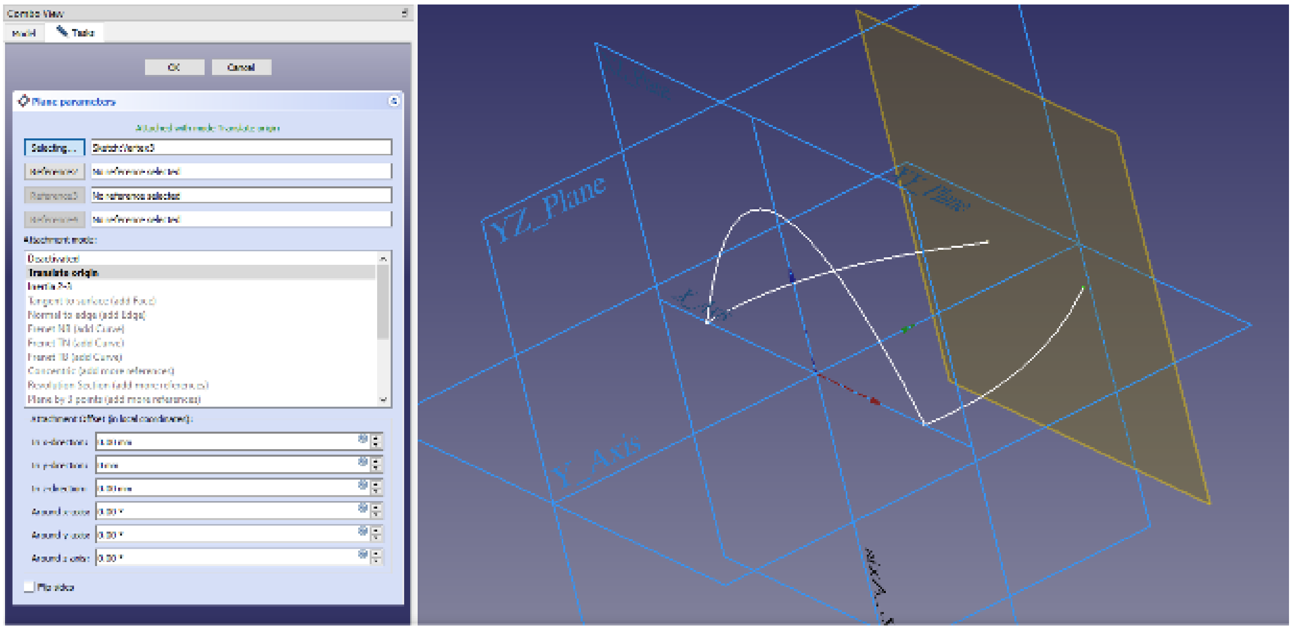


You now need to add a similar smaller B-spline that will connect the front ends of the original mirrored B-splines to form the front edge of your canopy design. Of course, these points don't sit on a datum plane, so you need to add a datum plane to work on first. Creating datum planes is an incredibly useful tool in many projects. To help us set up your new datum plane, let's make the three standard plane origins visible. To do this in the file tree, highlight 'Origin' and press the SPACE bar. You should now see the blue lines and labels for the XY, XZ, and YZ planes. Click the 'Create a new datum plane' tool icon on the Part Design Workbench.



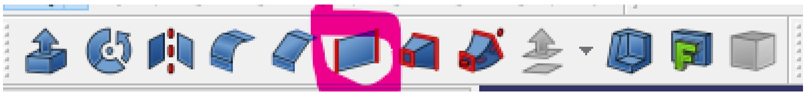
You should see a 'Plane Parameters' dialog box appear in the Combo view area. At the top of the dialog, you will see four boxes with 'No reference selected' written in each box. To the left of the first box, the button text should read 'Selecting' – if for any reason it doesn't, click on that box until it does. If you imagine your desired new datum plane by visualising where you want to sketch, you could describe the plane as a sort of XZ plane moved to the far end of your original mirrored B-splines. This kind of hints as to how you will create the new datum plane. First, let's click on the XZ plane that you made visible in the preview window. You should now see that the new brown-coloured datum plane moves onto the XZ plane. Double-check the dialog. You should find that the first box now has 'XZ_plane' in the input, and the button now says 'Plane'. Click the button until it returns to 'Selecting'.
Finally, you need to move and zoom to select the endpoint of one of the mirrored B-splines you made in the original sketch. This should make that vertex point the reference of your new datum plane, which stays oriented as parallel to the XZ plane. Once it's in place, you can click OK to close the datum plane dialog.

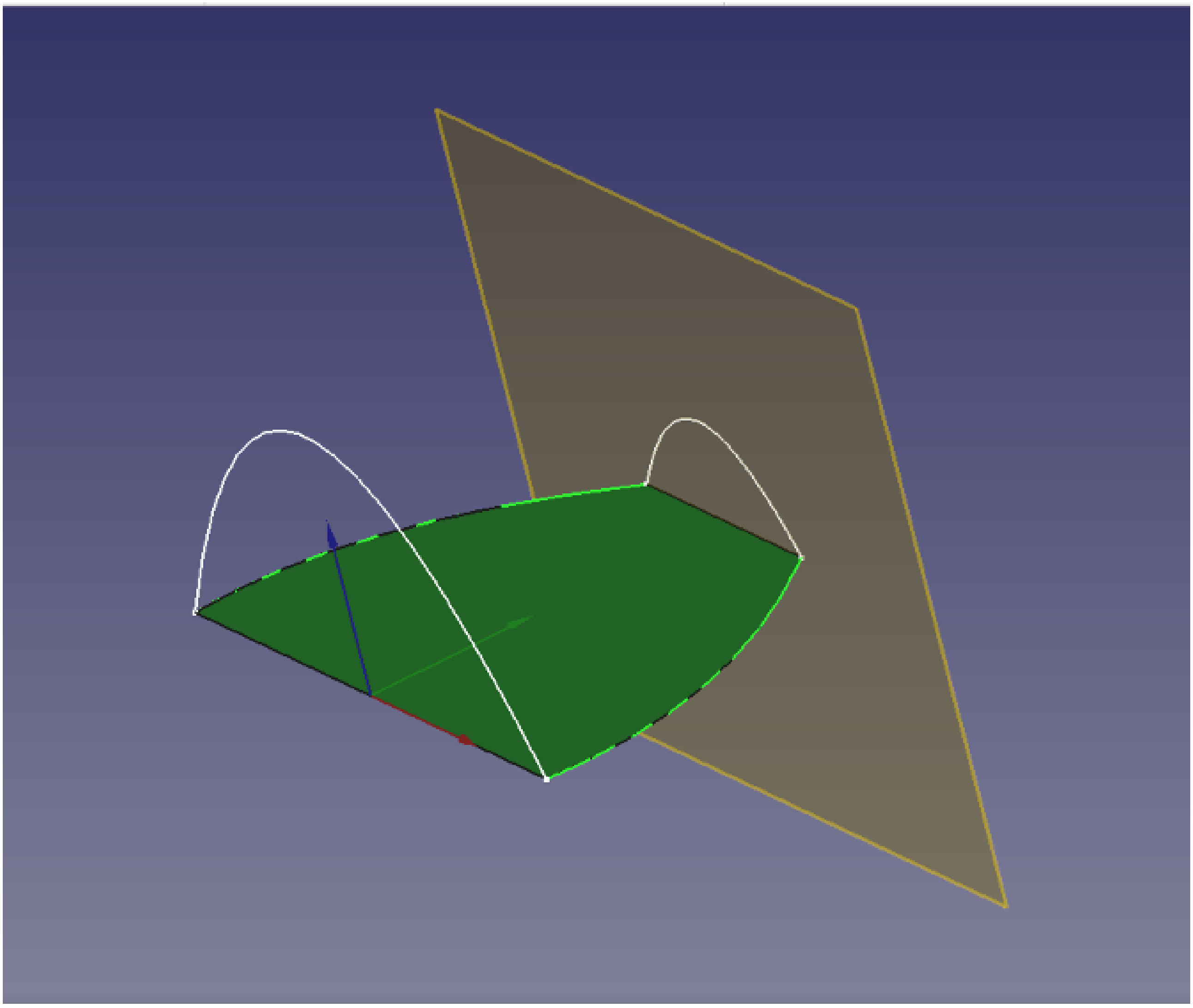


With the new datum plane highlighted in the file tree, click the 'Create a new sketch' tool to open a new sketch. Import the B-spline endpoints to the sketch and make a smaller B-spline to form the front of the canopy in the same way that you did the sketch for the canopy rear on the XY plane.

To create a surface for your canopy, first you are going to make a flat ruled surface between the original sketch lines and then you are going to use the 'Sweep profiles on 2 rails' tool and the 'Approximate points to NURBS curve or surface' tool.

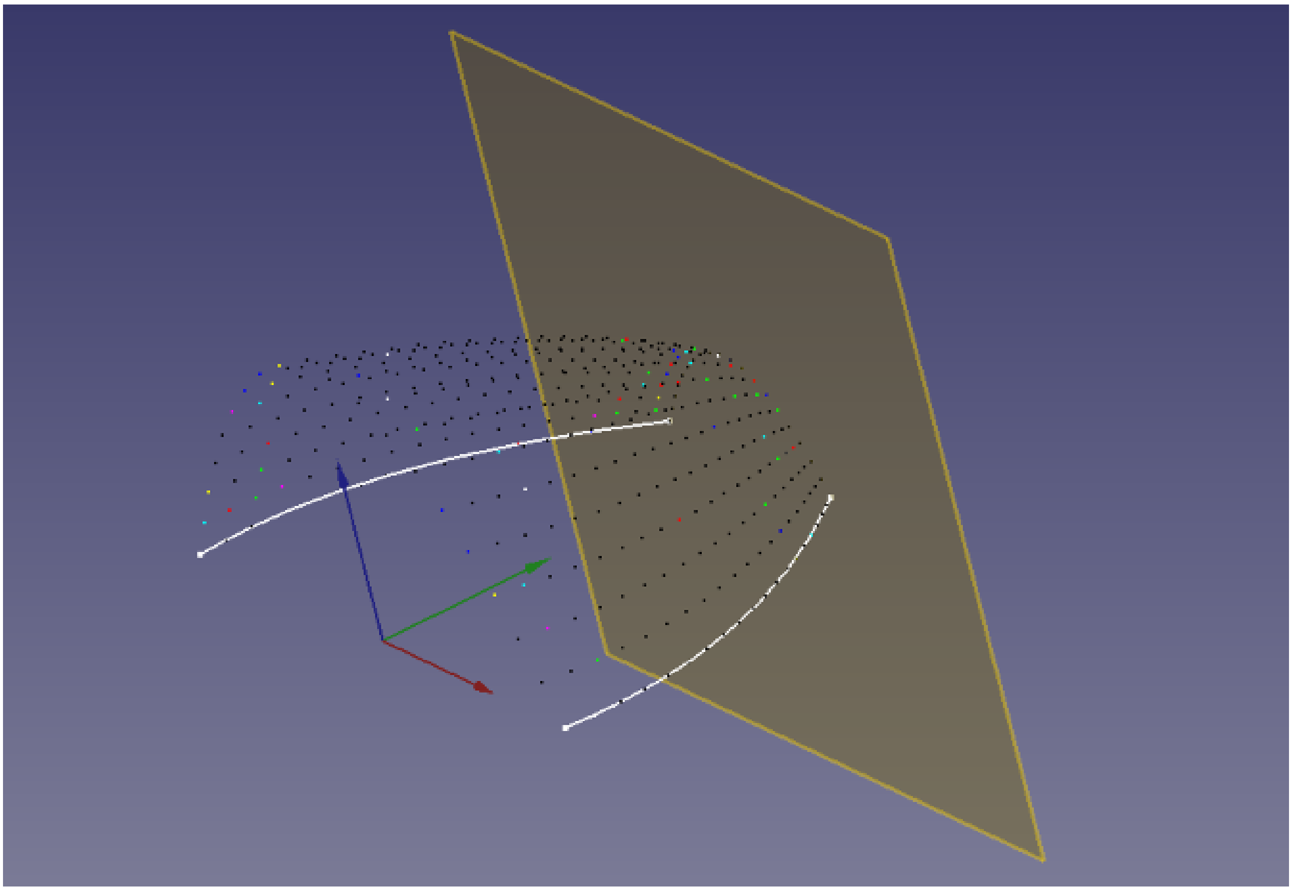
To create the ruled surface, move to the Part Workbench, select the two B-splines of the original sketch, and click the 'Create a ruled surface' tool.



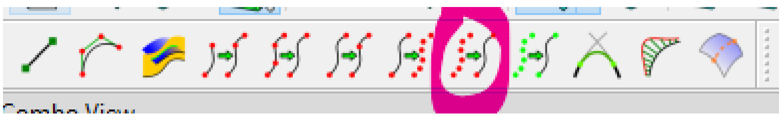


Moving to the Curves Workbench, in the file tree view, select the new ruled surface object and select the two sketches that create the rear and front edge shape of your canopy design. With these three objects highlighted, click the 'Sweep profiles on 2 rails' tool.

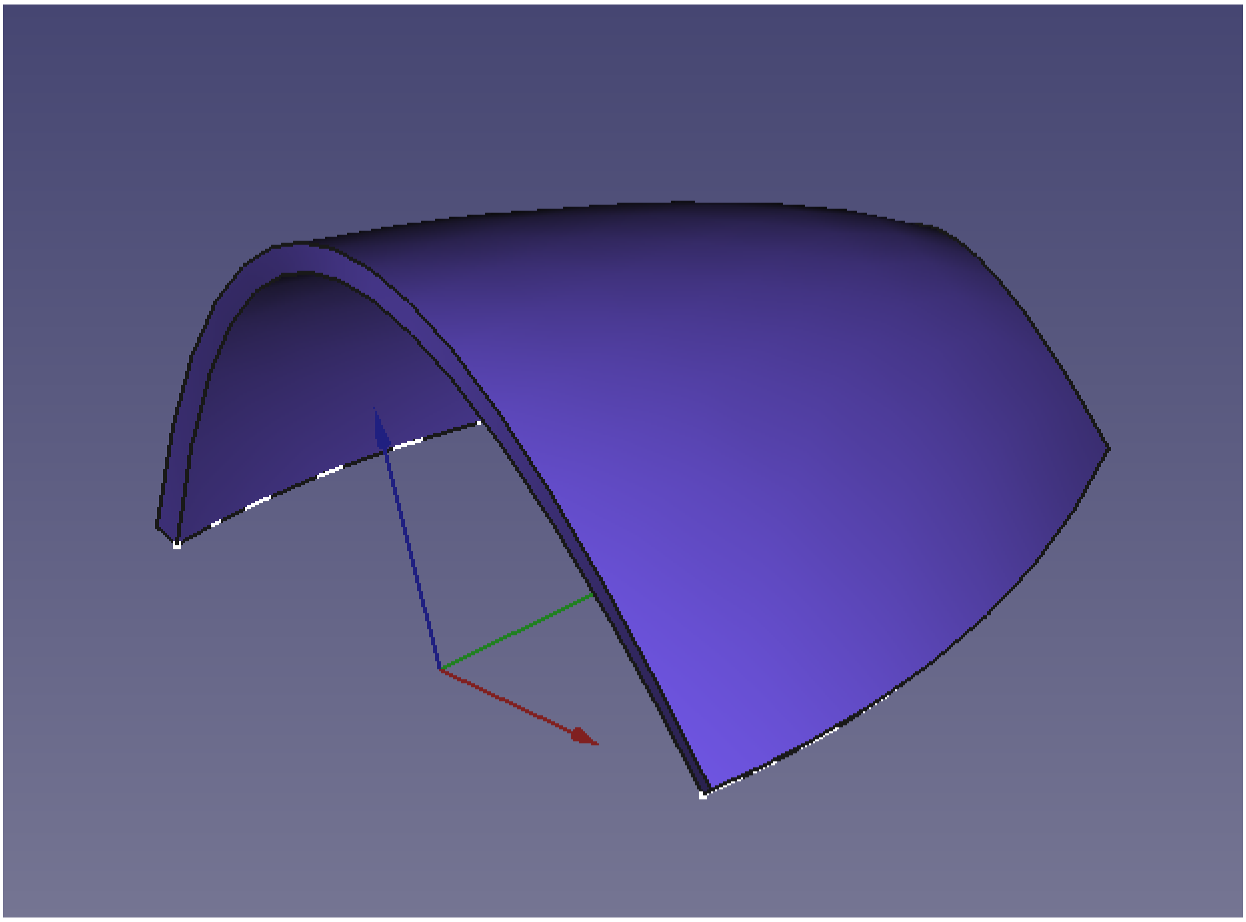
Your computer may take a little time to process this, but once it’s complete, you should have a collection of points, or a ’point cloud’ in the preview window that is beginning to look like the shape of your aeroplane canopy.



There are plenty of options to increase and decrease the resolution of the point cloud, but for this example the default settings are adequate.
To convert the point cloud into a surface, make sure that the Sweep2Rails object in the file tree view is highlighted, and then click the ’Approximate points to NURBS curve or surface’ tool.



You should now have a perfect canopy surface. Of course, the surface created is similar to your earlier ruled surface in that it has no thickness dimension – you can, however, use familiar tools to create a thicker solid object. With the ’Approximation Curve’ object highlighted in the file tree, you can move to the Part Workbench and click the ’Offset’ tool. In the offset tool dialog, you can create an offset and check the ’fill offset’ box to create a thicker canopy.

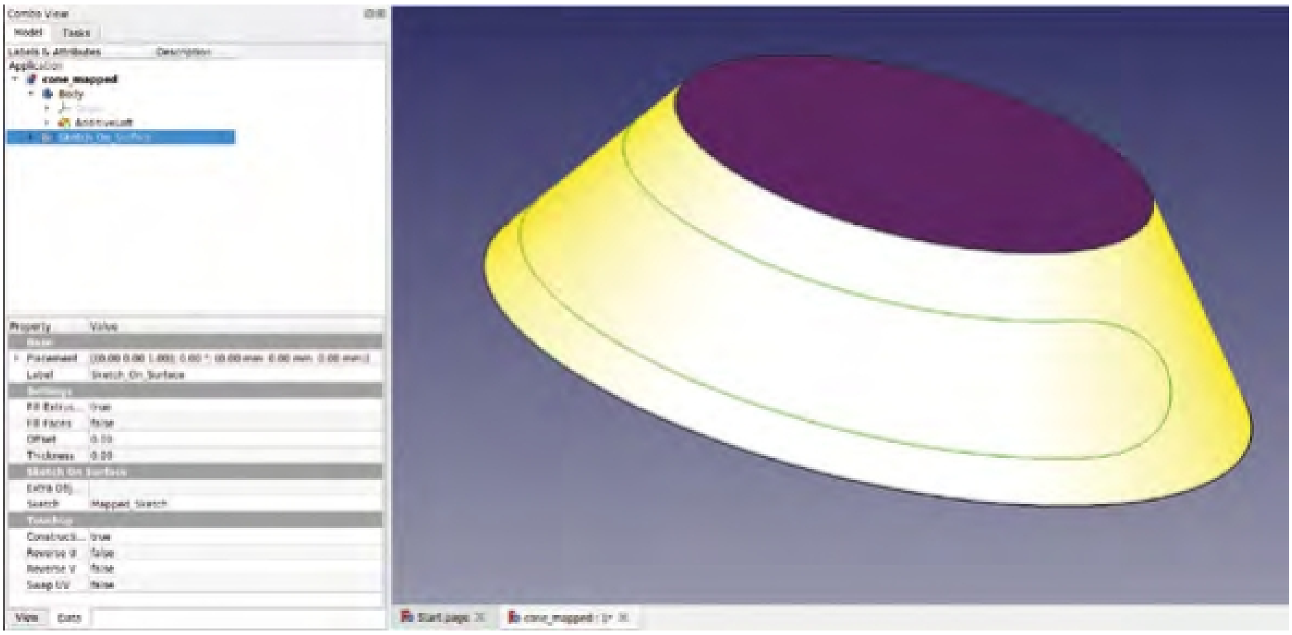


We’ve seen a small range of tools on the Curves Workbench in this tutorial, but we’ve only scratched the surface of what this workbench is capable of. As ever, there is a heap of knowledge on the FreeCAD community forum as well as lots of videos and tutorials online to help you explore the rest.

Curved Cone
If you want to map a sketch onto a cone shape and you want to work in Part Design, the 'Map a sketch to a surface' tool works a little differently. You have to draw your own construction geometry and do things in a slightly different order. As an example, in a new project, move to the Part Design Workbench. Create a new body and create a first sketch in the XY plane. Create a circle roughly 100 mm in diameter around the origin point. Close this sketch. Create a second sketch in the XY plane and draw a smaller circle for the top of your cone around the origin point – again, don't worry about constraining these sketches too much. Closing the second sketch to return to Part Design, highlight the second sketch and in the dialog, under the 'Attachment' submenu, go to 'Position' and raise the Z axis value – you went with 25 mm, to move the sketch upwards.

Select both the sketches in the file tree and then click the yellow and red 'Loft a selected profile through other profile sections' tool icon. Click OK and you should have a cone. Moving to the Curves Workbench, select the curved outer face of the cone and then click the 'Map a selected sketch' button. Different to earlier examples, you will see that a mapped sketch object appears without the SketchOnSurface object. Double-click the mapped sketch object and you will move to the Sketcher Workbench, but there won't be any construction geometry.
Toggle the 'Toggles the toolbar or selected geometry to/from construction mode' button to turn the sketching tools blue, and then draw a construction geometry rectangle from the origin point. You can work out the exact shape and size of the curved cone surface limits, but for this example you just drew a rectangle that would fit on the cone surface with room to spare.

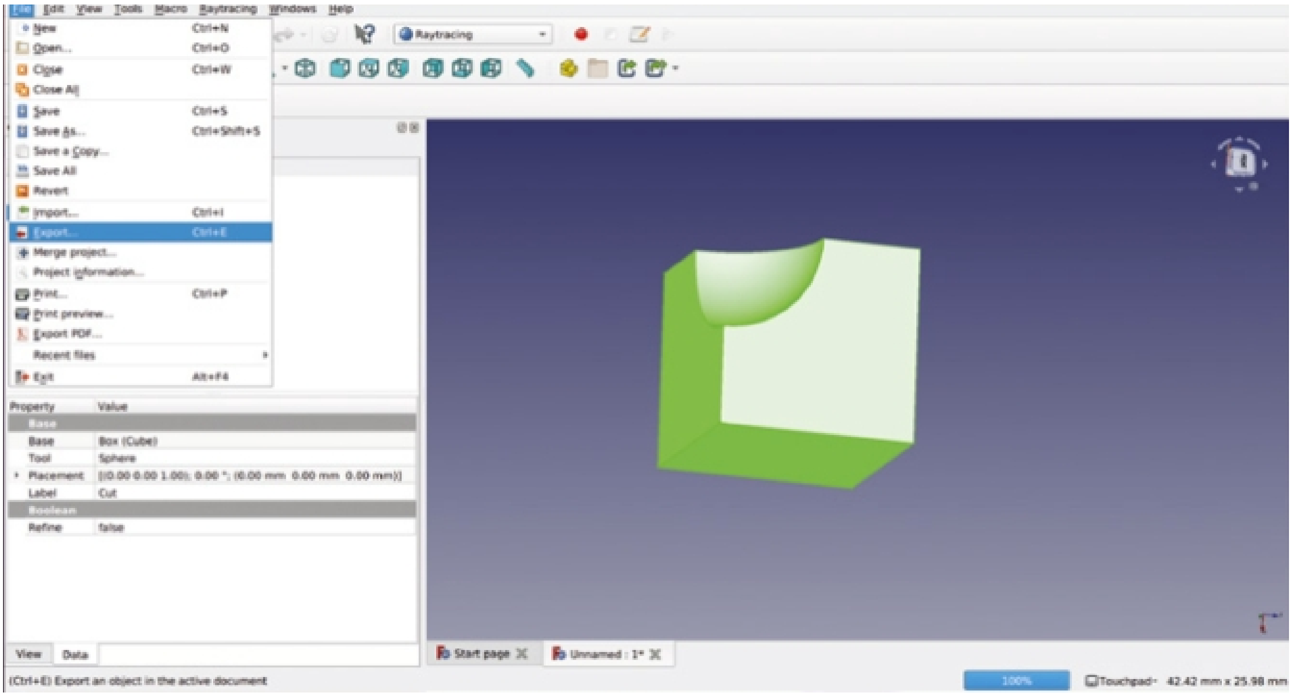
Toggle the Sketcher tools back so that they no longer draw geometry objects, and draw an object inside the rectangle. You went for a 'slot' object. Close the sketch to return to the Curves Workbench. Finally, select both the 'additive loft' object that is your cone and the mapped sketch in the file tree and then click the 'Map a selected sketch' button again. It should now create the SketchOnSurface object and map your slot onto the cone's surface. Similar to earlier examples, you can add thicknesses and perform cut operations to create your design.



Quick Tips
Often in mapped sketches objects cover the construction geometry. Click the 'When in edit mode switch between section view and full view' tool to solve this.

Getting Meshy in FreeCAD

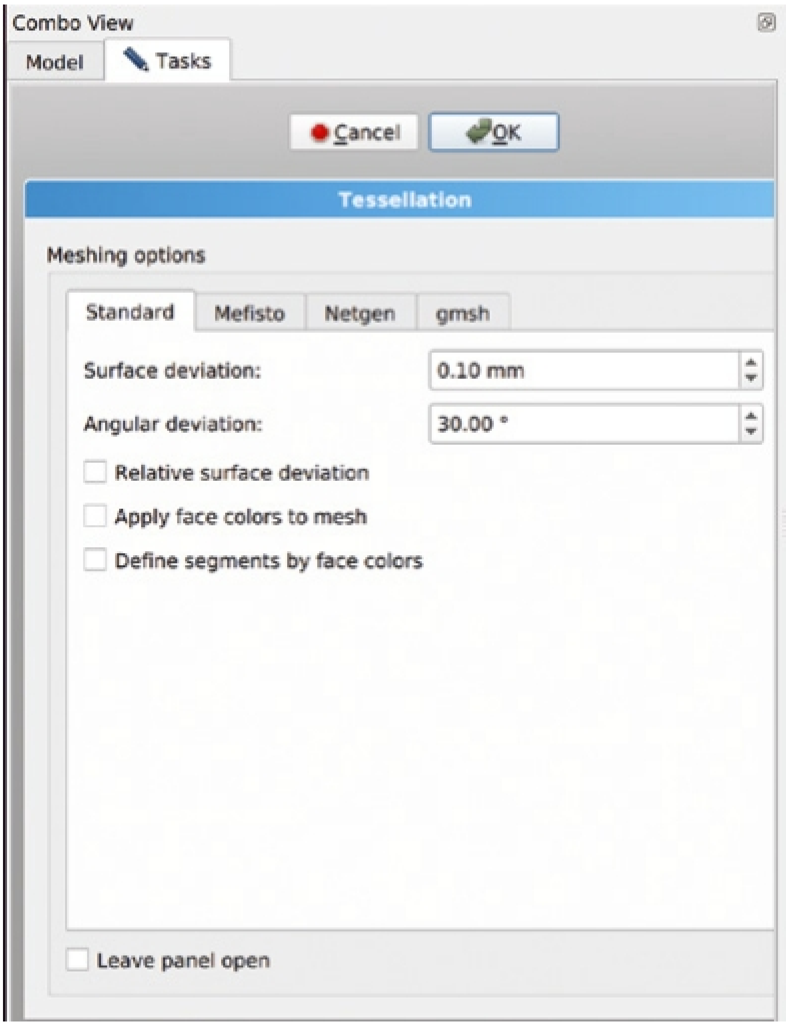
STL files have been around since the 1990s, and in simple terms, they contain a mesh of triangular facets that describe the shape of the design. STL files are a common mesh format to use in 3D printing where they are imported into a 'slicer' piece of software that generates the G-code for the 3D printer to follow.
As such, STL has become a standard file type for sharing 3D print designs on sites such as Thingiverse and GrabCAD and more. Whilst heavily associated with 3D printing, STL can often be used for other processes such as CNC milling or routing.
Most of the time, for simple parts, creating an STL file in FreeCAD is simple and somewhat automated.
As an example, let's quickly create any object in the Part or Part Design workbench. You can create a cube object and a sphere object in the Part workbench and cut part of the sphere out of the cube.



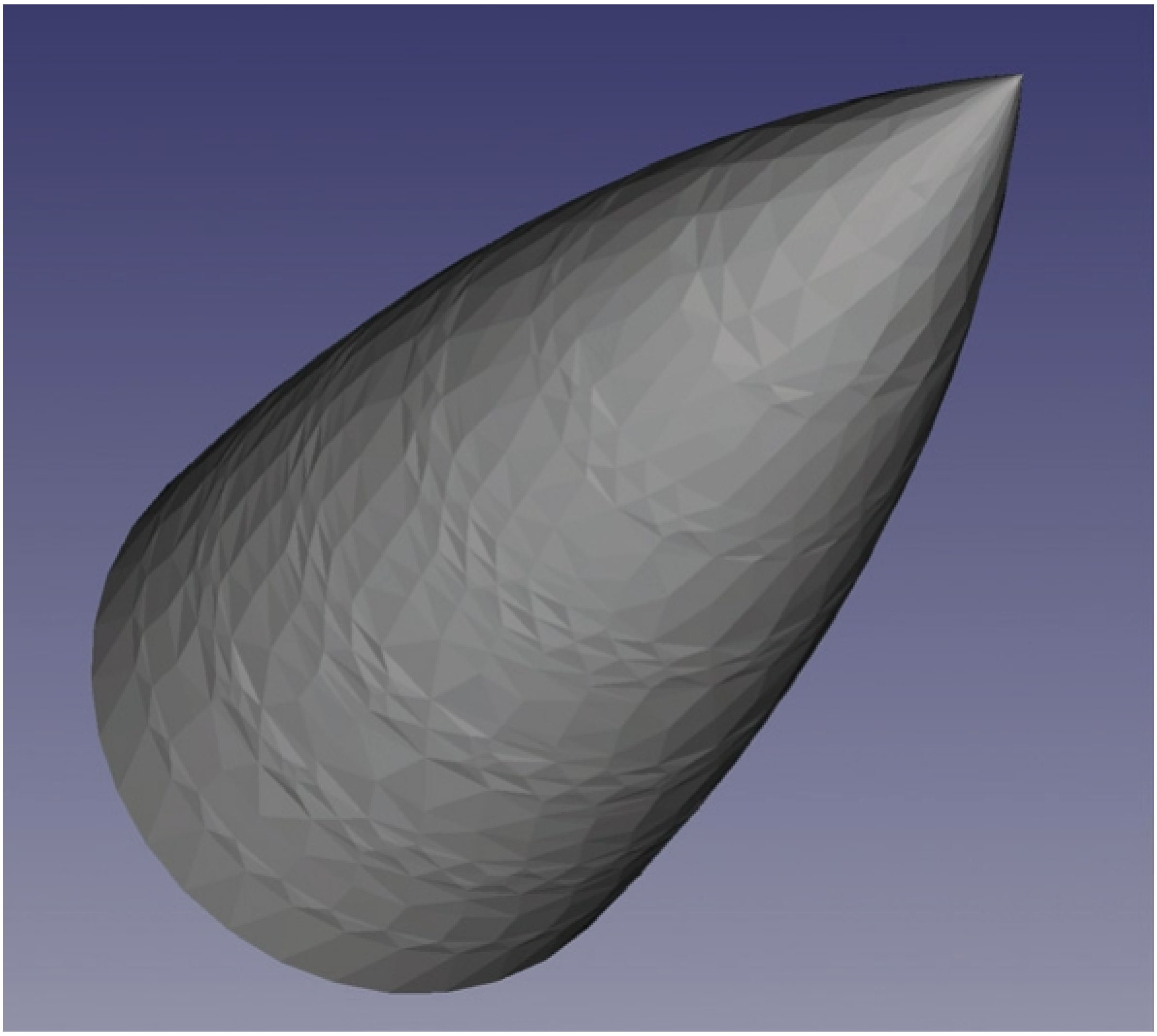
With the cut object selected in the file tree, click File > Export and you should see that FreeCAD suggests you export the object as an STL file. Give the file a name and export it, and it's ready to be sliced for 3D printing. In the background, FreeCAD has made a lot of sensible choices about the resolution of the mesh of your exported part.
Put simply, FreeCAD makes a choice about the number of triangle facets your exported STL mesh object contains.

Sometimes you might want to have more control over the mesh that's created for your part, and you might want to change the resolution to get the best results for your project. This is one area where the more specialised tools on the Mesh workbench can help.
As an example, you have a nose cone part for a rocket designed – let's move to the Mesh workbench using the drop-down menu to create some different resolution meshes to compare.
On the Mesh workbench, highlight the nose cone part and then click the tool that reads 'tessellate shape' on the tooltip as you hover over it.

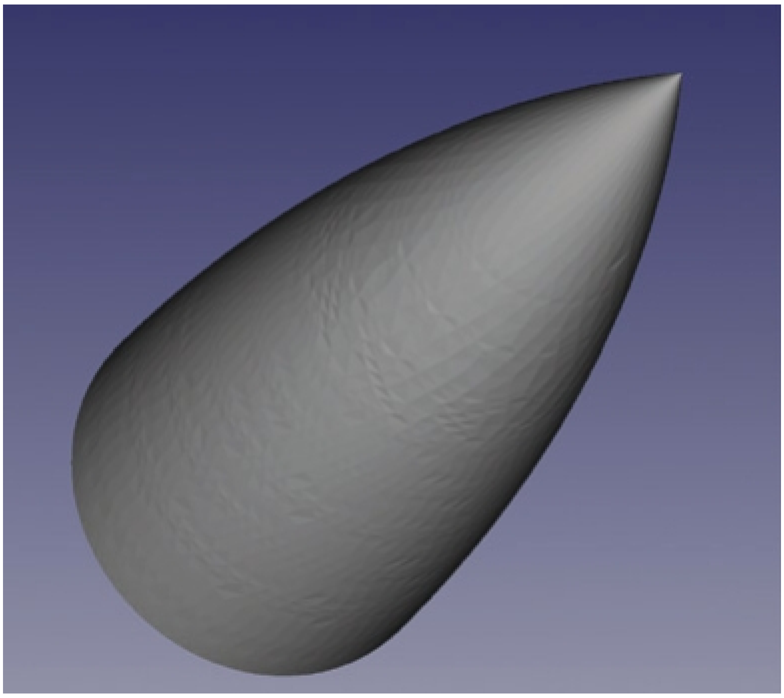
This tool is also accessible by clicking Meshes > Create mesh from shape. You should see a dialog box in the combo view area, and you should see two changeable values on the Standard tab.
The first value, 'Surface deviation', should be 0.10 mm, and the second value, 'Angular deviation', should be 30 degrees. If they aren't, set them to these values and then click OK.



Resolving Resolutions
You should now have two objects in the file tree: your original nose cone part and the new mesh object. Similar to other items in the file tree, you can toggle the visibility of the mesh and the part by highlighting the object and pressing the SPACE bar. Select the original nose cone part and set it as invisible so that you just see the mesh object in the preview window. You can see that the created mesh consists of lots of triangles closely approximating the part, but you can adjust the values to increase the resolution of the mesh if you delete your first mesh, make the nose cone part visible, and highlight it so you can create a new mesh.



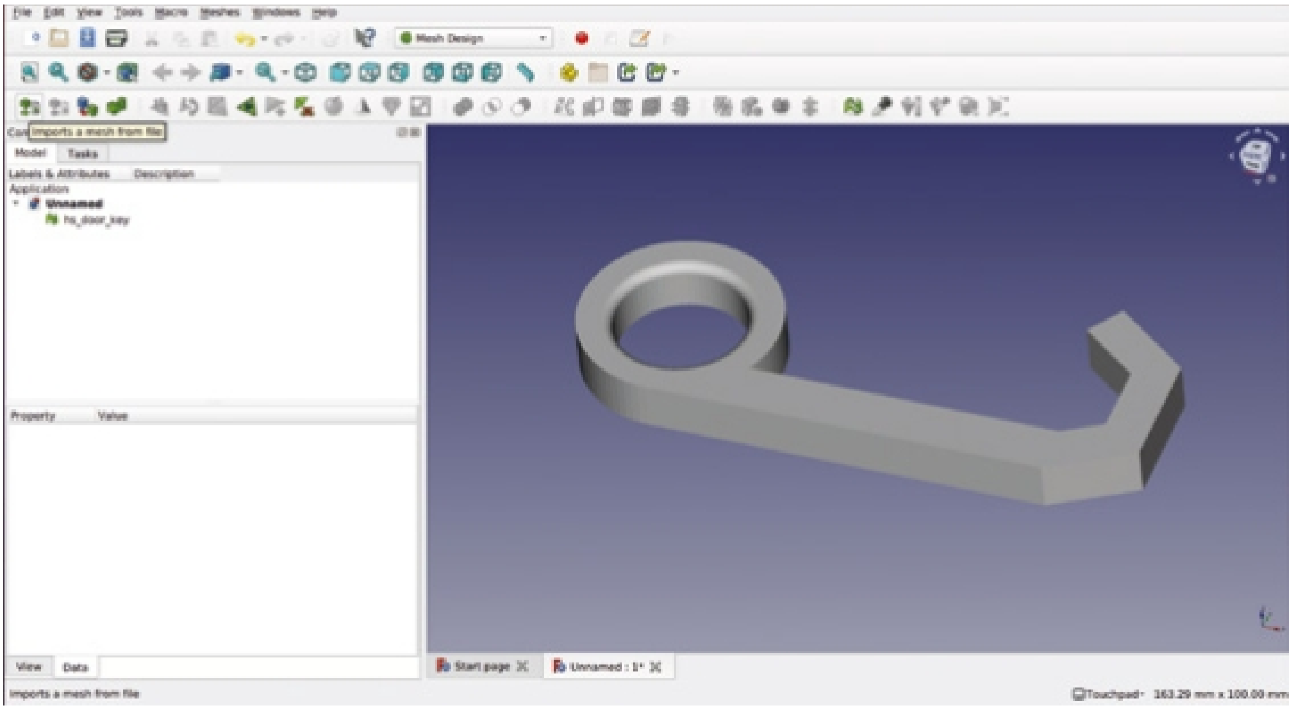
For this mesh, let’s set the surface deviation to 0.03 mm and the angular deviation to 10 degrees. You should see that the resulting mesh is a higher resolution and more precisely matches the original part.



Of course, this also increases the complexity of the mesh and, if taken too far, may well begin to slow down your graphics card performance.

To export a mesh from the Mesh workbench, you can highlight the mesh in the file tree, right-click, and select ‘Export mesh’. There are numerous options of STL types, but you tend to export as ‘Binary STL’ and, these seem to work well with your slicing software for 3D printing. One thing to note is that the Export mesh function seems to want us to add the ‘.stl’ to the file name in the name input box.
Of course, creating a mesh file from an object you made in FreeCAD is just one small aspect of the work of the Mesh workbench. One area it is particularly useful in is for editing and amending mesh files. This scenario often arises when we’ve found an object online that you wish to 3D-print but will require some reworking. Often, if you are downloading an object from a website such as Thingiverse or GrabCAD, you may only have the option to download a mesh file/STL, or indeed you might have the project file but it is made in software that you don’t own.

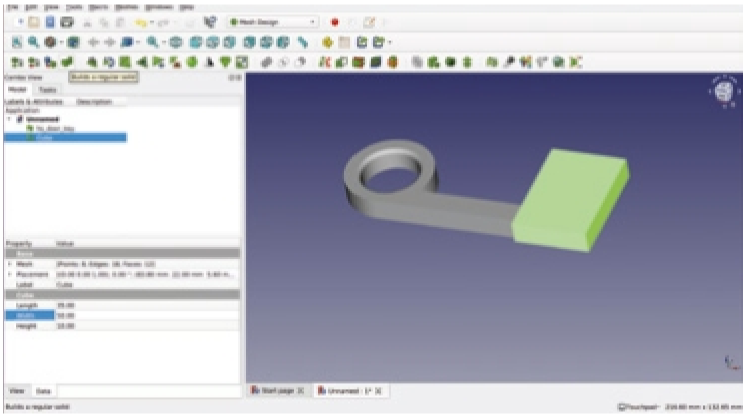
As an example, you have an STL file of a hands-free door puller.
Let's pretend that, in use, you decided it would be better if the hook faced the other way round. You could, of course, with the knowledge within this FreeCAD series of articles, draw this part from scratch in a matter of minutes, but let's use the Mesh workbench to make some changes. Click the 'Imports a mesh from a file' tool icon and navigate to the STL file and select to import
it.



The Mesh workbench has some tools that are similar in approach to the Part workbench, where you create simple solid objects and combine them using Boolean functions to stick objects together, or subtract one shape out of another. If you want to turn around the hook part of your mesh file, you will create a copy of the hooked end and rotate it using these tools.

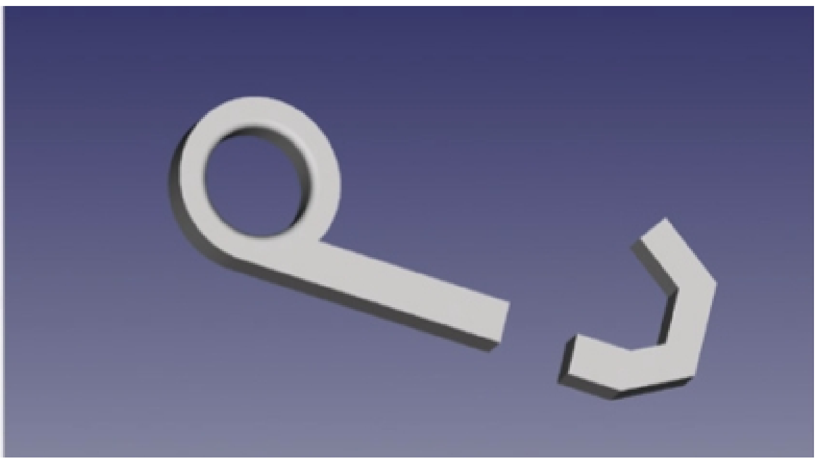
Regular Solids
First of all, click the 'Build a regular solid' tool icon or select the same tool by clicking Meshes > Regular solid.

You should now see a small window with a drop-down to select various shapes, cubes, cylinders, etc., and some entry boxes to add dimensions. You will create a cube 35 mm long, 50 mm wide, and 10 mm high.
Similar again to working with solids on the Part workbench, if you highlight the cube object in the file tree, right-click and select Transform, you can then move the cube into place over the hooked end of the door puller.

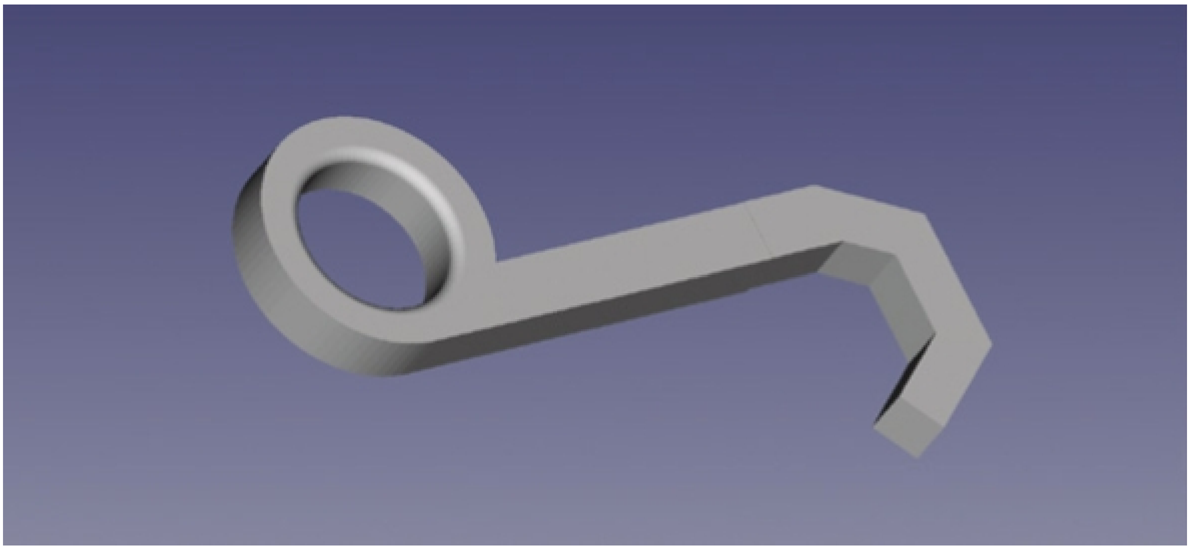


Next, select the two objects in the file tree and click the 'Intersection' tool icon. You will now see an additional item in the file tree labelled 'Intersection'.

If you toggle the cube item to be invisible and then select the intersection item, you can then right-click and use Transform to move the intersection object, which is a copy of the hook section, away from the rest of the door puller object. Next, make the cube visible once more, select the door puller object and the cube and then perform a 'Difference' Boolean operation on it. You should now have a version of the original door puller with no end and a copy of the hooked end.

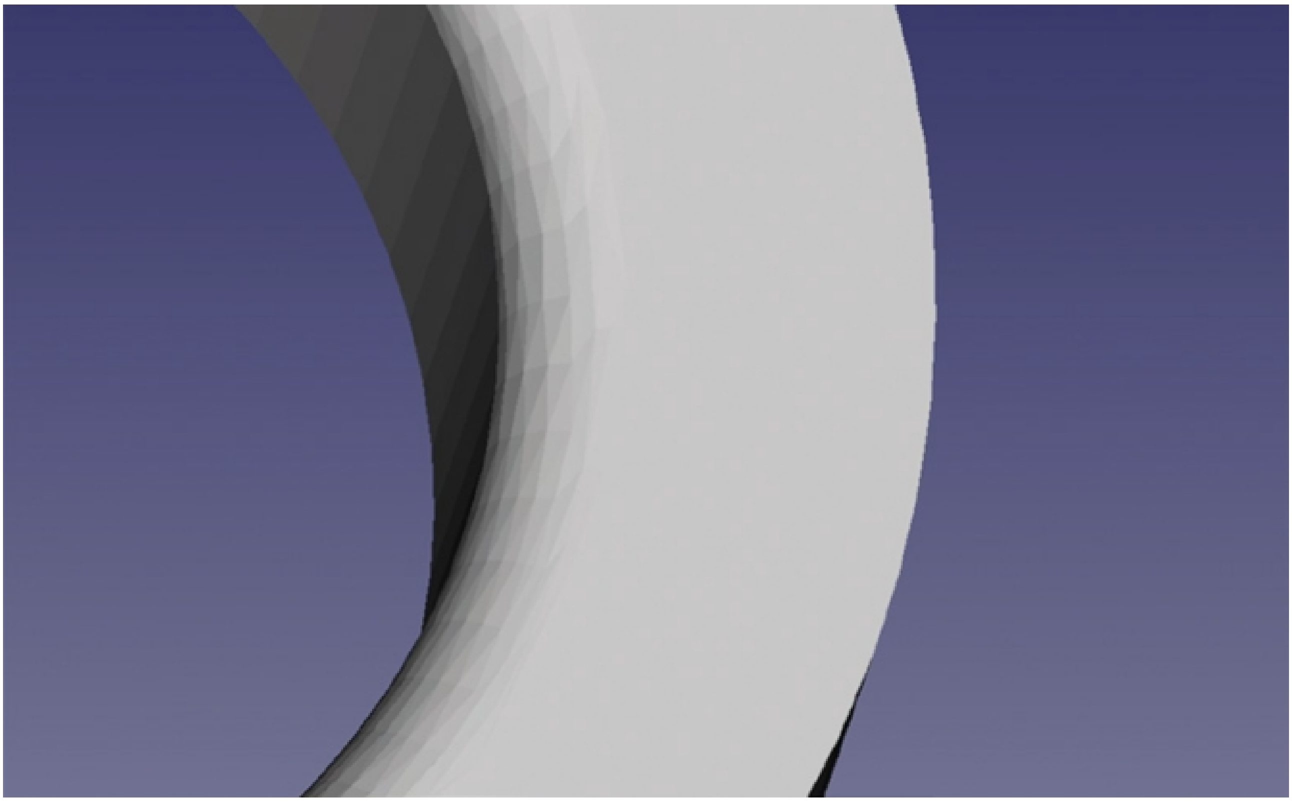


All that remains is to move and rotate everything into the new position, and then you can use the 'Union' Boolean operation to create a new single mesh object. You can then highlight the new door puller union object and click the 'Export a mesh to file' tool to create your new STL file ready for 3D printing.Continuing to use your door puller tool project as an example, let's explore how you create a Part object from a mesh. Moving to the Part workbench, you can highlight the mesh – in your case called 'Union' in the file tree view – then click Part > Create shape from mesh. You should now see that a part has been made of your door puller object.



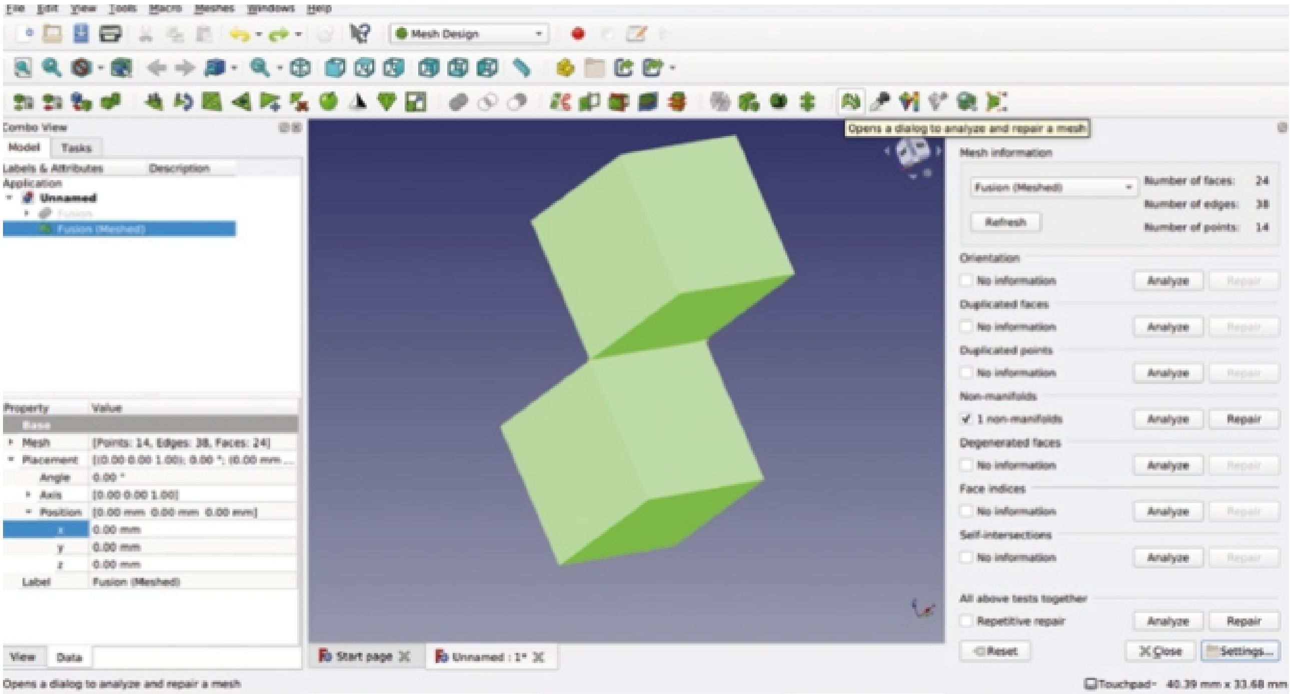
Let's make your mesh object invisible in the file tree to explore your part.
You should see that the part object creates its faces and edges based on the triangles in the mesh, which means that it is an accurate copy, but also means it can be a little challenging to work with. You can, of course, select faces and map sketches to faces, which is pretty straightforward on larger flat sections of a part but gets challenging if you consider the curved part of your puller. It can also be useful to convert meshes to parts if you want to recreate aspects of a mesh design. Converting a mesh to a part means that you can use the 'Link to an external geometry' tool in the Sketcher workbench to pull in reference points to create sketches over parts converted from meshes.
Sometimes you might want to work with extremely complex mesh files. If you've ever downloaded a complex model, such as a model of a person or object that's full of complex organic curves, you might notice that your graphics card starts to slow down when you manipulate the object. The FreeCAD Mesh workbench has a handy tool to play with optimising the mesh resolution that may be able to reduce the data, or put simply, the number of triangles in a given mesh while still retaining enough detail. This tool is called the 'Decimation' tool. Mesh decimation must be used in moderation, as taken too far, it's easy to completely destroy the object, which can also be a fun experiment. In the preview window, let's zoom in so that you have a close view of a part of the internal chamfer on the large hole in your door puller object.

Notice that when zoomed in, you can see the facets that make up the mesh surface. As this is a small file, we've made this mesh a pretty high resolution, but let's explore the effects of decimation. First, select the door puller object, in your case called 'Union' in the file tree. Notice that in the dialog box, once selected, you get some details of the number of points, edges, and faces your mesh contains. For us, these are 1606 points, 4818 edges, and 3212 faces. Next, click the 'Decimates a mesh' tool, which appears as a green diamond-shaped tool icon. In the Decimating dialog box that appears, set the reduction level to 50 percent and leave the absolute number box unchecked. Click OK. In the preview window, if you are still zoomed in, you should notice the resolution of your puller part change, it should look cruder and the triangles should be more obvious.



If you check the details of your mesh object in the dialog box, you can see it has reduced the number of points, edges, and faces by 50 percent to 803 points, 2409 edges, and 1606 faces. This carries through to affect the file size of an exported mesh, and indeed, exporting this decimated version creates a file size percent smaller than the none decimated mesh. If you zoom out on your 50 percent decimated mesh version of the door puller, you'll see that it makes
very little real-world difference to the resolution of the object, and the size and shape of the object is unchanged. You can experiment with the decimation value – it can be quite amusing to see what happens to a shape when you approach the near 100 percent decimation values. Sometimes you might work with mesh files, or indeed create mesh files, that have issues and problems. The Mesh workbench has a tool that enables us to quickly and repeatedly analyse meshes for problems. With your mesh object highlighted, click the 'Opens a dialog to analyse and repair a mesh' tool.
This should open a large dialog box on the right-hand side of the preview window. There is a list of potential problem items that FreeCAD can analyse a mesh for. You can analyse a mesh for each item on the list individually, or you can click 'Analyse' on the 'All above tests together' option at the bottom of the list. Close that dialog box for now, and let's quickly make a new simple object with a problem! In a new project, move to the Part workbench and create two identical default cube solid objects.

These default to 10 mm cubes, using Transform, move the second cube object up 10 mm in the Z axis and across 10 mm in the X axis. Finally, highlight both cubes in the file tree view and perform a Union operation to make the two cubes – one part called 'Fusion'. Moving to the Mesh workbench, with your fusion object highlighted, click the 'Tessalate shape' tool icon to create a mesh object. You can see the resulting mesh imported.



Make the original solid part fusion object invisible in the file tree, highlight the mesh object, and then reopen the mesh analysis dialog as before. Double-check that your mesh object is the object listed in the Mesh Information section (especially if you have multiple projects open), then click the Analyse button on the 'All above tests together' option. The tests will run quickly, and you should see just one error reported.You should see that in the list of analysis areas in the dialog, the 'Non-Manifolds' section has a check box with one non-manifold issue found. A simple way to think of this common error to understand when something is non-manifold is to consider if it is possible to be manufactured. In this instance, your two cubes are only connected at one edge and they do not overlap at all, which means, in essence, that the connected edge is infinitesimally small.

Of course, it's impossible to create a connection between these objects, and as such, they are described as non-manifold. Another way to create a non-manifold example is to create a tube with no ends where the surface of the tube has no thickness – this can be done using lofting tools or sweeping operations in FreeCAD, for example.

Fixing The Issue

Having analysed your cubes example and found the non-manifold edge issue, you may well be tempted to click the Repair button next to the offending issue in the dialog box. If you try this, FreeCAD will automatically apply a repair, but be prepared for this to fail as it will only work occasionally on simple problems, such as a small gap on a surface that needs closing. It's always worth a try, however, and you can always undo any odd changes the repair functions apply. For the sake of completeness, let's manually create a solution. If you move back to your part object and delete the fusion object so that you have your two separate cubes, move the upper cube back over and into the lower cube creating a couple of millimetres where they overlap. Then create a new mesh and analyse it – you should see that you have cured the manifold issue.

Finally, while we've looked at some useful stuff on the Mesh workbench, you have only really scratched the surface of what's available. As ever, there is lots more information, and tutorials, to be found online on both the FreeCAD community forums and in the FreeCAD wiki.

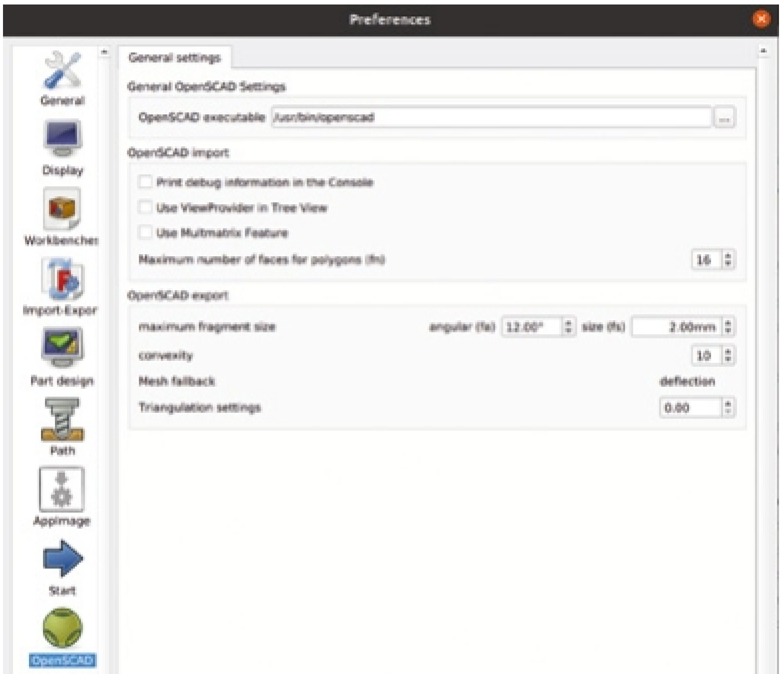
OpenSCAD

FreeCAD is a community-developed, open-source piece of software that uses other open-source pieces of code and modules within it. Another excellent open-source piece of CAD software is OpenSCAD, which uses a code-based approach to 3D modelling.

Files and models created with OpenSCAD can be imported directly to FreeCAD. FreeCAD has an OpenSCAD workbench to enable people to use the best of both software approaches.

In the Mesh workbench some of the functionality, including the Boolean operations you are using in this article, require OpenSCAD to be installed and a path to the OpenSCAD executable to be set up in FreeCAD.

Head over to openscad.org and download and install the latest version for your operating system. Once installed, back in FreeCAD, move to the OpenSCAD workbench and then click Edit > Preferences and scroll down to select the OpenSCAD icon. In the Preferences dialog, the first item is an input box to set the file path to the OpenSCAD executable – your system should have detected the correct path, but it's worth opening the file location and checking it is all set correctly. Once checked, click the Apply button and close the preferences.



Quick Tips

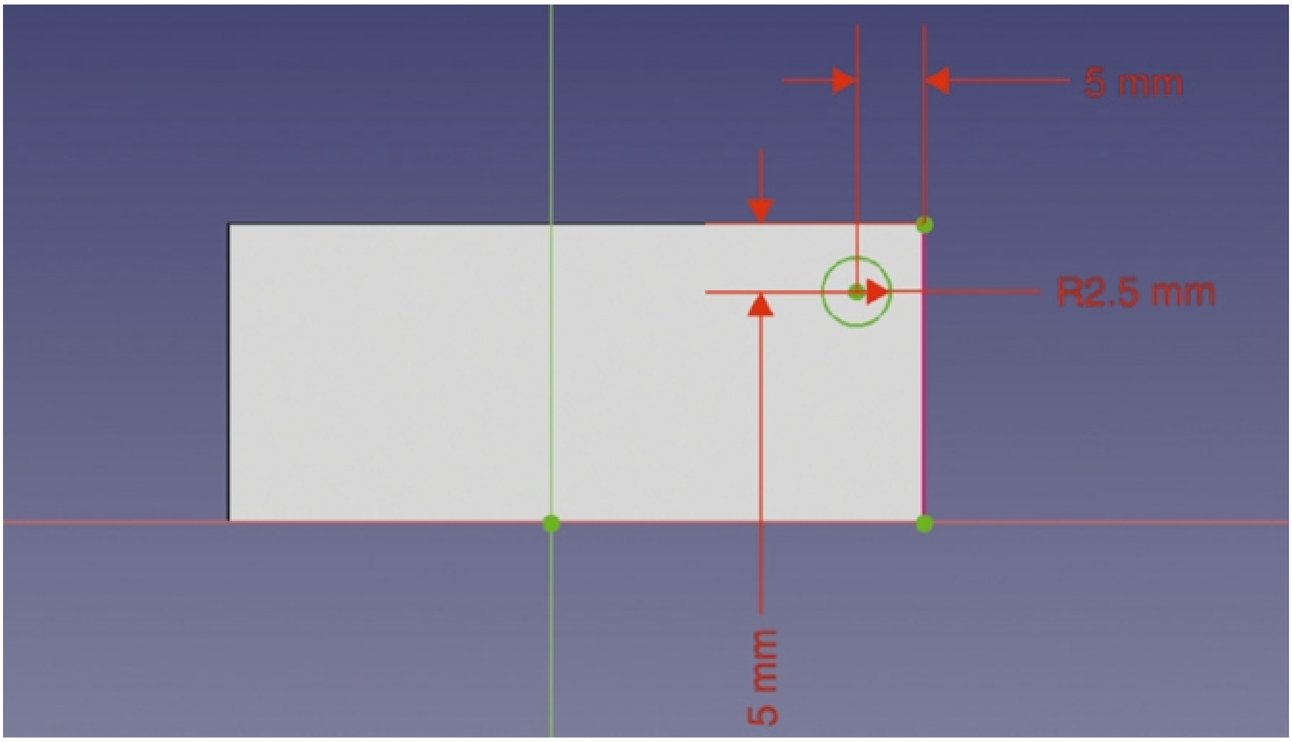
- While STL files are the most common mesh file, they aren't the only ones. In fact, many would argue that STL are not great files for mesh objects.
- The Intersection, Union, and Difference Boolean operation tool icons look like green versions of their counterparts on the Part workbench.
- You might want to install the brilliant Rocket workbench using the Add-on Manager if you want to play with the nose cone design.
- Another useful tool on the Mesh workbench allows us to scale mesh object. Simply select the mesh in the file tree and click the 'Scale selected meshes' tool icon.

Intermediate sketching

Often a sketch is the foundation of a part, and we have looked at the basics. Let’s delve deeper into more intermediate features and approaches for sketching in FreeCAD

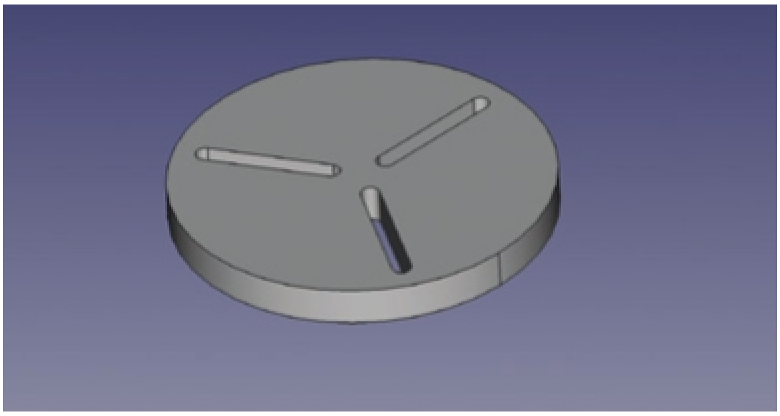
Previously, we have looked at the basics of sketching in the Sketcher workbench and then occasionally looked at other Sketcher features, such as cloning and making multiples of objects. As such a fundamental approach to creating parts, it’s certainly worth looking at in more depth.
To begin, let’s revisit an approach we have used previously which we can use as an example. Open up FreeCAD and start a new project, move to the Part Design workbench, create a body, and then create a sketch in the XY plane.
In your new sketch, draw a rectangle using the ‘Create a rectangle in the sketch’ tool and don’t worry about constraining it. Close the sketch and, back on the Part Design workbench, use the Pad function to pad the rectangle sketch into a larger cuboid.
Next, select any face on the cuboid in the preview window and click the ‘Create a sketch’ tool to open a sketch that is attached to that face.

If we imagine that we want to add a hole to the upper right-hand corner of our selected face, and that we want the hole to be specifically placed with respect to the upper right-hand corner, this is slightly tricky. It’s tricky, as the sketch underlying the cuboid object is unconstrained and we don’t know its size or its position relative to the origin point. We have used a tool previously though that allows us to import underlying geometry and help us solve this issue. The tool we have used is the ‘Create an edge linked to an external geometry’ tool which, when selected, allows us to select underlying geometry objects, edges, and points, and make them selectable in the current sketch. So, to accurately place a hole positionally relative to the upper right-hand corner, we can simply select the right-hand edge of the cuboid and then, in turn, use that imported geometry to set horizontal and vertical distance constraints to the centre of our hole circle sketch.



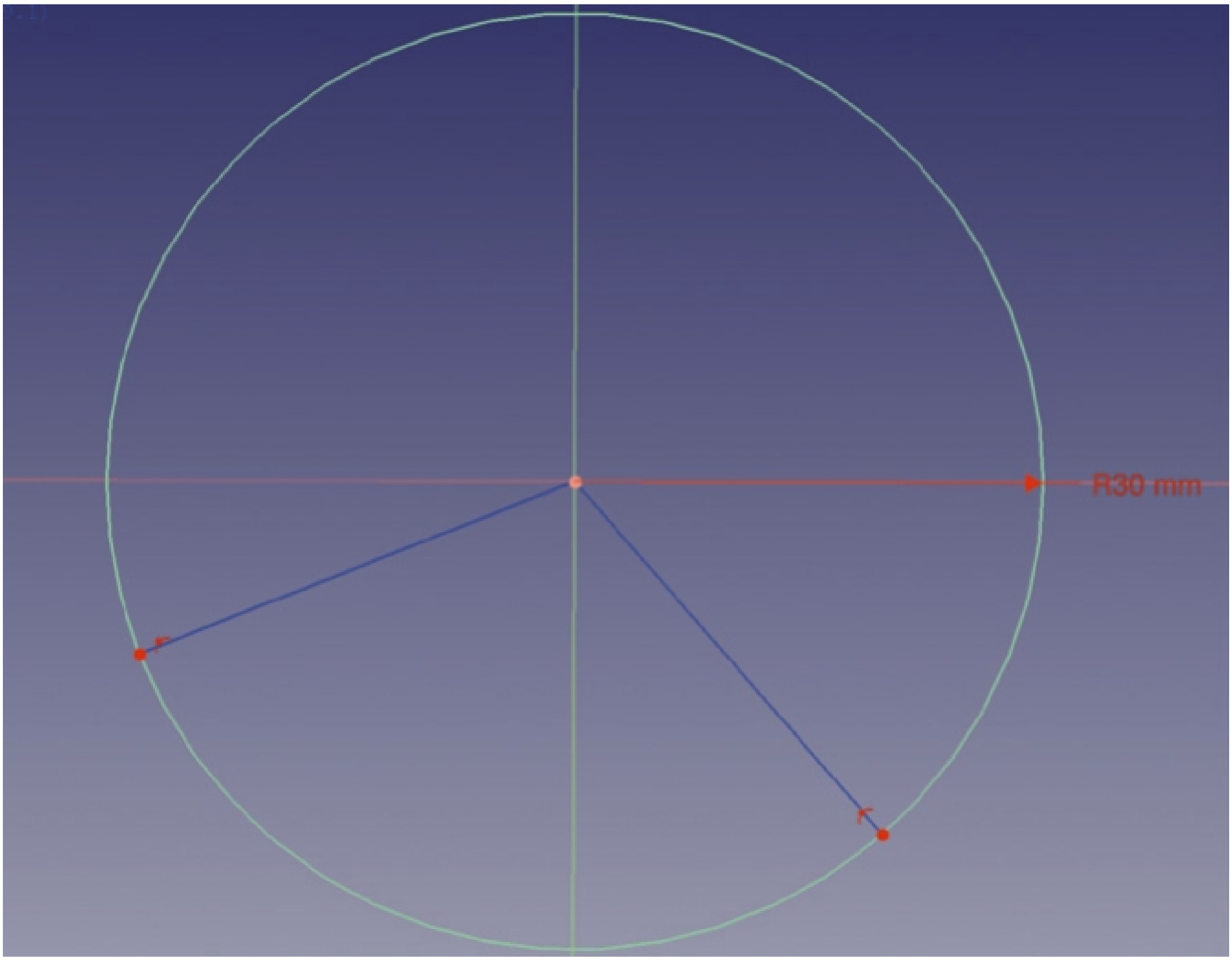
In essence, what we have done in our hole, in a cuboid example, is use an underlying geometry to create a guide point in the new sketch onto which we can attach constraints. We can see, therefore, that guide points can be incredibly useful, but often we may need a guide point in a position where there is no underlying geometry that we can import. An incredibly powerful feature of the Sketcher workbench is that we can switch all the drawing tools to 'construction' mode.
In a new project, again move to Part Design, create a new body, and create a sketch in the XY plane. Find the tool icon that looks like a red box over a blue dashed box, where the tooltip reads 'Toggles the toolbar or selected geometry to/from constructions mode', and click it.

You should see that all your drawing tools now turn from white lines in the icon to blue.
You should note, at this point, that drawing whilst in construction mode is exactly the same as standard mode. All the tools work in the same way. However, the geometries drawn will appear as blue lines and, when you move back to the Part Design workbench or other workbenches, you won't be able to see any construction geometry. The power of construction geometry is that, when you toggle back to normal geometry to draw visible parts of the sketch, you can, of course, attach points or constrain visible sketch edges and point to the construction geometry. Toggle back to normal drawing tools, and let's work through an example to show this in use.

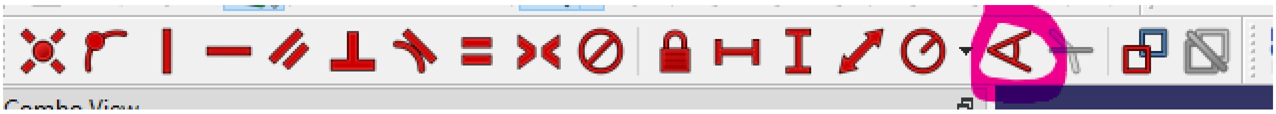


Here you can see the completed object we are aiming to make. We'll use some construction geometry to help, but along the way we will also explore the 'Create a slot' tool that we haven't used yet in this series. First, let's draw a standard circle in our sketch. Select the circle drawing tool and, starting from the origin point, draw a circle. Double-check your circle is centred to the origin, and then use the 'Constrain an arc or circle' tool to set the diameter of the circle to 30 mm. Due to the nature of the plane origin point and axis lines, you might now appreciate that we have the Y axis line onto which we can attach one of our slots correctly but that, as there are three equally spaced slots, we don't have geometry onto which we can constrain the further two slots. We can solve this by drawing two construction mode lines.

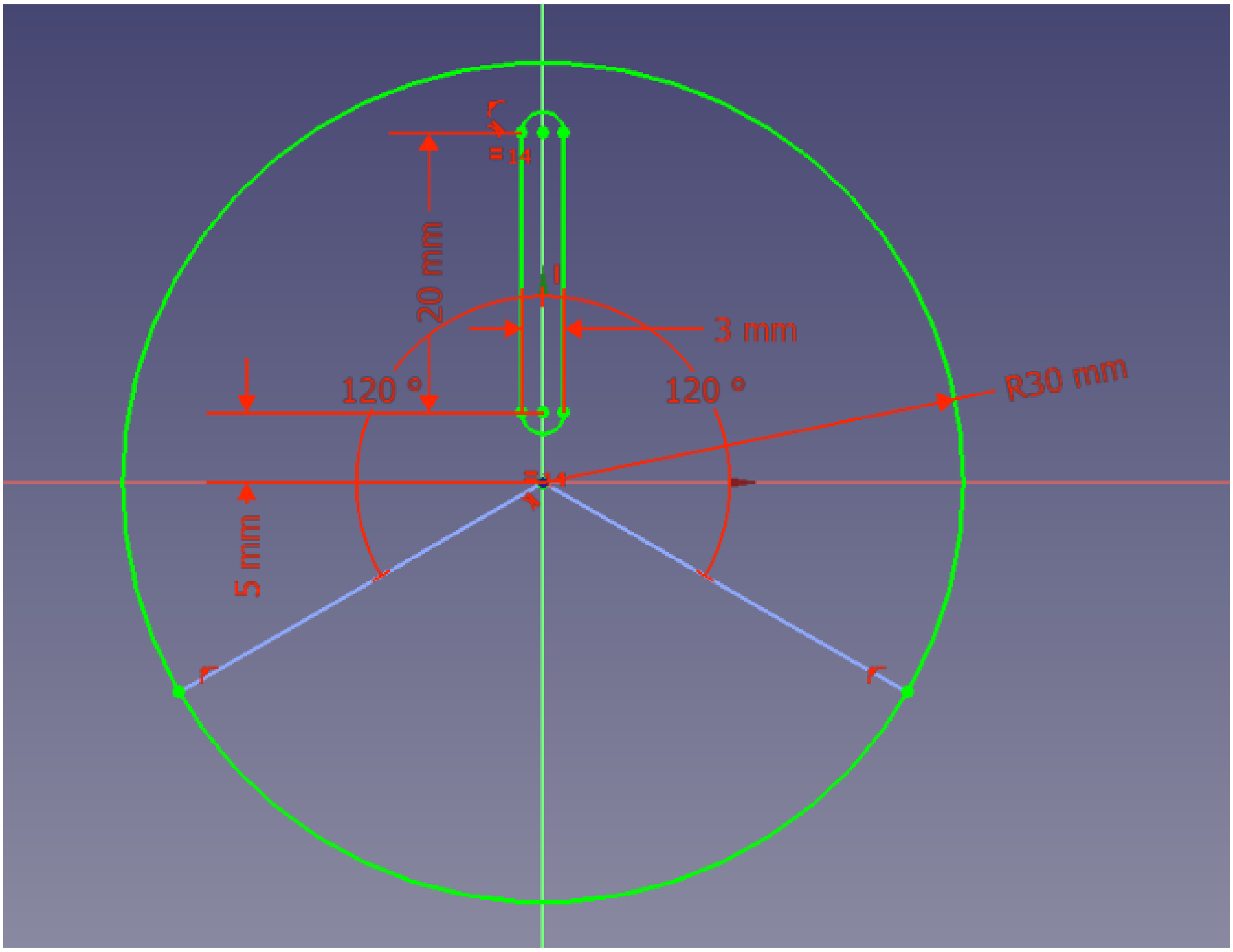
Click the 'Toggles the toolbar or selected geometry to/from constructions mode' once more, and then select the 'Line' tool. Click and draw a line that starts at the origin point and moves out at an angle to meet the edge of the circle. Do this twice, with one construction line in the lower right-hand quadrant of the circle, and one in the lower left.



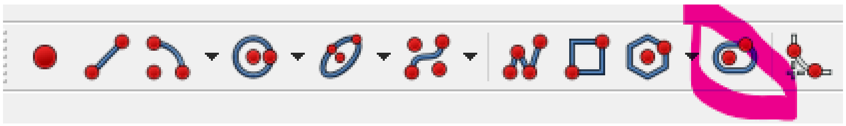
Next, click the 'Fix the angle of a line or the angle between two lines' tool, and select the vertical Y axis line and then the lower right-hand construction line.



As we want the three slots to be equally spaced, set this angle to 120 degrees. Repeat this, selecting the Y axis line and the lower left construction line so that we have both our construction lines correctly positioned.



Select the 'Create a slot in the sketch' tool, and make sure you have toggled the drawing tools back to normal mode and you aren't still drawing in construction mode.



Draw a slot by clicking somewhere on the vertical Y axis line and dragging down towards the centre of the circle, but don't connect the sketch to the centre origin point. Notice that when you draw with the slot tool, numerous constraints are automatically added, and this includes that the slot is either vertically or horizontally constrained. Let's fully constrain this slot. Begin by selecting the lower point on the Y axis line and set a vertical distance between it and the origin point of 5 mm. Next, select the two points in the slot that are on the centreline, and create a vertical distance between them of 20 mm. To set the width of the slot, select either set of two points that are at the end of the outer line, and mark the connecting point of the arc which forms the slot end. With two selected, set the distance between them as 3 mm. This results in a fully constrained slot with a length of 23 mm (slot length, plus the radius of the arc at each end) and a width of 3 mm.

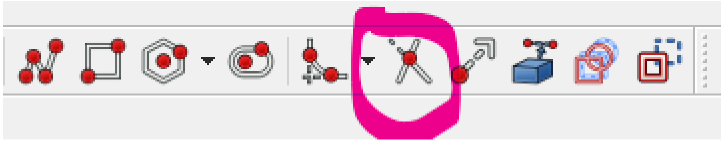
Sorting Your Slots

To draw the other two slots is slightly more complex. Draw a slot of any size in the lower left-hand quadrant. Again, the drawn slot will be constrained to vertical or horizontal automatically, and so we need to delete those constraints to be able to position it on an angle. Find the small red horizontal or vertical dash that is the constraint. Highlight this with a left-click and delete it with the DELETE key. You should now be able to drag the slot into an angled position.

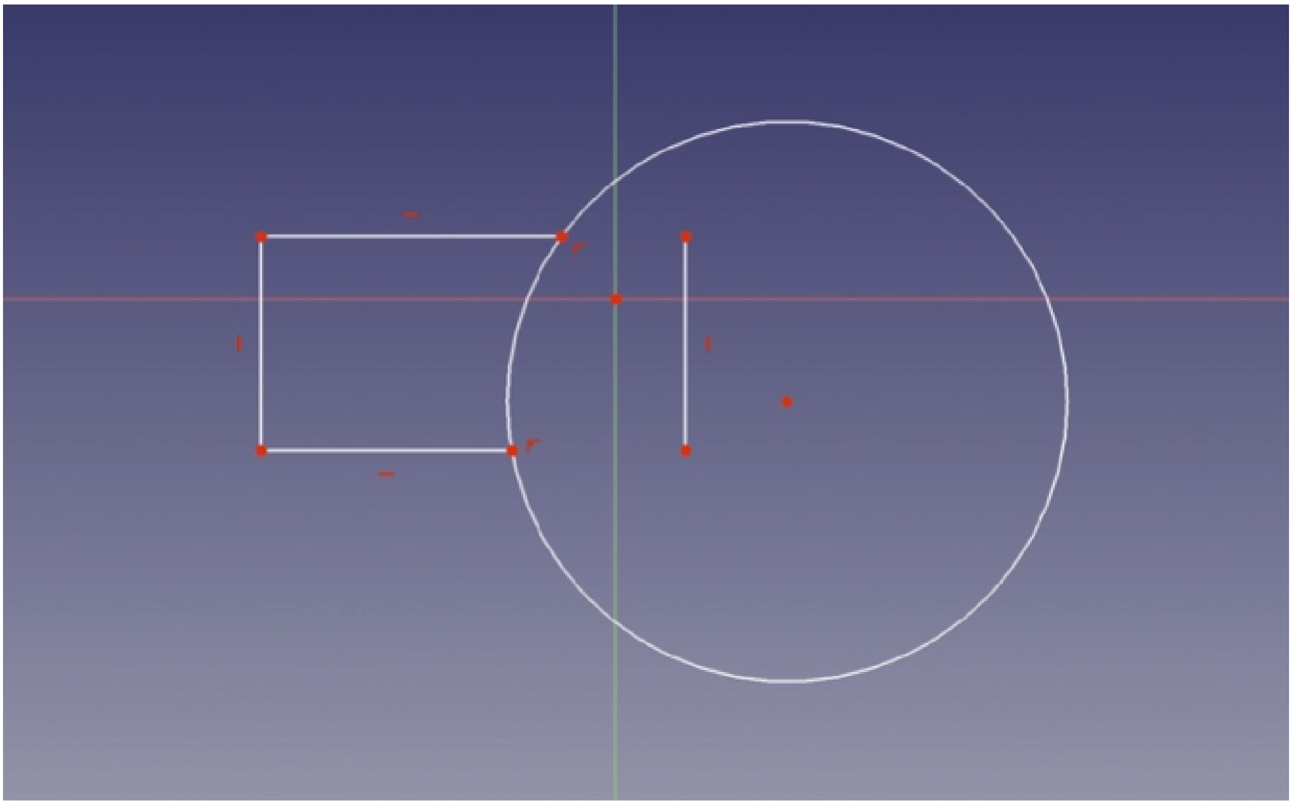
To correctly place the slot following the construction line, we can use a single symmetry constraint. First, select two points that are at the end of the same arc forming the end of the slot, and then select the construction line. Next, click the 'Create a symmetry constraint between two points with respect to a line or third point' tool – this should now constrain the slot can now add distance constraints to bring the slot to the same relative position as our first slot. However, note that, due to the angle, we can't use vertical or horizontal distance constraints, so we need to use the 'Fix the length of a line or the distance between a line and a vertex' tool.

Repeat the above to get all three slots fully constrained, and then close and pad the sketch to get the final result. Whilst this is a relatively simple example, it does show how combining construction geometry alongside regular geometry is a powerful sketching approach. You can use all the drawing tools in construction mode and, as such, there is no end to the possibilities.

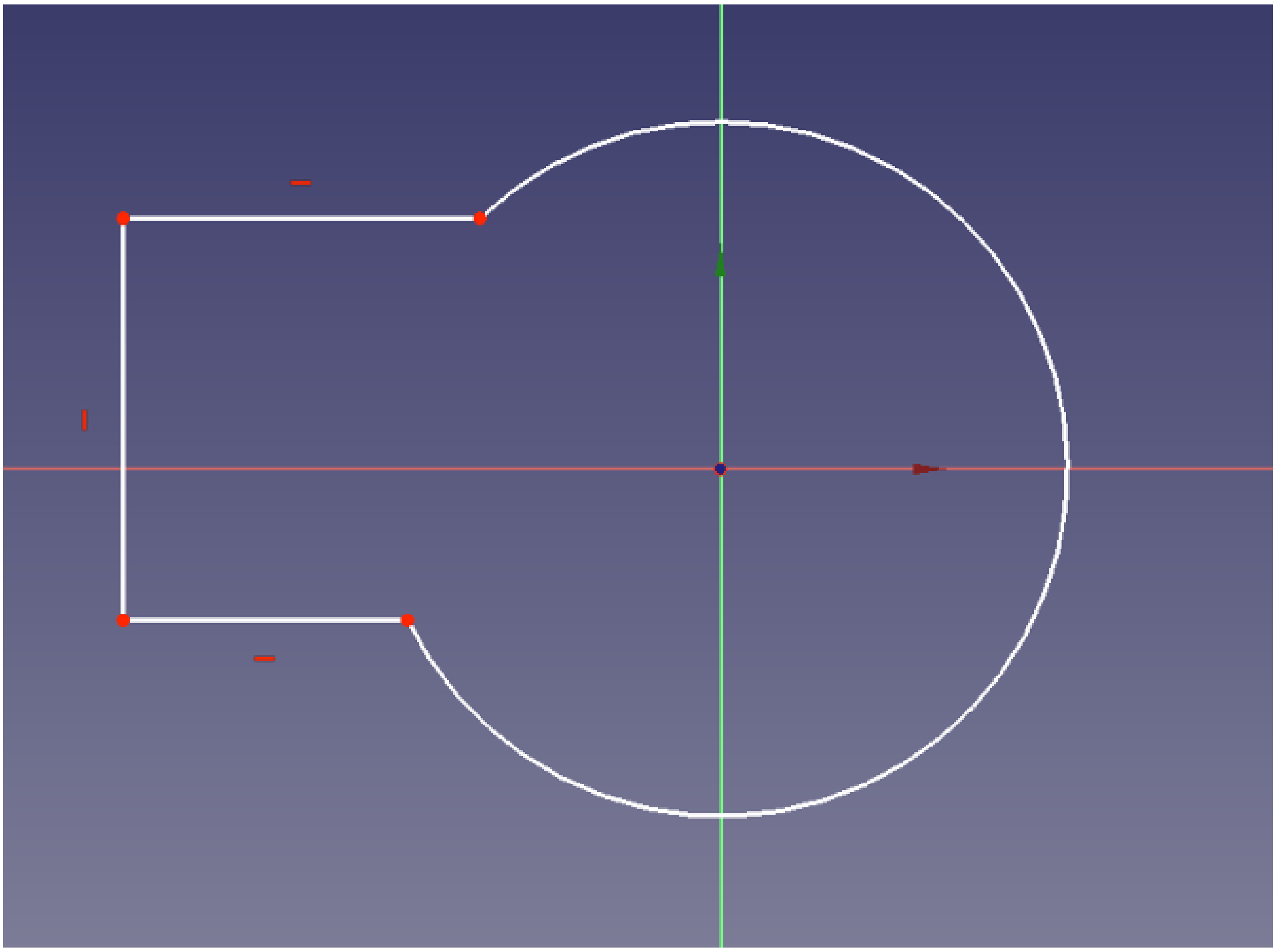
Trim An Edge
Moving away from construction mode, another really useful pair of tools are the 'Extend an edge with respect to the picked position' and the 'Trim an edge with respect to the picked position' tools, the latter of which is particularly powerful. If you have used Inkscape, the excellent open-source 2D vector graphics software, you probably have used the concept of Boolean operations, union, difference, intersection, and more, to combine different primitive shapes. This is similar to how, in the Part workbench, we use Boolean operations to combine primitive solids. It's a feature in Sketcher that I missed at first, until I discovered the 'Trim an edge with respect to the picked position' tool.
As a simple example, let's open a new project and again move to the Part Design workbench, create a new body, and then a sketch on the XY plane.
In this sketch, first let's select the 'Create a circle in the sketcher' drawing tool, and draw a circle anywhere on the plane. Next, select the 'Create a rectangle in the sketch' tool, and draw a rectangle that overlaps the circle. If we now select the 'Trim an edge with respect to the picked position' tool, we can use this to get rid of parts of lines we don't require.



If you click a line that intersects another line, that line section will be deleted, even if they are from separate objects like our circle and our rectangle.

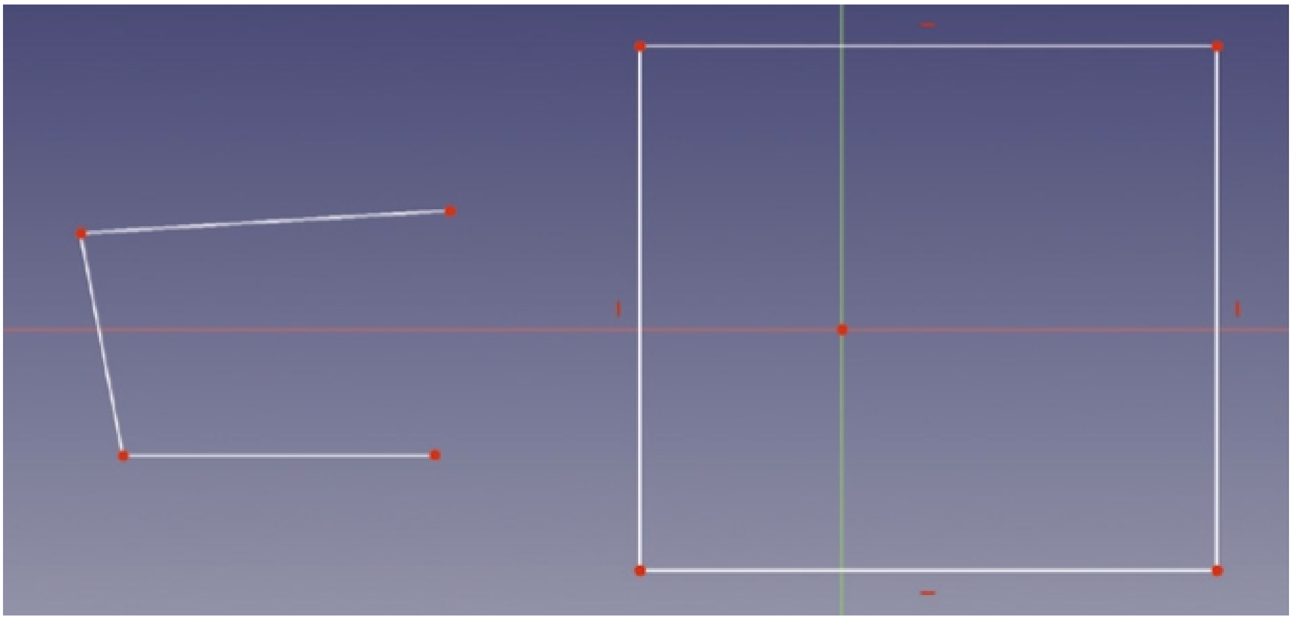


Here you can see that we have deleted two line segments inside the circle that belonged to the rectangle circle.
We can of course also remove the line that belongs to the circle that is in between the lines of the rectangle in exactly the same manner. That just leaves the now disconnected line left over from the rectangle that is free-floating inside the circle.

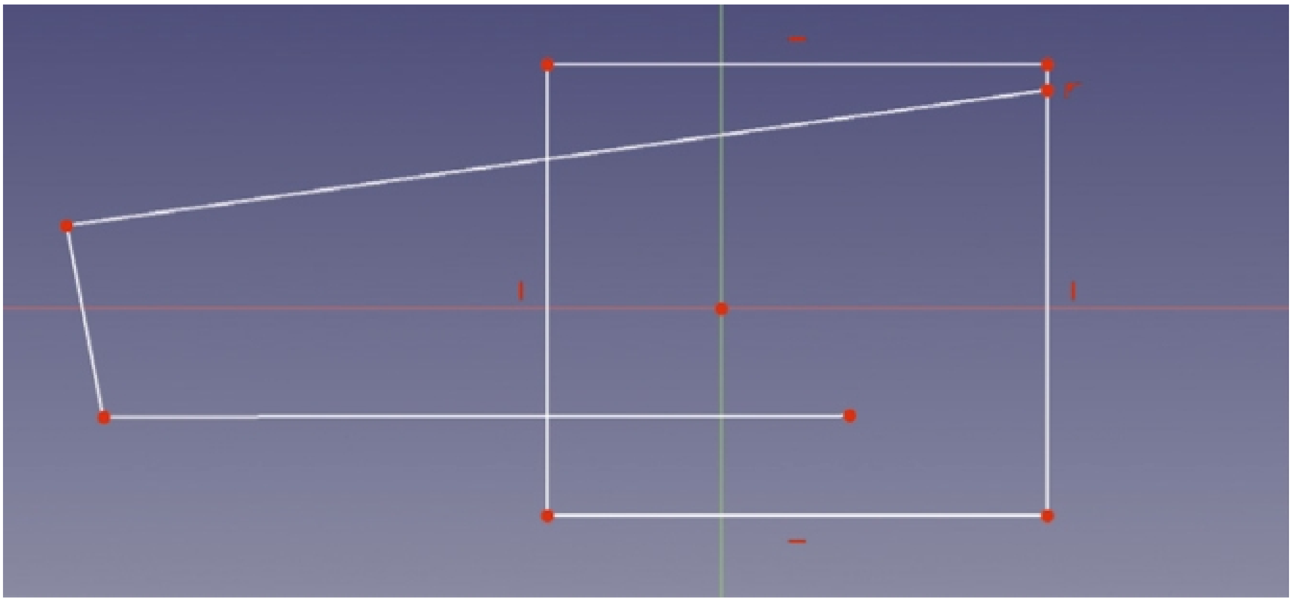


If we now try and use the 'Trim and edge with respect to the picked position' tool, we will get an error message. The Trim tool is only of use when a line intersects or is connected to another line.
If we had deleted this line first using this tool, it would have worked as it was still connected to the rectangle, but now this tool will fail. For non-connected lines such as this one, you can simply left-click on it to select it as an item and press the DELETE key.

Extend An Edge
A similar useful tool is the 'Extend an edge with respect to the picked position' tool. In a new project and a new sketch, draw a rectangle and then draw some polylines near the rectangle that don't quite meet the rectangle lines, similar to this.



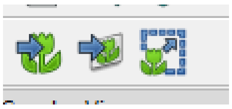
Next, select the 'Extend an edge with respect to the picked position' tool and left-click on one of the lines that doesn't meet the rectangle. If you move the cursor, you can now extend this line to any position you like or, as it crosses the line forming part of the rectangle, that line will become highlighted meaning that you can attach the line to the rectangle at that point.



This tool works not just on straight line geometry, but will also work to extend arcs and more.

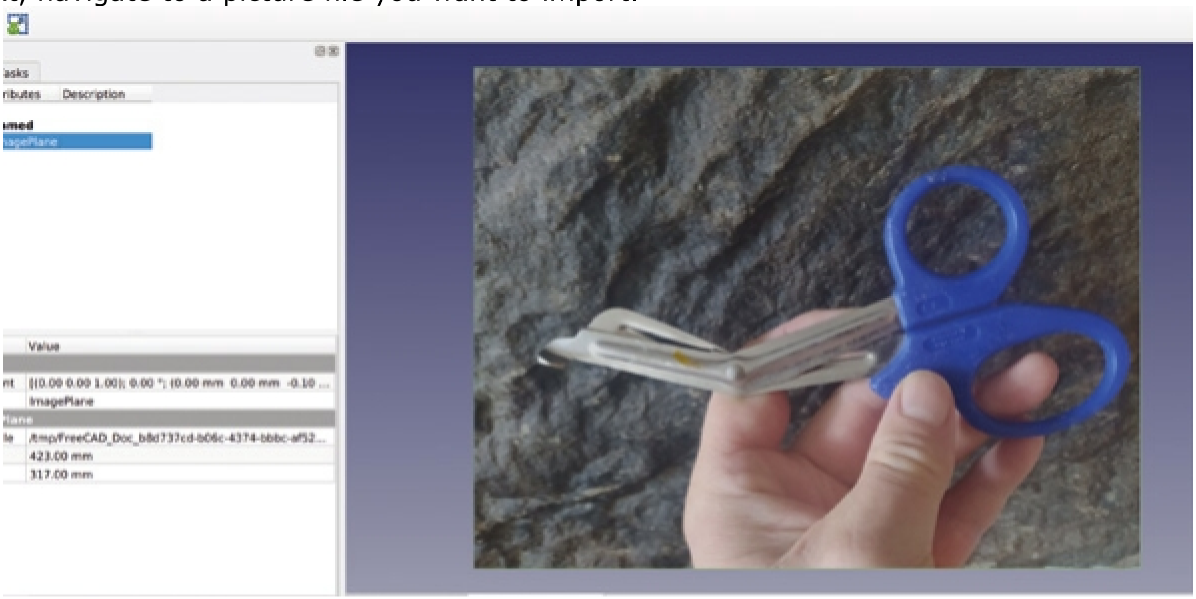
Image

Another slightly advanced sketching approach is the concept of sketching over an imported image, or images, to create some geometry that matches a real-world object. To do this we can use a separate workbench, 'Image', to import picture files to particular planes. We also can then scale the images correctly and more. Let's create a new project and move to the Image workbench. On the Image workbench, you will see that there are three main tool icons.



Find the tool icon that reads 'Create a planar image in the 3D space' and click it.

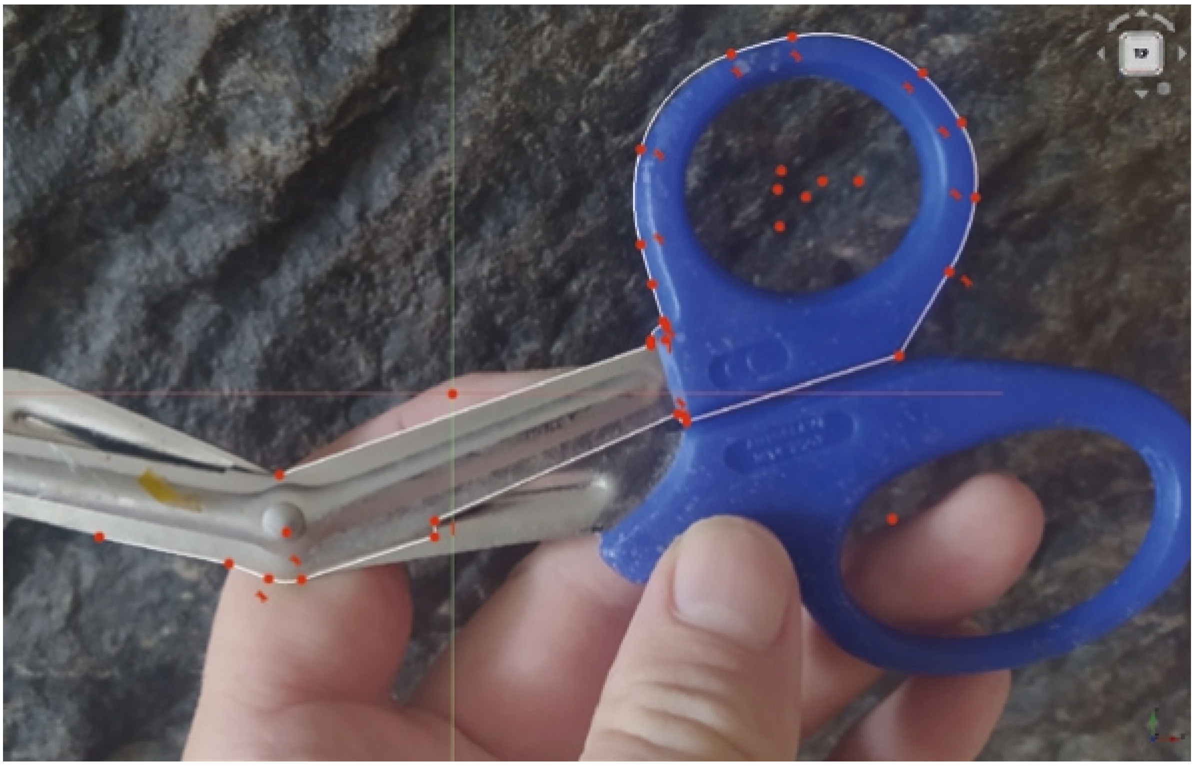
Next, navigate to a picture file you want to import.



Final Steps

Once you select an image, you will get a dialog box asking which plane you would like to attach the image to. We opted for XY in the first instance. The dialog box also has an 'Offset' input box – this is the distance from the selected plane where the image is placed. It's useful, when intending to trace parts of the image in Sketcher, to have the image placed slightly under the plane, so we added a -0.1 mm offset before clicking OK. You should now see your image loaded onto the selected plane and you can see an 'imageplane' object appear in the file tree view. Of course, like most items in the file tree, you can toggle the visibility of the imageplane item on and off using the SPACE bar.
Having imported our image successfully, we can switch to the Part Design workbench. Create a new body and then create a new sketch in the XY plane.

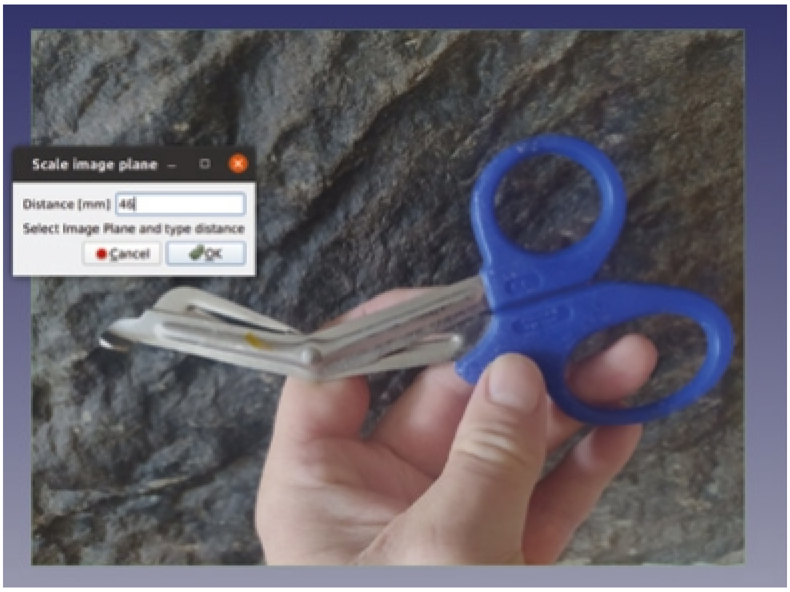
If you imported the image with the correct negative offset, you should now see the image in the Sketcher workbench but with the datum lines above the image.
To double-check, draw a shape in the sketch and make sure it appears over the image. You can now begin to draw and trace parts of your image using the Polyline tool. Using the M key to cycle through the different polyline modes, as described earlier in this article, can allow you to quite quickly create outline-traced sketches of quite complex items.



Tracing imageplane objects is an excellent approach, but one problem is that images, especially photographs, rarely scale correctly when imported.
If you have followed along and imported an image and traced (or partially traced) parts of it, as a test, select a line and click the 'Fix a length of a line or the distance between a line and a vertex' constraint tool.
You'll probably see that the length of the line doesn't match the reality of the object. In our case, if we click the first line on the upper part of the metal blade of our shear, we get a length of 105 mm, which is way too big. Whilst there are ways to scale a complete sketch using the Draft workbench, a better approach in this instance is to scale the imageplane object before we do too much work with it. If we have access to the object in the image, we can simply measure a part of the object.

To scale our image plane object correctly, let's hide the sketch by toggling its visibility in the file tree using the SPACE bar. Then, return to the Image workbench. Click the 'Scales an image plane by defining a distance between two points' tool icon and a 'Scale image plane' dialog box will appear. First, input a known distance that applies to two points on the object. So we input 46 mm. You then click on the first point on the imageplane object, so we clicked on the point on the top of the blade at the fulcrum point on the shears. Next, you click the second point representing the other end of the distance you input.

In our case, this is the point at which the upper blade joins the handle. Finally, the dialog asks you to click the imageplane object you want to scale. Then you click OK, and the imageplane object is now scaled correctly to the object.



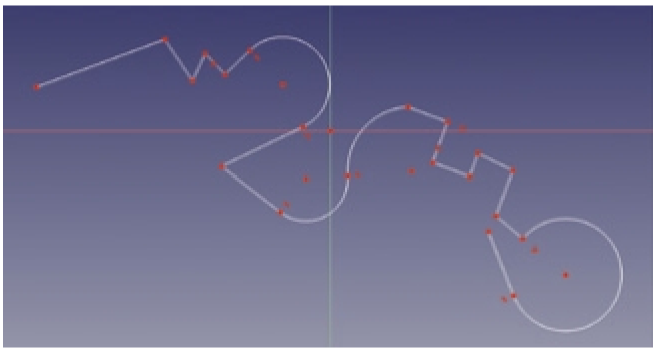
With a little forethought, we can make our image scaling process a little easier and more accurate. If we are creating the image ourselves, there are a couple of things we can do. Firstly, ensuring that our image is parallel to the object means that when we trace the item, we aren't tracing a skewed version of it. Making sure a photograph is taken directly from above can help; or, if sizes allow, even better is to create a scan image of the object on a flatbed scanner. Finally, if you add a ruler or a cutting mat with a grid to the background of your image, you can use this as the reference points for the 'Scale image plane' tool for really accurate results.

'M' Hot Key

A very useful feature within the Sketcher workbench, when using the Polyline tool, is the M hot key. When the Polyline tool is in use, pressing the M key cycles the tool through various modes that are incredibly useful. To explore this in a new sketch, select the Polyline tool and draw a few random lines in the standard mode.

Without right-clicking to complete a polyline, press the M key once. This first new mode enables you to automatically place a right angle line segment. Click the M key once more, and it will cycle the angle placement relative to the last polyline point. Click M again, and you can now draw an arc that is anchored tangentially on the last point of the line. Another M press changes the arc orientation, and a final M press allows you to draw an arc anchored to the last point, but not tangential.

Whilst this sounds complex in description, it's simply a case of playing with the Polyline tool and it soon becomes instinctive.

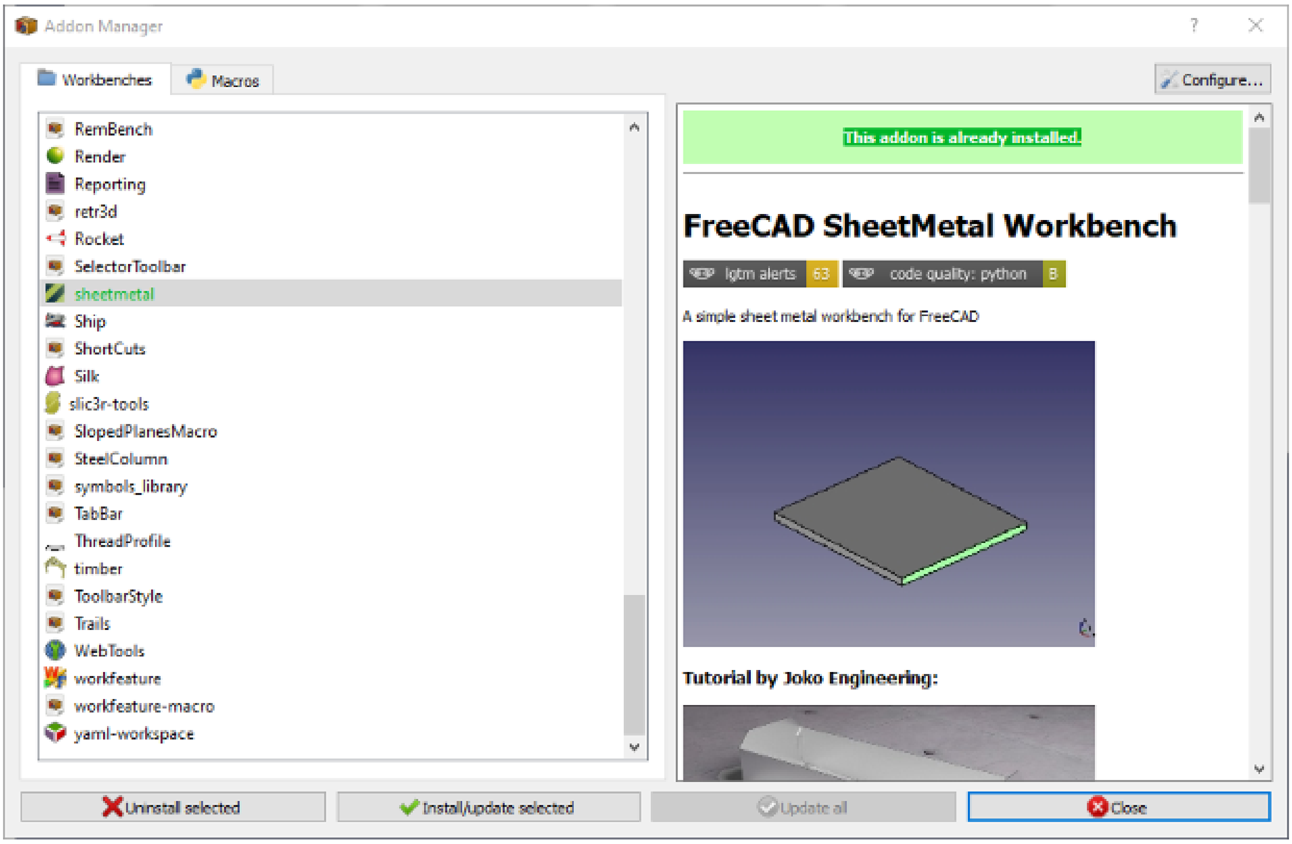


Quick Tip

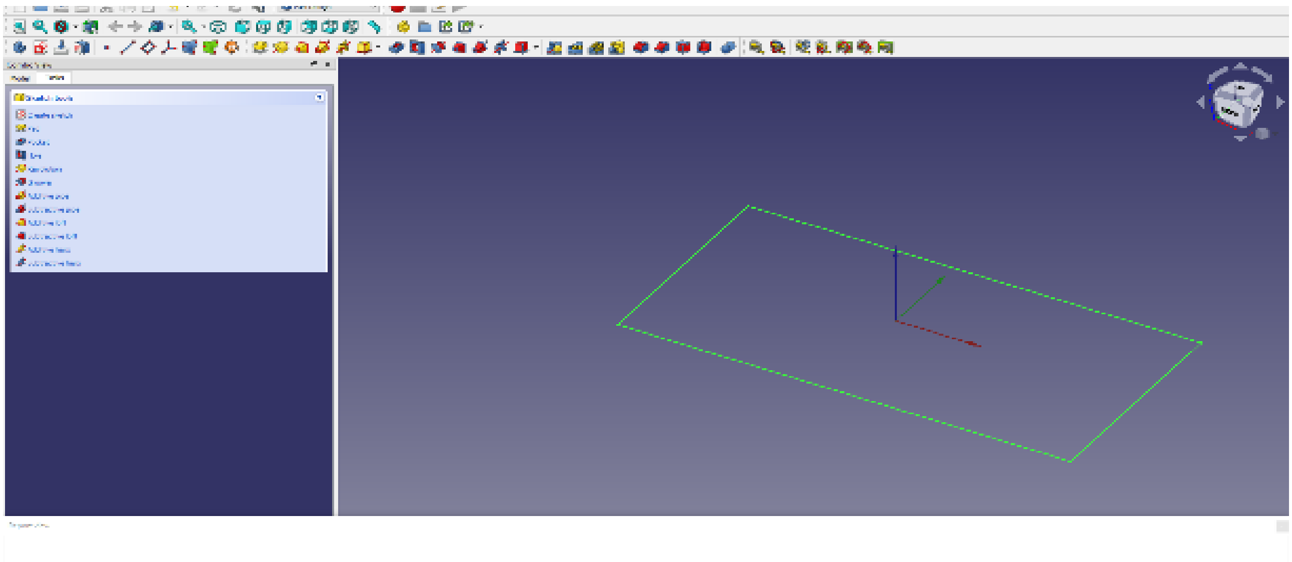
- The Image workbench works with lots of common file types, including SVG. However, you might be able to import an SVG as geometry, negating the need to sketch it!
- Most of the functions we have shown in this article work whether you are in construction or normal mode in Sketcher.
- When you import an image, it tends to import in an angled view. The view buttons and cube work exactly the same with the image plane, and pressing 'top' will give you a top-down view

Sheet Folding

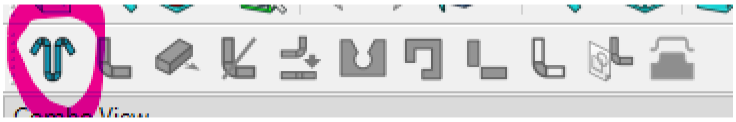
We are going to work mostly on an add-on workbench called 'Sheet Metal', so let's get that installed. Open up FreeCAD and click Tools > Addon manager, and then scroll down the list to find 'SheetMetal'. Click to install the workbench, and then close the Addon manager. You will get prompted to restart FreeCAD.



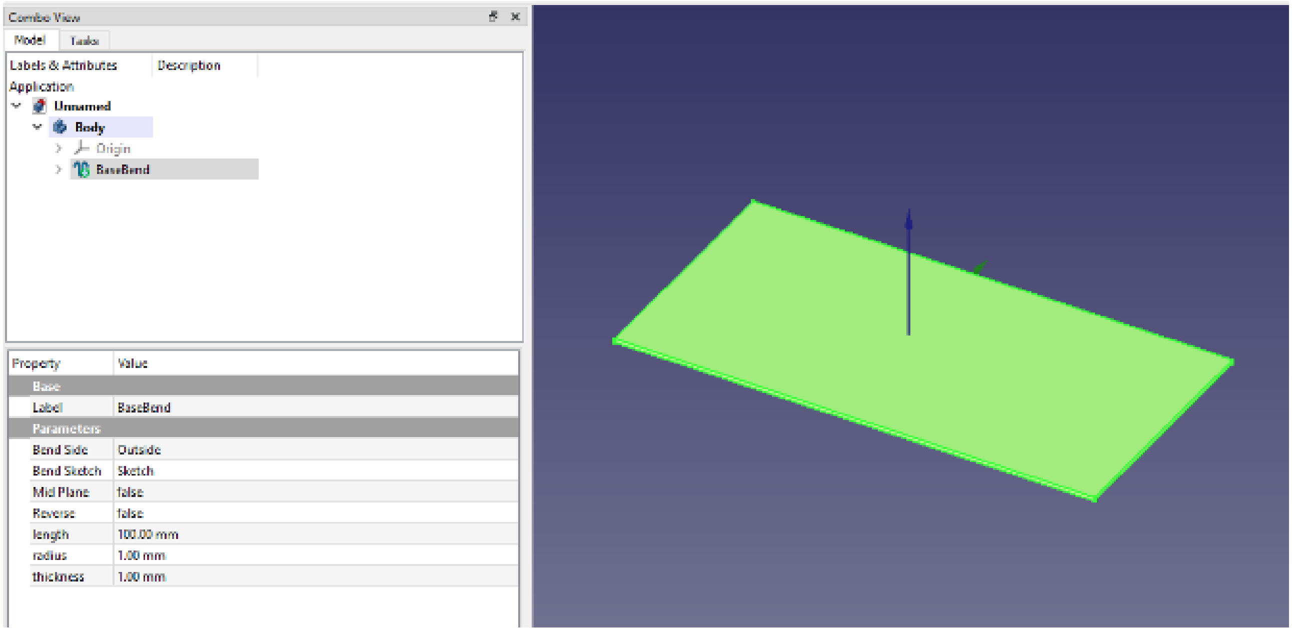
Once you have restarted FreeCAD, start a new project and, as a very simple first example, let's first go to the Part Design workbench, create a body, and then create a sketch in the XY plane. Inside the sketch, let's create a rectangle around the origin point. We don't need to fully constrain this sketch, but let's give it a horizontal and a vertical constraint, making it roughly 12cm by 6cm. Once this is done, close the sketch.



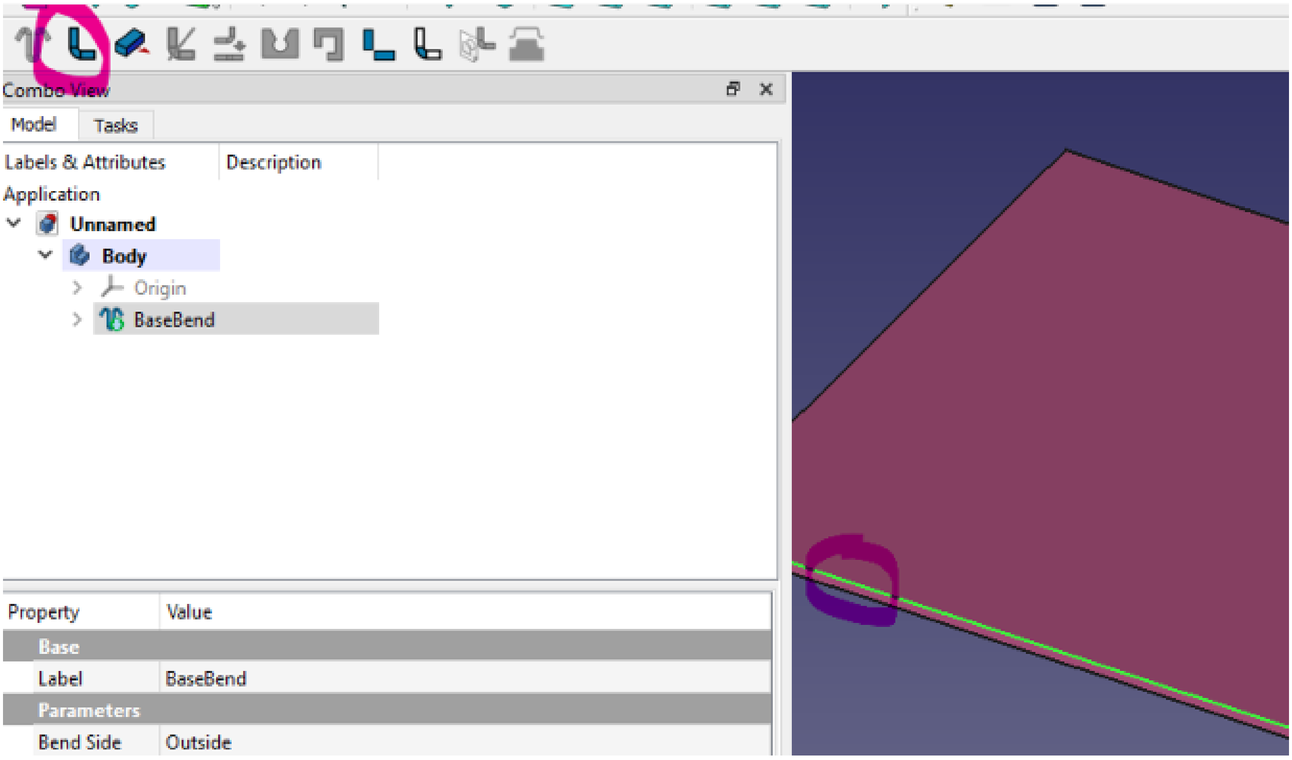
Without making further changes to this sketch, we can move straight away to the Sheet Metal workbench. In this workbench, with our sketch highlighted in the file tree, let’s click the ‘Create a sheet metal wall from a sketch’ tool icon.



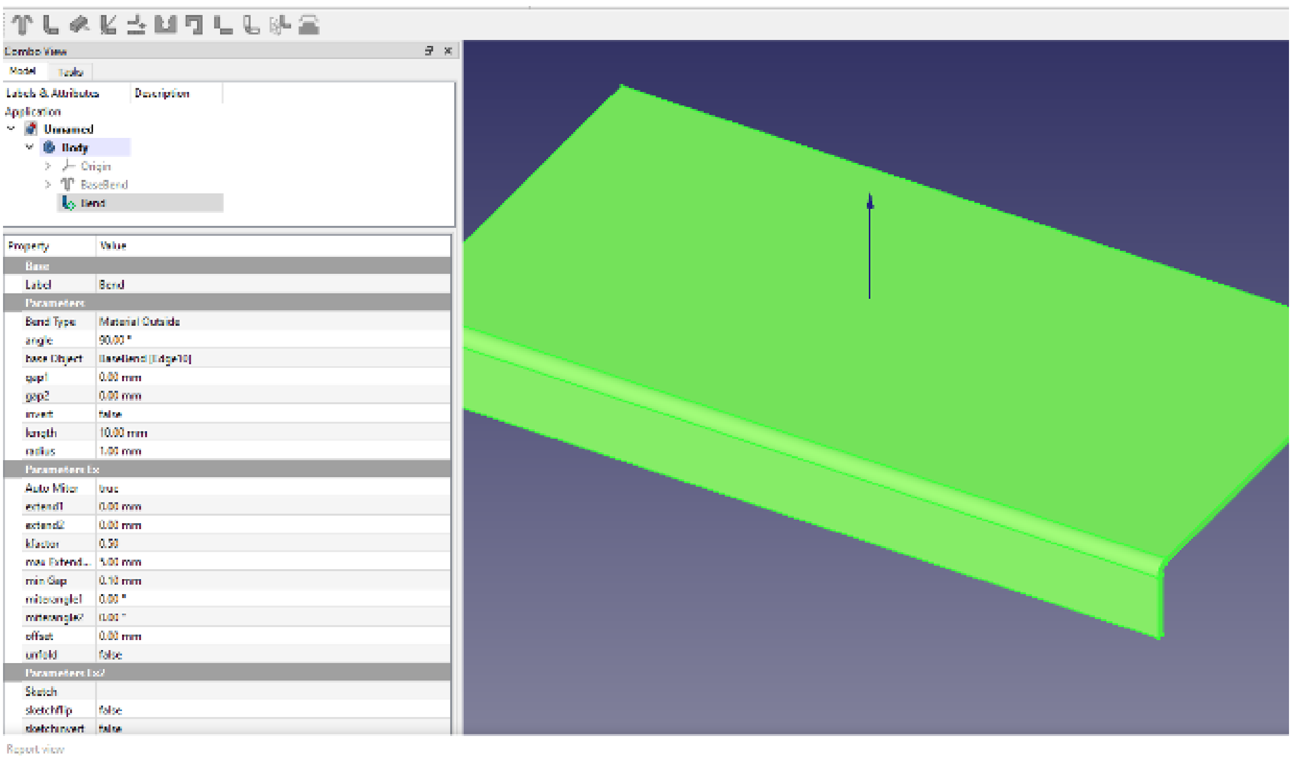
You should see that our sketch has been extruded slightly to form a sheet, and that a new object called ‘BaseBend’ has appeared in our file tree. Similar to if we had extruded our part, we can still make changes to the underlying sketch, and those will be pushed through to the BaseBend object. There are also numerous parameters in the BaseBend object dialog. Notably, you can adjust the thickness of the sheet – this is useful since the BaseBend object, as you will see, defines the sheet thickness for the rest of the attached design.



Next, let’s select one of the edges on the BaseBend object. We selected the upmost edge line of one of the longer sides. With that selected, click the ‘Extends one or more face, connected by a bend on existing sheet metal’ tool icon, which should be second from the left in the Sheet Metal workbench toolbar.



You should now see that we have created a new section of our sheet design, folded at 90 degrees to the original sheet (Figure 2).



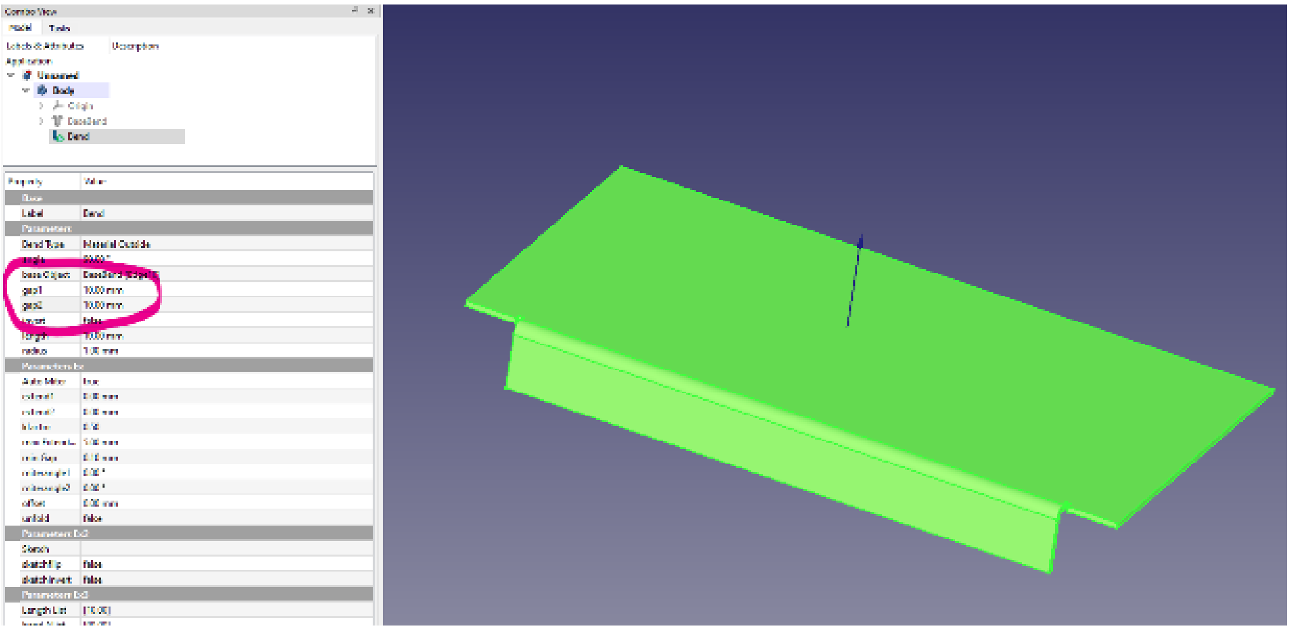
This new object appears in the file tree labelled ‘Bend’ and subsequent bends will be labelled ‘Bend001’, ‘Bend002’, etc.

Highlighting the 'Bend' object in the file tree, we can adjust plenty of parameters.

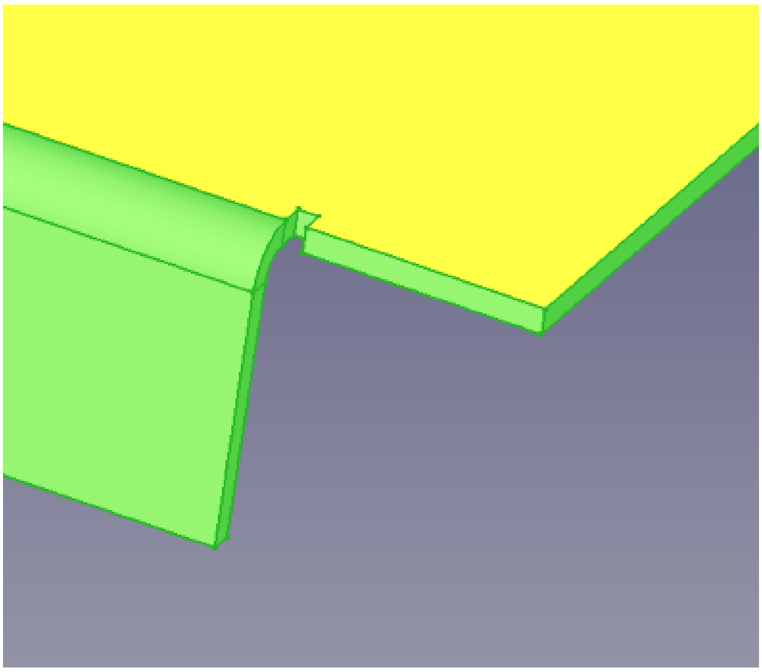
Model | Tasks

Property	Value
Base	
Label	Bend
Parameters	
Bend Type	Material Outside
angle	90.00 °
base Object	BaseBend [Edge10]
gap1	0.00 mm
gap2	0.00 mm
invert	false
length	10.00 mm
radius	1.00 mm
Parameters Ex	
Auto Miter	true
extend1	0.00 mm
extend2	0.00 mm
kfactor	0.50
max Extend...	5.00 mm
min Gap	0.10 mm
miterangle1	0.00 °
miterangle2	0.00 °
offset	0.00 mm
unfold	false
Parameters Ex2	
Sketch	
sketchflip	false
sketchinvert	false
Parameters Ex3	
Length List	[10.00]
bend AList	[90.00]
Parameters Relief	
Relief Factor	0.70
Use Relief F...	false
min Relief G...	1.00 mm
relief Type	Rectangle
reliefd	1.00 mm
reliefw	0.80 mm

We'll leave the first parameter, Bend Type, as it is, set at 'Material Outside', but have a glance at the options for later on. You'll note that our new folded section is folded down from the original BaseBend object in the Z axis. If we wanted this section to fold upwards, we can change the 'angle' parameter in the dialog. Setting this to -90 rotates the folded section to point upwards. Similarly, we can set the angle of this fold to any amount we choose. Moving down the parameter list, we can use the 'gap1' and 'gap2' parameters to make the folded object not the full length of our attached BaseBend edge.
So, for example, we have added a gap value of 10mm to each of those parameters.



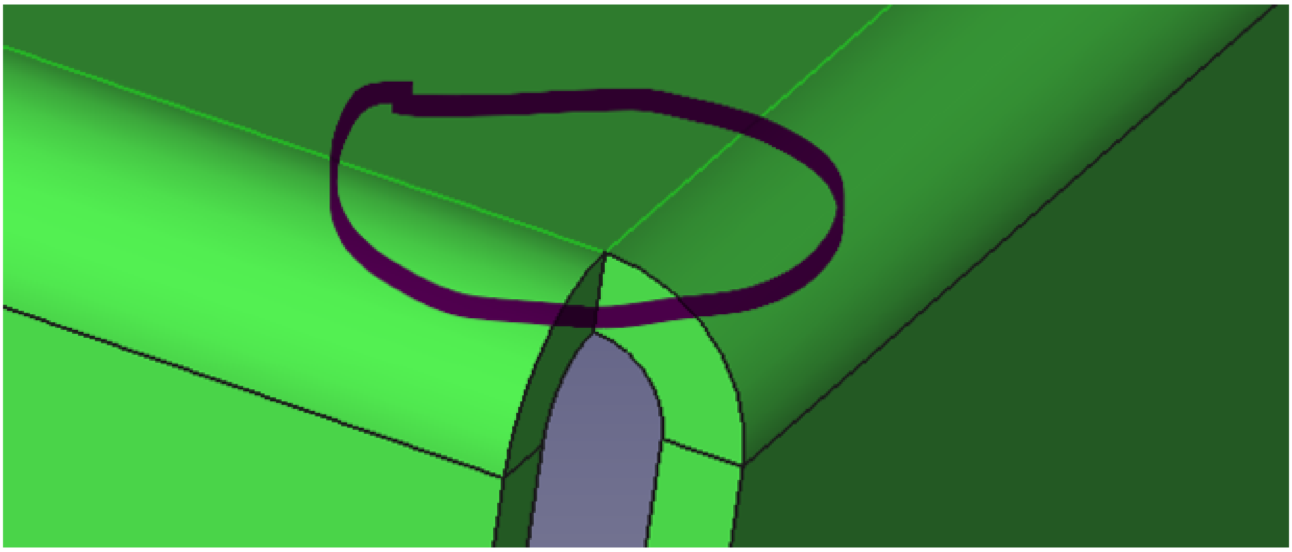
You may need to click the 'Recomputes the current active document' button or press CTRL+R to see the changes in your model.
You'll notice that not only does the bend object reduce in length by the gap amount, but also that there are some small 'relief' notches automatically added at the end point of the bend. This is good practice in sheet metal work as, if you fold a bend that connects to a straight edge, the end of the bend will distort the straight edge at the point it joins it. Adding the notch stops this distortion from occurring.



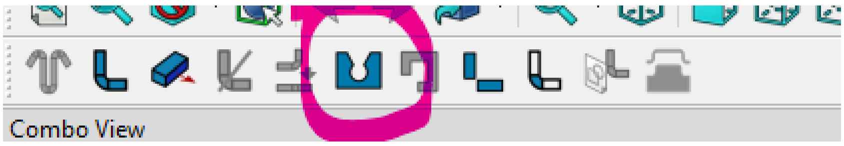
If you wanted to not have a relief notch added, you can scroll down to the 'Parameters relief' section and set the relief factor to '0' and then set the 'use relief factor' setting to 'true'. This option might be useful if you are using the Sheet Metal workbench to design using different materials, such as thin card or plastics.

Before we move on, please also note that, in the 'bend' object dialog, you can change the length of the added bend part. Before we add more bends, undo or delete any changes you have made so that we have our simple BaseBend, and our first bend object that is full length with no gaps, etc.

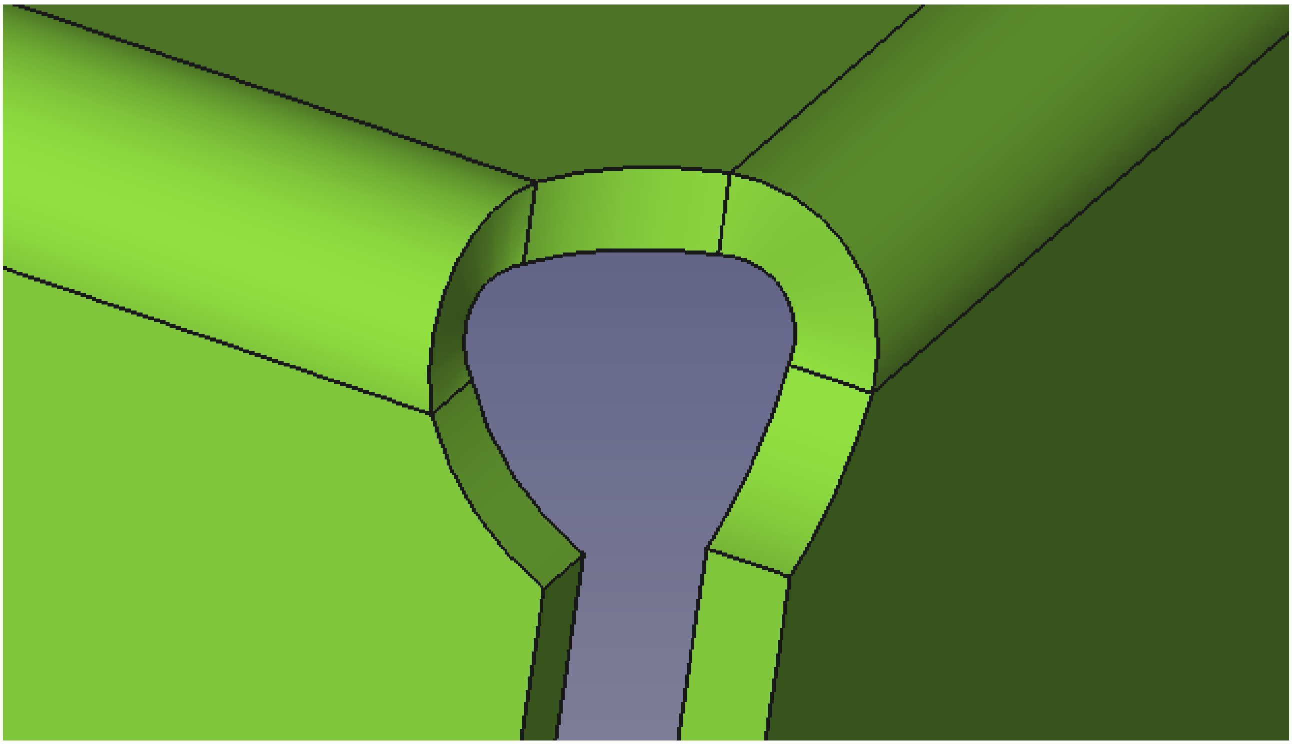
Next, let's select one of the short edges of our BaseBend rectangle sheet and add a second bend to create a wall at 90 degrees. You'll notice that at the corner where the two folds meet, there is a small gap. This is a by-product of working in sheet metal, where the radius of a fold means that folded sections stand slightly off the BaseBend object. You also may note that the two folds join together at a very sharp point where they join the BaseBend object. Again, in real life, that sharp convergence point could create distortions in the piece as it is folded, so it can be a good idea to add some corner relief. Corner relief is a small cutaway that, similar to the earlier notches, stops the metal distorting when folded. To add corner relief, we need to select the two edges which run into the corner on the BaseBend object. You can see these edges selected.



With the two lines selected, the 'Corner Relief to sheet metal corner' tool icon should become active.

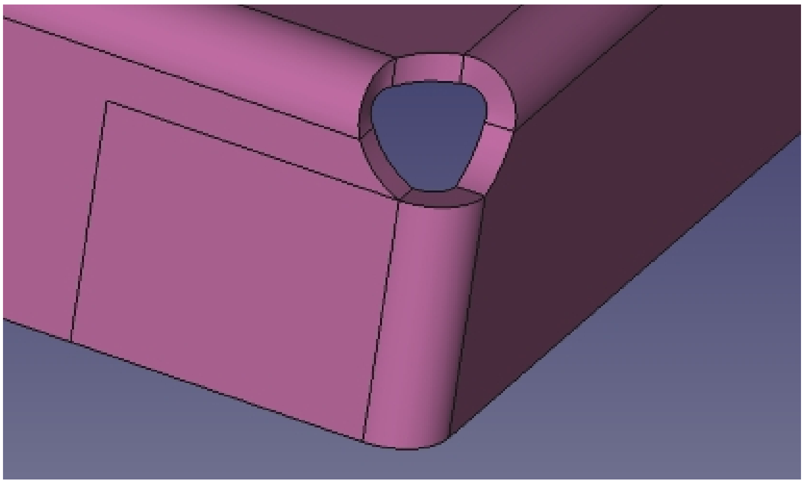


Clicking the tool should create a corner relief object in the file tree, and also create a circular hole in the corner of the two folds.



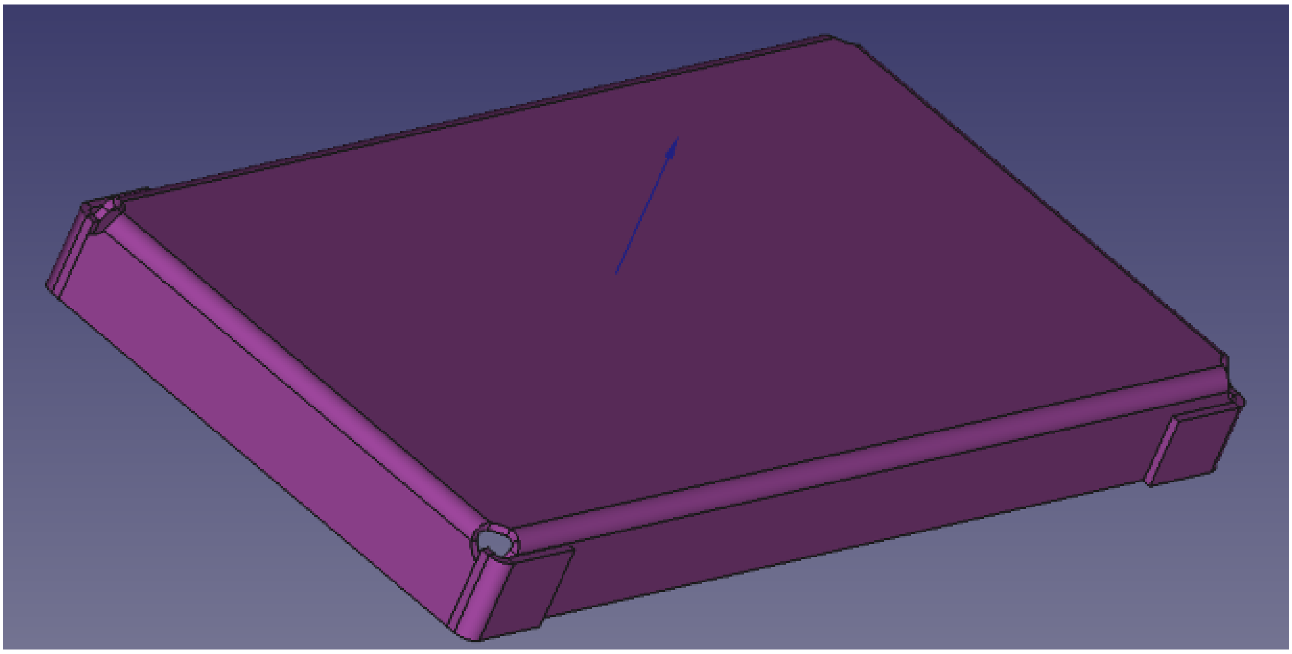
There are options for other corner relief geometries in the corner relief object dialog. Whilst a circular relief might be easy to create in a flat sheet design using a drill, you might prefer a square relief or another shape. Finally, there are also options for ‘circle scaled’ and ‘square scaled’ geometries. These are useful as, if we use a scaled corner relief but then add other folds around that corner, as we are about to do, the scaled relief is sized so that it won’t interfere with the subsequent fold. Therefore, after experimenting, set the corner relief to ‘circle scaled’.

As you can see, we automatically appear to be making a tiny metal tray or lid! We might in real life want to create folds around the corners of the side walls to create a very rigid structure. If we select one of the outer edges of the shorter sides folded wall and click the ‘Extends one or more faces’ tool again, another bend object appears and we now have a folded metal piece that completes the corner.

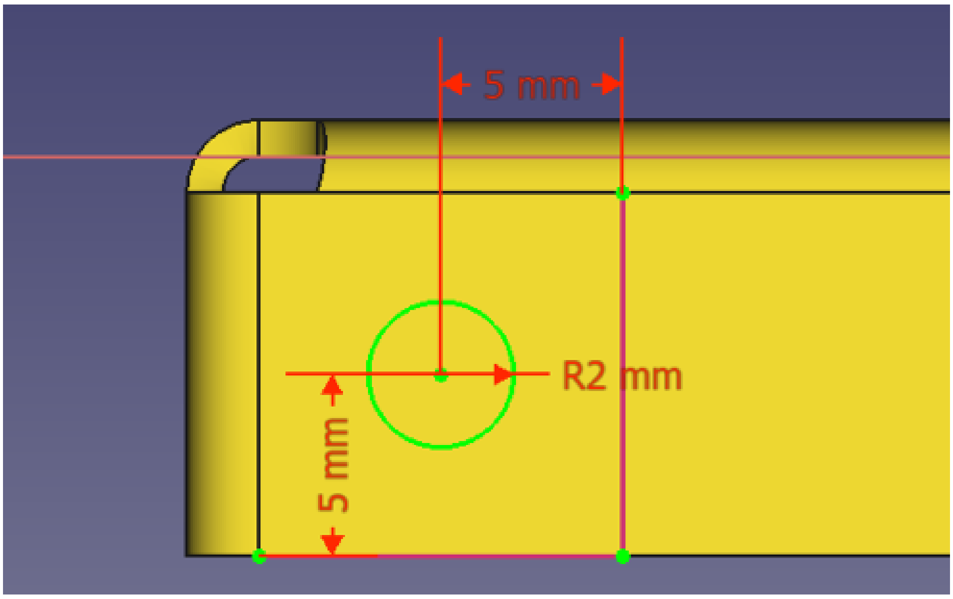


However, this piece currently sits inside/on top of the other folded wall section. Of course, this would be impossible to make, so we need to use an offset to place this fold correctly.

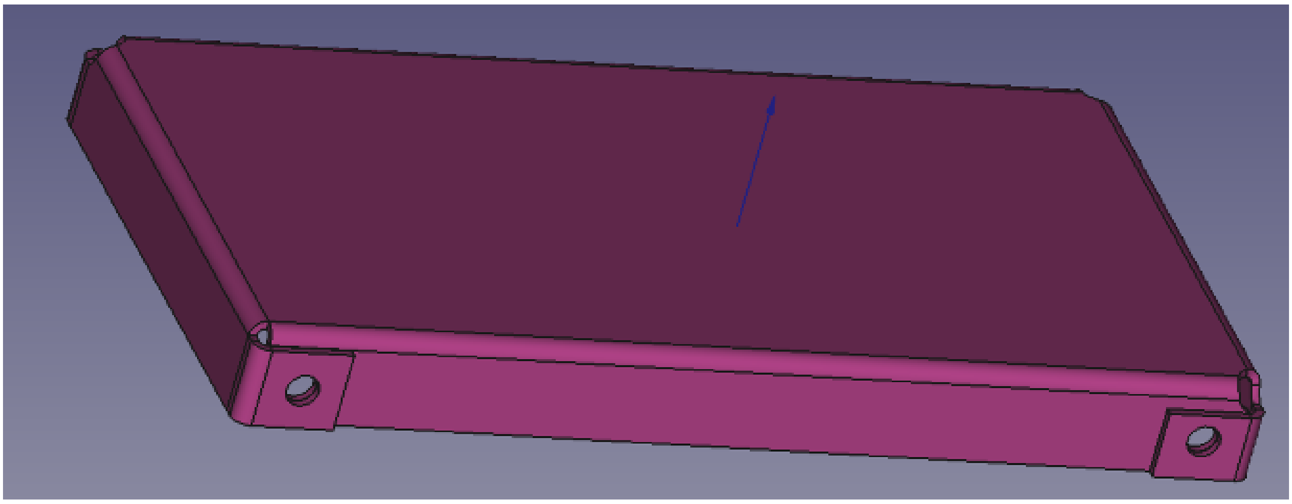
Add An Offset
Highlight the Bend object in the file tree and, in the dialog, click the drop-down menu for the 'Bend Type' option. Select 'Offset', and now the new fold will be offset by the amount specified by the 'offset' field further down the dialog. Our material is 1mm thick, so if we change the offset amount to 1mm, the new section is on the outside of the two tray walls.
Or, if we input -1mm, then the fold sits on the inside of the walls. Whilst this works, it does mean that our material touches inside the folded overlapping sections. This becomes an issue later in the process when we want to flatten our design, as it requires a clearance between folded surfaces. Therefore, set your offsets to 1.1mm to create external folds with a small clearance. Having created all the parts to make one corner of our metal tray, we can continue around the BaseBend object, adding further folds and corner reliefs until we have four walls with folded corners.



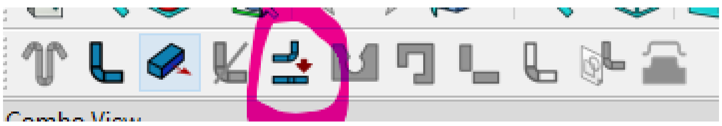
One interesting thing to note about designs made this way on the Sheet Metal workbench is that they can still be manipulated and worked on by tools on other workbenches as normal. One quick example is to move to the Part Design workbench, select a suitable edge, and apply a chamfer or a fillet. A more useful example is to perhaps add sketches to our metal tray design. If we were to fabricate this tray, we might like to add holes in the design for bolts or rivets to fix the folded corners firmly. To add a sketch to this design, you work exactly as if you were adding a sketch to any face in any design. First, move to the Part Design workbench and select one of the external folded tab faces and click the 'Create a sketch' button. We then drew a 2mm radius circle, imagining we might use a 4mm diameter rivet. Again, just like any sketch on a face, we used the 'Create an edge linked to an external geometry' tool to be able to position our circle relative to the end of the folded tab.



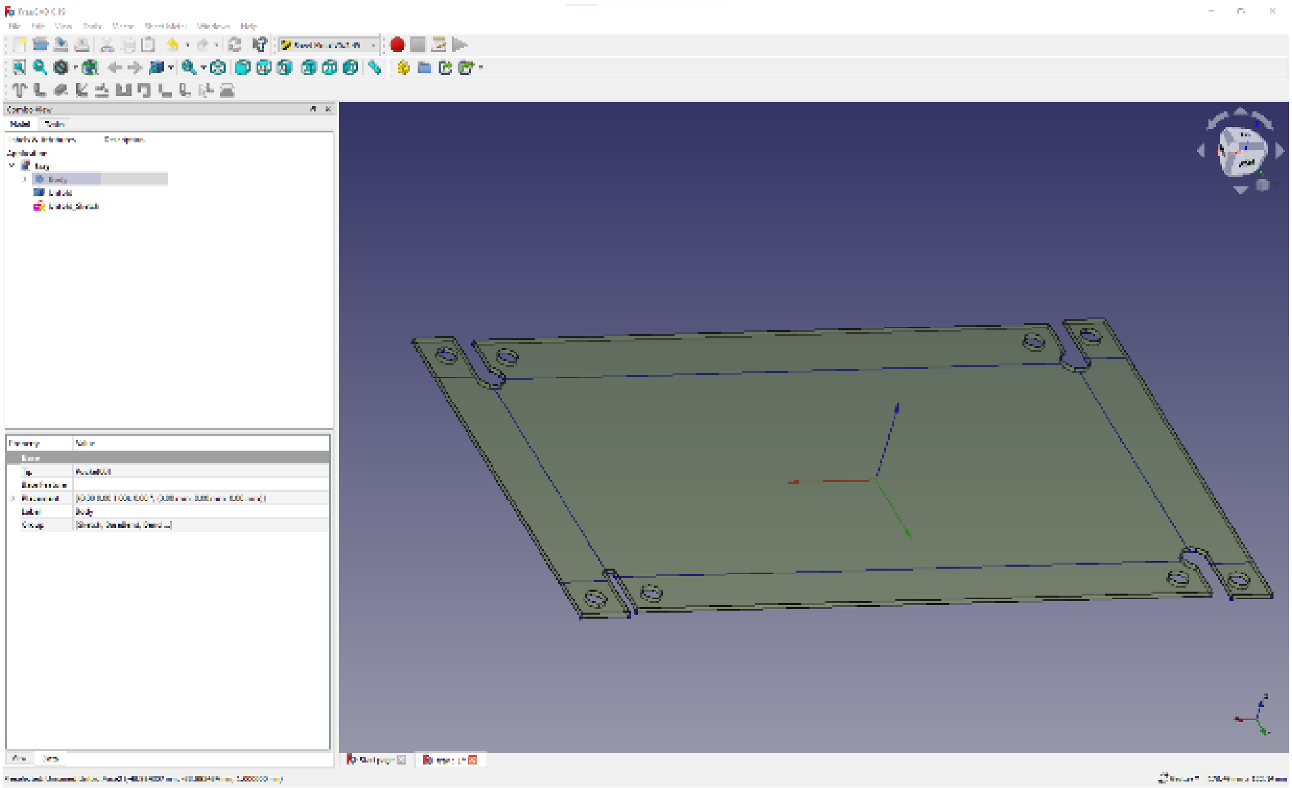
Closing the sketch, we then used the pocket tool on the Part Design workbench to create a hole, and selected 'through all' as the type so that the holes would be created on both sides of the design. Whilst we could have done this in a single sketch, we added another sketch to create the same holes at the other end of our tray.



To fabricate an object, such as our tray created from a folded sheet, we need to be able to flatten our design back out after we have completed it.
Flattening and unfolding the design then allows us to see the basic shape of the flat sheet that we need to cut to create our object. To flatten the design on the Sheet Metal workbench, we need to select a face which will be the reference plane to flatten the part to. With our metal tray design, the obvious choice is to pick the outer large surface of the BaseBend object. With that face highlighted, you then need to click the 'Flatten sheet metal folded object' tool icon, which is fifth from the left on the Sheet Metal workbench toolbar. You should now see a dialog box appear, titled 'unfold sheet metal object'.

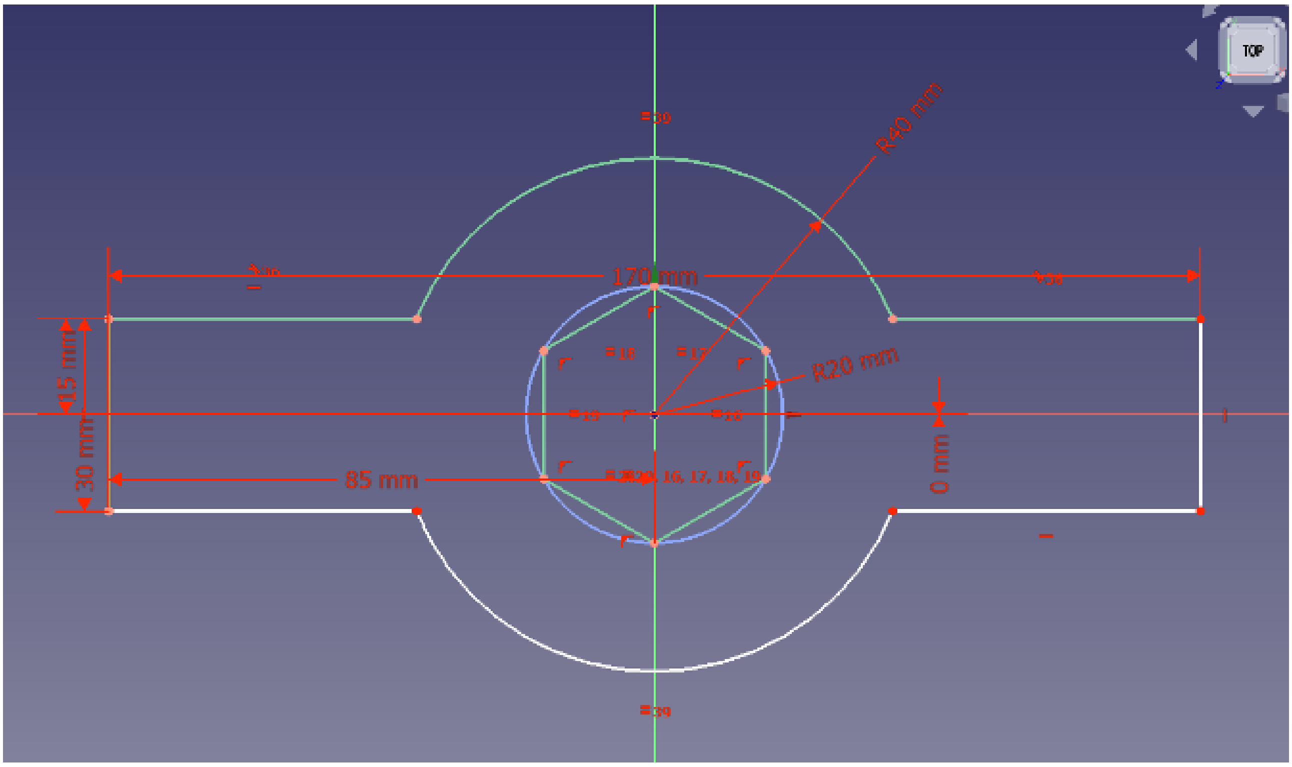


You may also see that the report window opens at the bottom of the screen with a report that 'engineering mode is not enabled'. Don't worry about the report window message: just close the report window. In the dialog box, make sure that 'generate projection sketch' is ticked and then also tick the 'Manual K-Factor' box. When you tick that box, you will need to input a K-factor value. We read that an 0.4 ANSI value was appropriate for our 1mm thick sheet metal design when working in aluminium.
Having input 0.4 and ticked 'ANSI', we can then click the 'OK' button. Depending on your computer, it may take a few moments but, once processed, you should now see two new objects in the file tree view – 'unfold' and 'unfold sketch'. These two new items exist outside of the active body containing our folded design. So, to tidy the file tree view, we can close the drop-down menu of the active body so that our file tree just has three visible list items: 'Body' and the two new unfolded items. You can also see in the preview window our two new items: a blue line sketch and a semi-transparent unfolded sheet of our design. Toggling off the Body item, using the space bar, in the file tree means we can inspect these items more clearly. Of course, like most objects, we can change the appearance and position of these items.

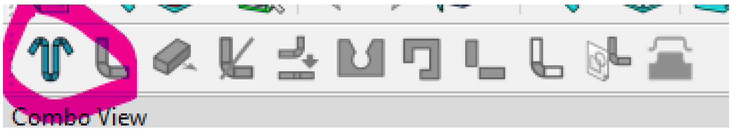


Numerous Uses
We can use these items in numerous ways. For example, we might use the 'Unfold' object to be the basis of an engineering drawing by pushing views of the object through to the TechDraw workbench and adding dimensions and coordinates. This is useful if you needed to traditionally mark or scribe a layout onto sheet metal. You could also make only the sketch item visible, and then highlight it and use File > Export to export a variety of file types (DXF, SVG, and many more) to create printable plans, or to use to create G-codes for a milling machine or to cut on a laser cutter.
Whilst there is a lot of mileage in what we have looked at so far, there is the fact that everything we designed in our tray example was a fold that started or extended from an existing edge of an object.

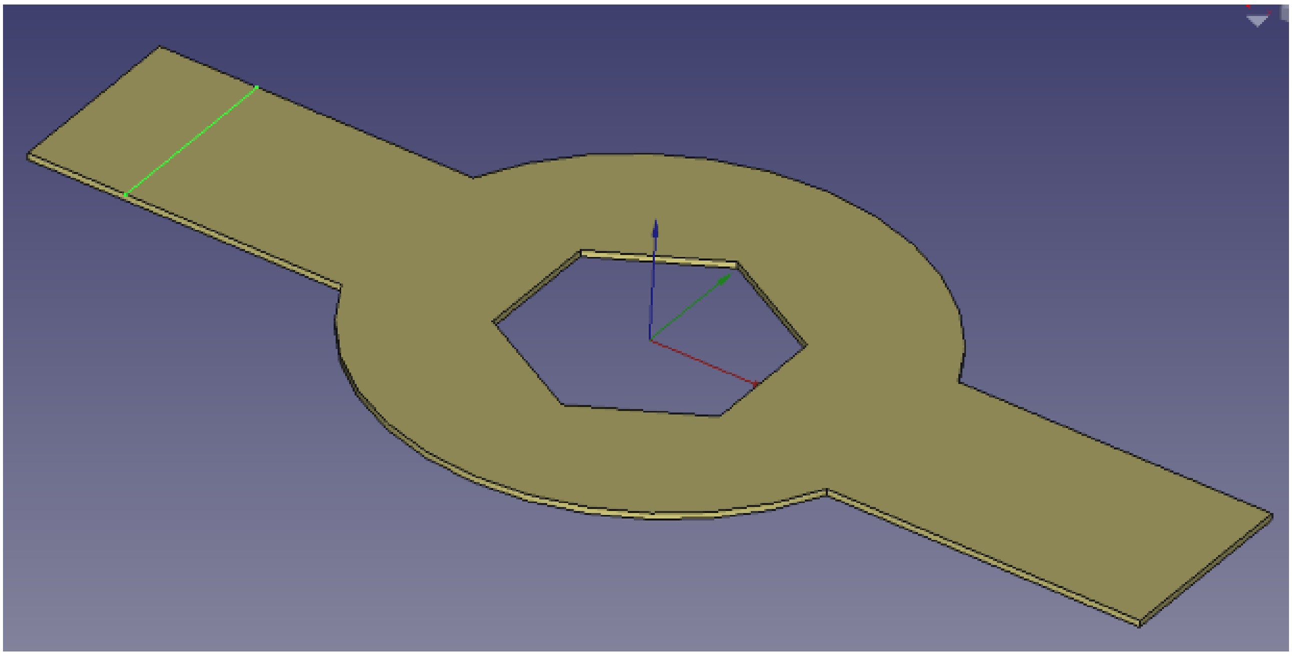
Let's look at using sketches to create fold lines that are not on the edge of items but inside the face of a sheet. To begin, we create a new project, move to the Part Design workbench, and create a sketch in the XY plane. In the sketch, we draw a similar object.



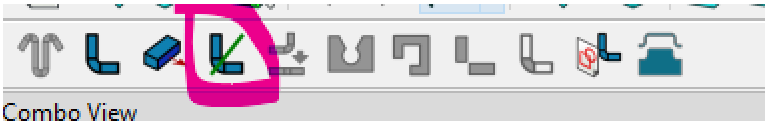
Moving back to the Sheet Metal workbench for a moment, we selected our sketch and clicked the 'Create a sheet metal wall from a sketch' tool icon again to create a basic BaseBend object.



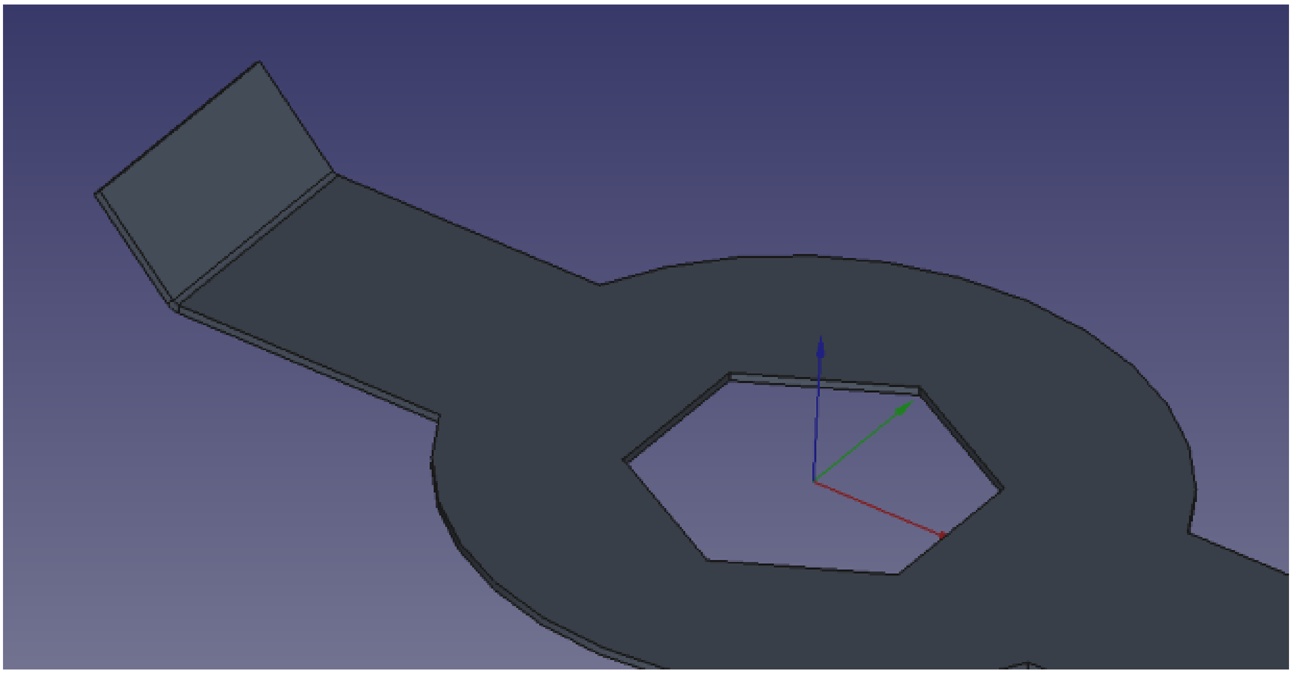
Notice that, similar to padding or extruding a sketch, this tool acknowledges that internal geometry is going to be a void.
Next, we want to create some further sketches that will act as folding lines for our project. Let's aim to fold each side wing twice at 45 degrees. Move again to the Part Design workbench, highlight the upper face of our BaseBend object, and create a sketch on it.
For our first fold sketch, we drew a vertical line across one of our wings 15mm in from the edge, which we imported using the 'Create a linked edge' tool. Once this single line was set up, we closed the sketch and returned to the Sheet Metal workbench.



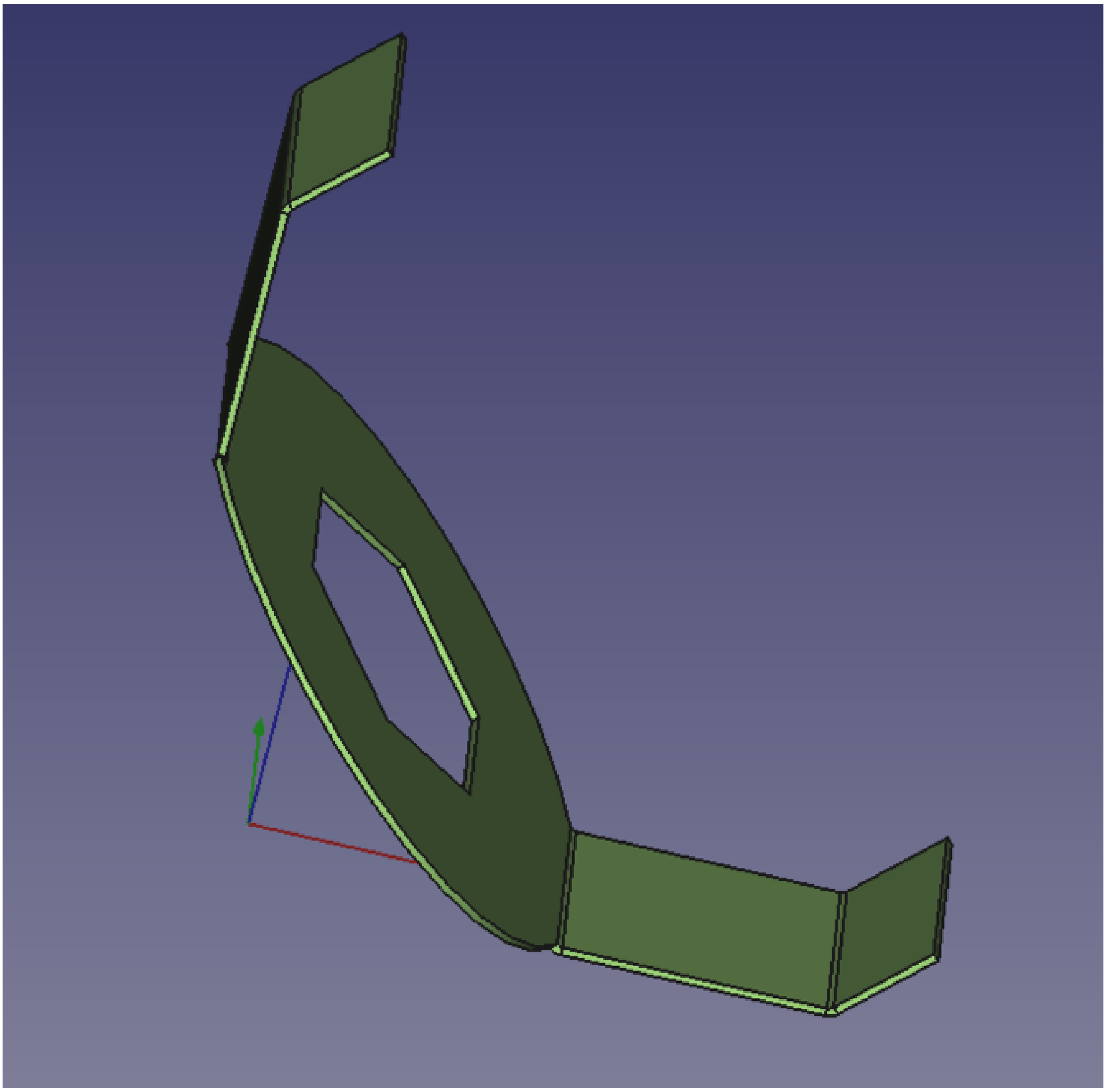
To create this fold, you first need to select the face where the sketch is attached on the BaseBend object in the preview window. This feels a little counter-intuitive and we naturally would select the whole BaseBend object in the file tree, but this doesn't work. With the correct face selected, hold the CTRL-key and click the sketch in the file tree. With these two items highlighted, you should now see that the 'Fold a wall of metal sheet' tool becomes visible.



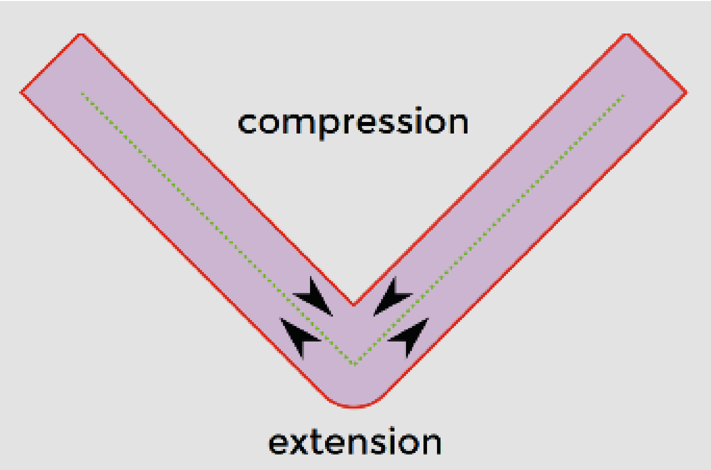
Click this tool icon and you should see a 90-degree bend appear in your part at the sketched line, a 'fold' item appears in the file tree, and the sketch should be toggled to not visible. Highlighting the fold item, you can then use the dialog to make changes – we set our fold to 45 degrees.



Summing Up
For our next fold, we again used the Part Design and the Sketcher workbench to create a sketch containing a line that went across the 'wing' at the point it connected with the circular centre section of our object. Again, we created edges linked to external geometry to be able to draw the line precisely at those points. Moving back to the Sheet Metal workbench, we again selected the upper face of the object and then selected our sketch containing the fold line. Applying the 'Fold a wall of sheet metal' tool, we again set the fold angle to 45 degrees and, this time, we also changed the 'position' to 'backwards' as this meant the radius of the fold lay outside the flat area of the circle rather than inside it.



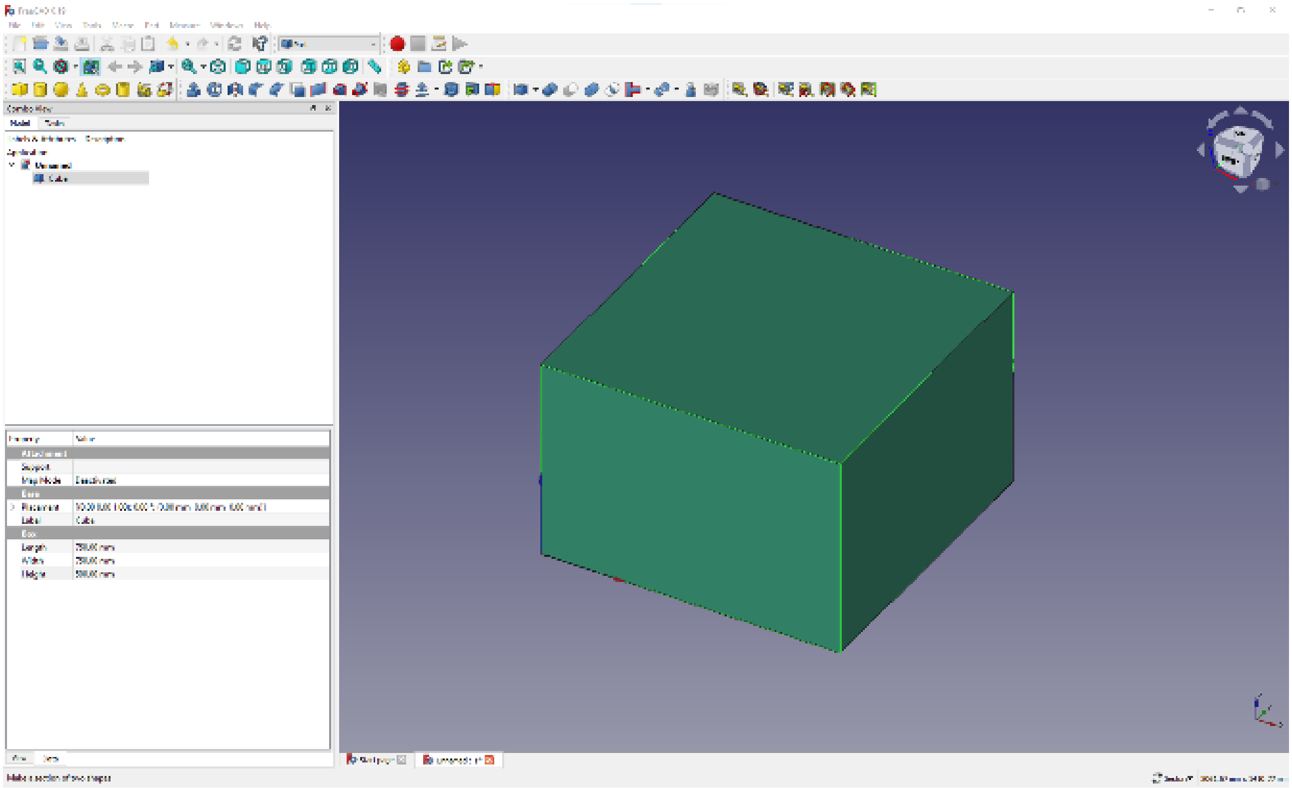
Fold Geometry
When you fold sheet materials, including sheet metals, there are numerous forces in play inside the fold. On the inside of the fold, the material is under a compression force and, on the outside of the fold, the material is being expanded. In the diagram below, the dotted green line represents the line through the material where these forces swap from compression to extension and vice versa. This line is called the neutral axis. In different materials and at different thicknesses of materials, this line will be in different positions. The reason this is important is that it dictates where fold lines are accurately placed to create the overall dimension of a piece accurately. The position of the neutral axis is given as a ratio of the material's thickness; this ratio value is called the K-factor. You can search online for the K-factor of a given material at a given thickness, but also be aware of what industrial standard the K-factor value is being stated in, either ANSI or DIN. We found that a K-factor ratio of 0.4 ANSI was commonly given for 1mm aluminium sheet. Using this value when using the 'Flatten sheet metal folded object' tool will ensure that all your fold geometry is accurate for the material you wish to use.



Quick Tip
When creating sketches that will act as fold lines, you need to create each line in a separate sketch, as you will need to adjust fold parameters individually.

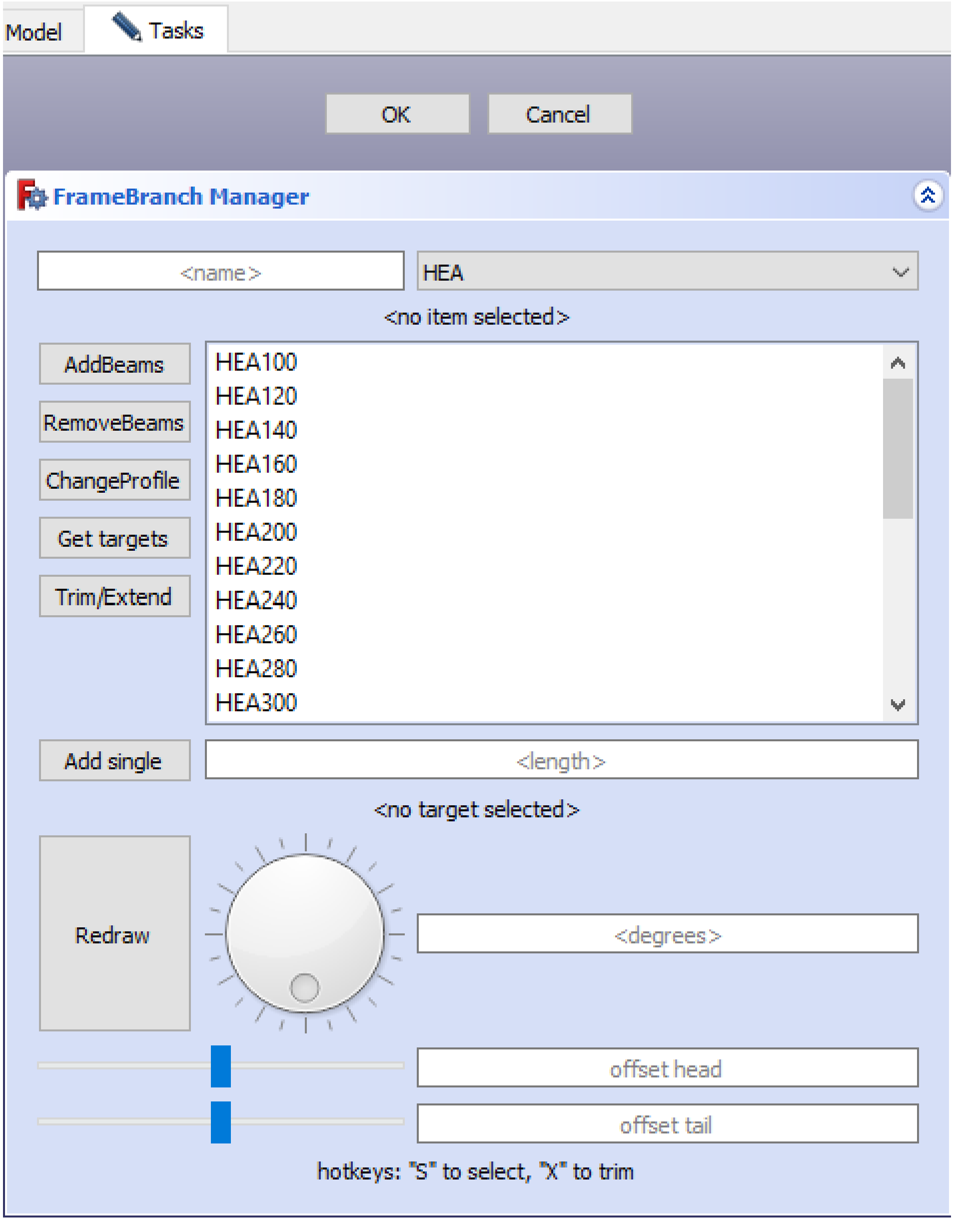
Frames And Pipes

Here we will be looking at the 'Dodo' workbench, which treats working with framed objects and designs or pipework.
To begin, open FreeCAD and click the Tools > Addon manager, then select "Dodo" on the list. Click install. When installed, you will be prompted to restart FreeCAD. Restart FreeCAD and then create a new project. Let's look at making a framed object first.
To begin, move to the Part workbench and use the 'Create a Cube Solid' tool to create a cube that is 750 × 750 × 500 mm.

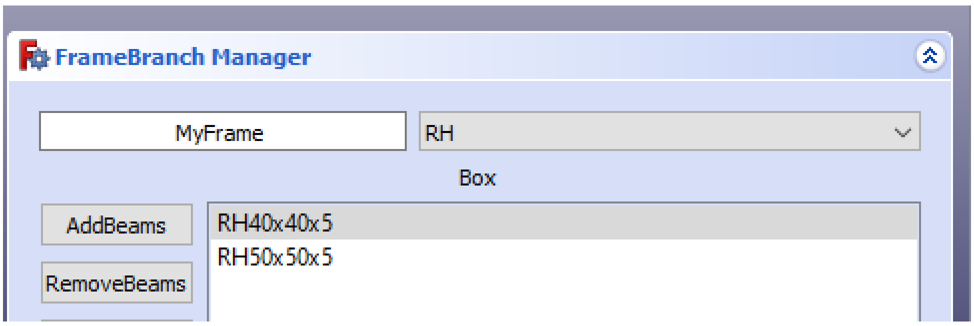


Next, let's move to the Dodo workbench, which is listed as 'Dodo WB' in the workbench drop-down menu. Click the 'Open FrameBranch Manager' tool,

and you should see a dialog box similar to the one shown.

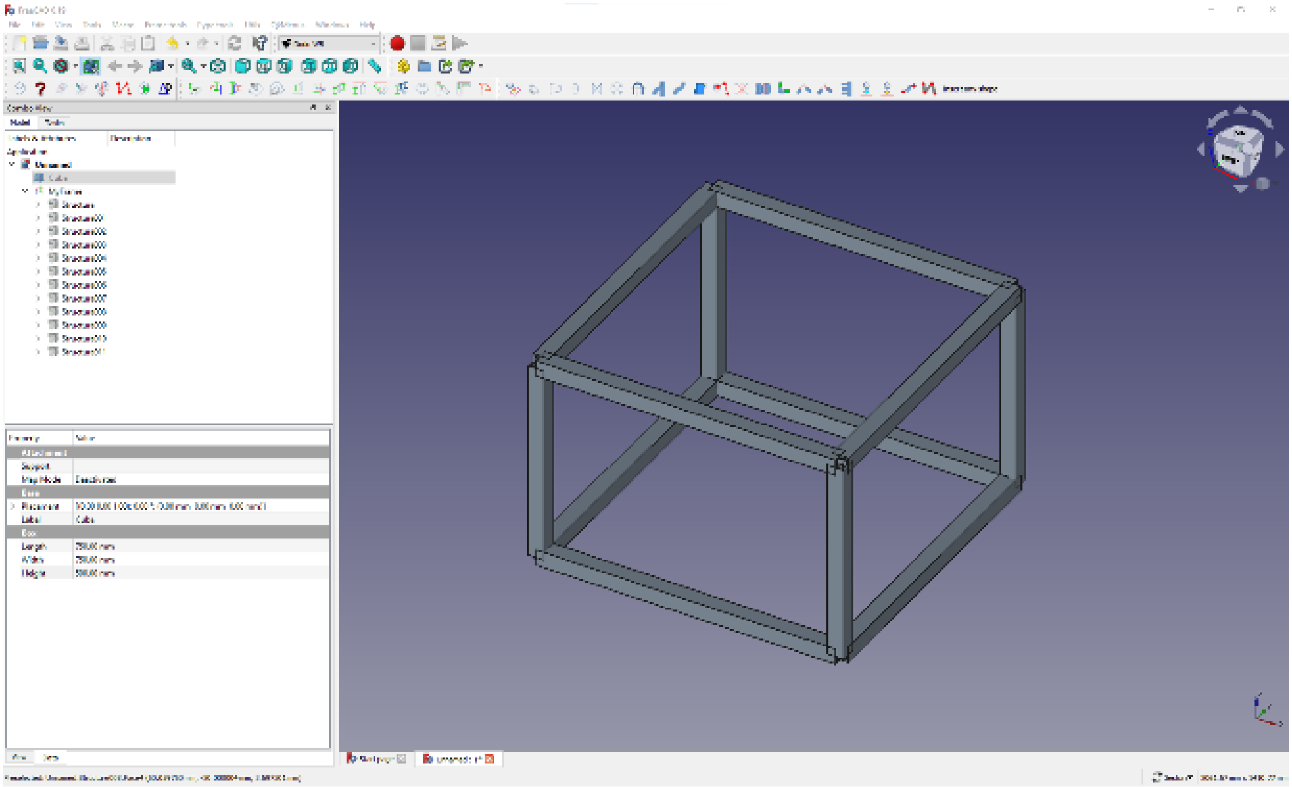


To create a frame, we need to select a profile for the frame extrusions. There's a collection of preset profiles included that model real-world standard metal extrusions. Let's select a simple 40 mm × 40 mm square extrusion with a 5 mm wall thickness. Click the drop-down menu that currently reads 'IPE' or 'HEA' and select 'RH', and then select the 'RH40×40×5' by highlighting it in the list below. Make sure to give it a name Eg: 'MyFrame'



Next, click on any edge of the cube in the preview window and then click 'OK' in the Open FrameBranch Manager dialog box. You should now see that every edge of the cube object has had the 40 mm × 40 mm extrusion added to create a frame.

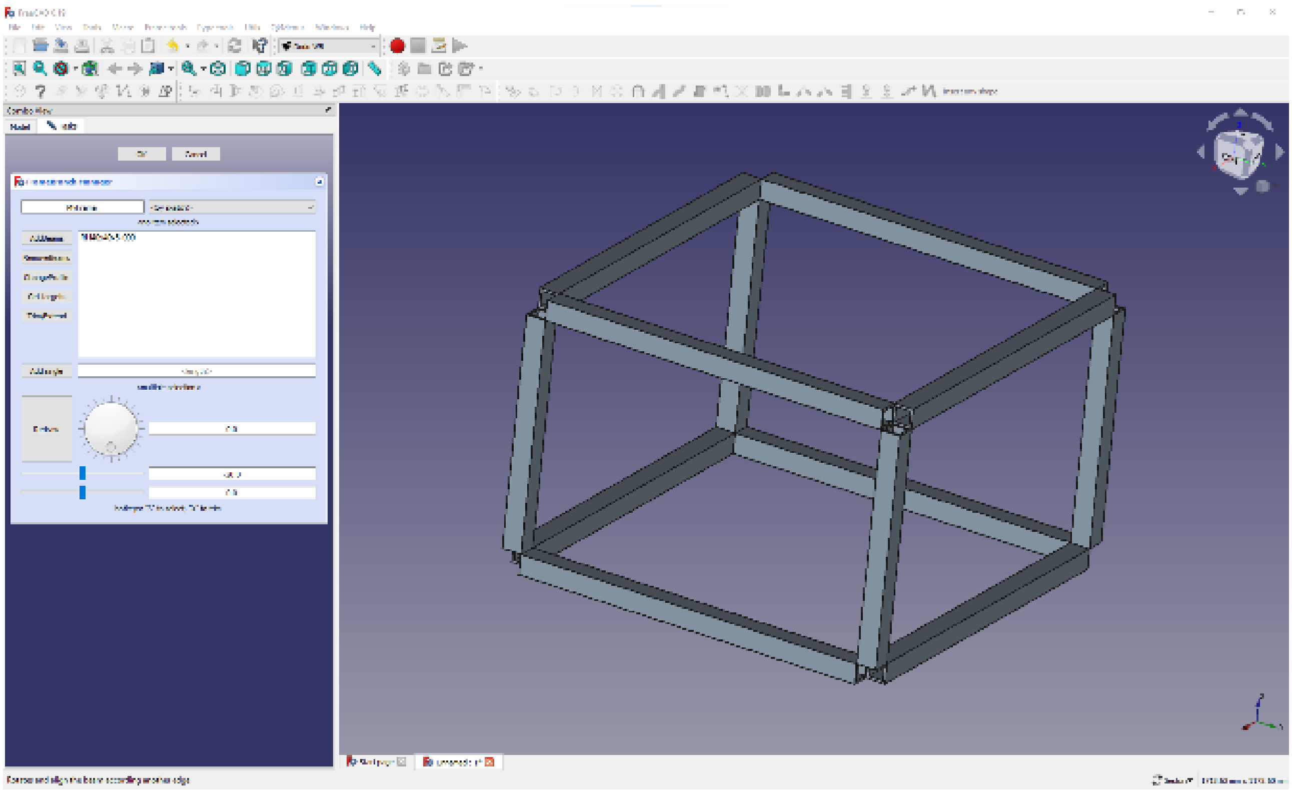
By selecting the ‘Model’ tab, you can, of course, in the file tree view, make the cube object invisible to get a better view of our frame.



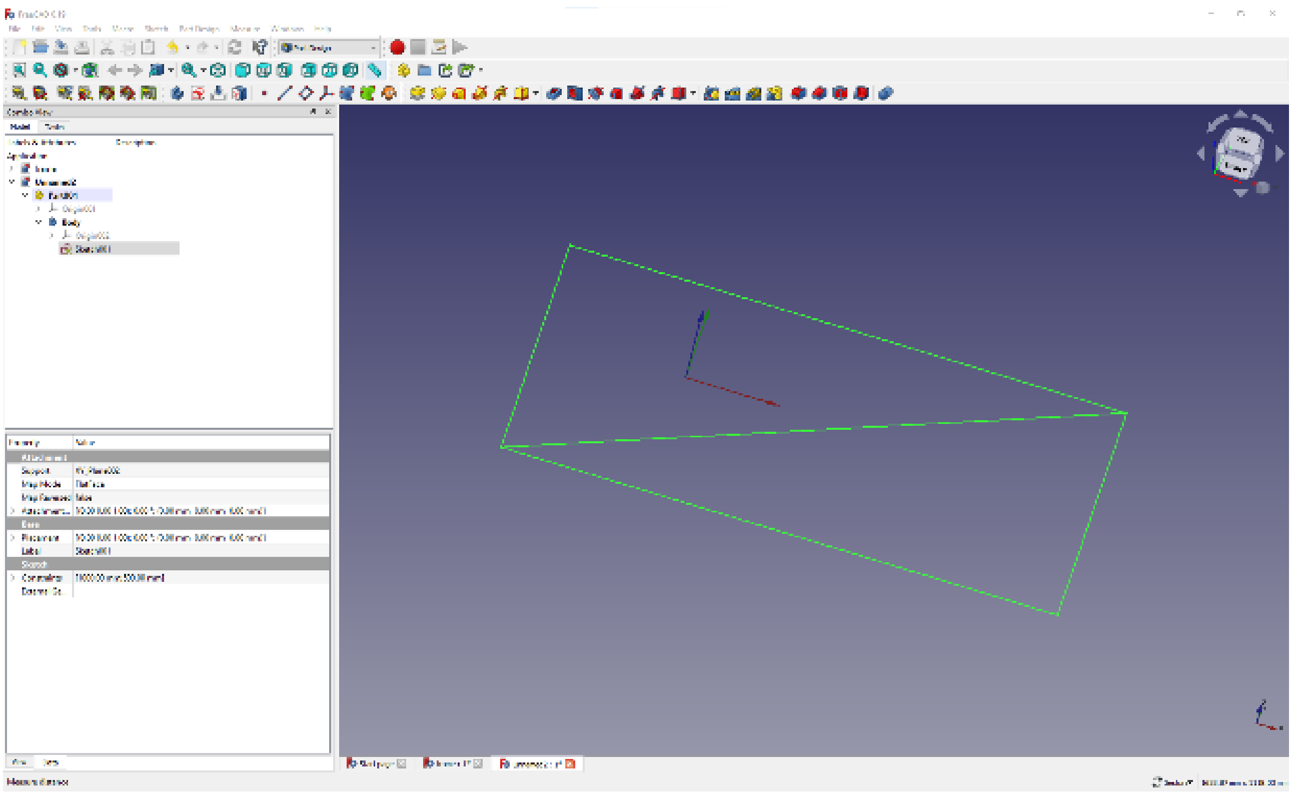
You’ll notice that our frame has all the required sections but that they all overlap at the corners. We need to extend or trim these frame parts to make a frame that could be fabricated in real life. With the Open FrameBranch Manager still open, we can use the ’Get Targets’ and ’Trim/Extend’ buttons to help us do this quickly.

First, select an inside face of one of the frame sections and then, holding down the 'Control'-key, select the opposite inside face – with both of those selected, click the ‘Get Targets’ button. We have now defined the reference as to where we want to trim or extend other frame structure objects.

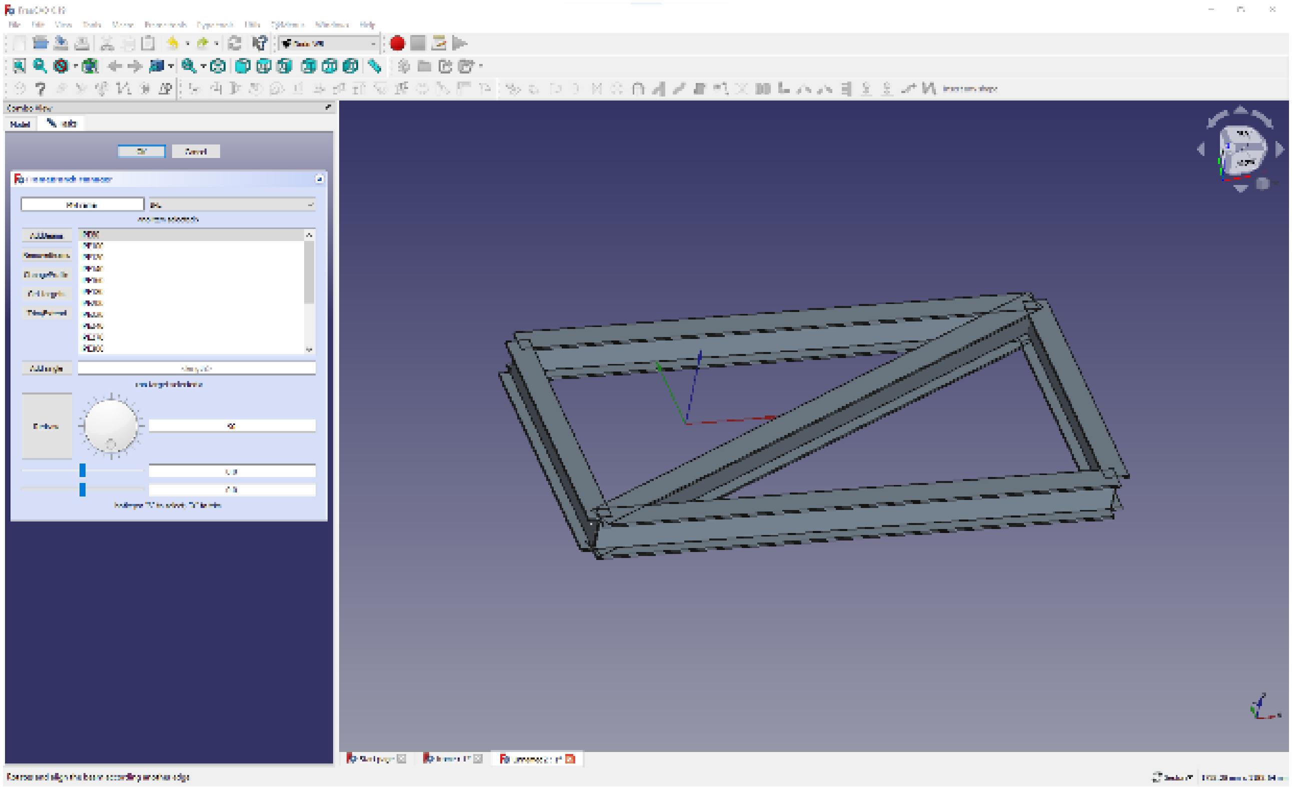
Next, select any face of a frame piece that crosses the target faces that we want to trim to between the target faces. Click the ‘Trim/Extend’ button and you should see that our cross member now is shortened to directly fit inside the other pieces. We don’t need to select new targets to also trim other frame pieces to the same length. Even the piece below the trimmed piece can be trimmed. We can select faces on the three other pieces and click the ‘Trim/Extend’ button to bring them all to length. You can also, when required, select the outside edge faces of objects as targets, and the ‘Trim/Extend’ button will extend pieces to the correct length.



It’s worth noting that although we have used a solid object as the basis of our frame, we can apply frame structures directly to sketches. As a quick example, create a new project and move to the ‘Part Designer’ workbench and create a sketch that has a rectangle with an extra line spanning across the rectangle inside – our rectangle was 1000 mm × 500 mm, in the XY-plane. Note that you might need to click ‘Fits the whole content on the screen’ to view the retangle.

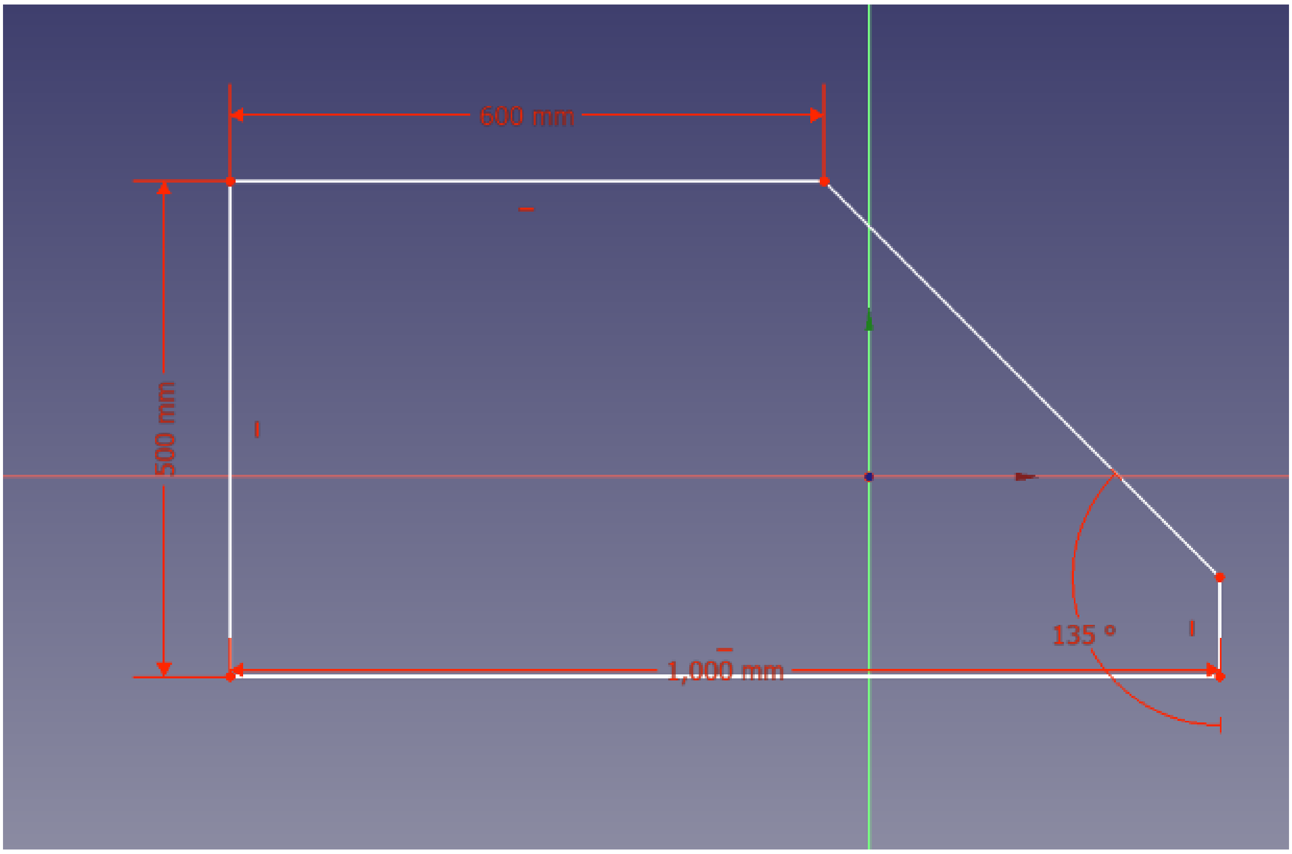


Moving to the “Dodo” workbench again, click to launch the ‘Open FrameBranch Manager’. Let’s select a different profile – we went with ’IPE80’, which is a common steel I-section used in buildings and structure fabrication.
Again, click any edge of the sketch in the preview window and then click ’OK’. You should now see a frame appear made from the I-section.
You might need to turn the beams 90°. Select every beam by clicking any surface of it, using the ‘control’-key and typing 90 in the field next to the dial button.



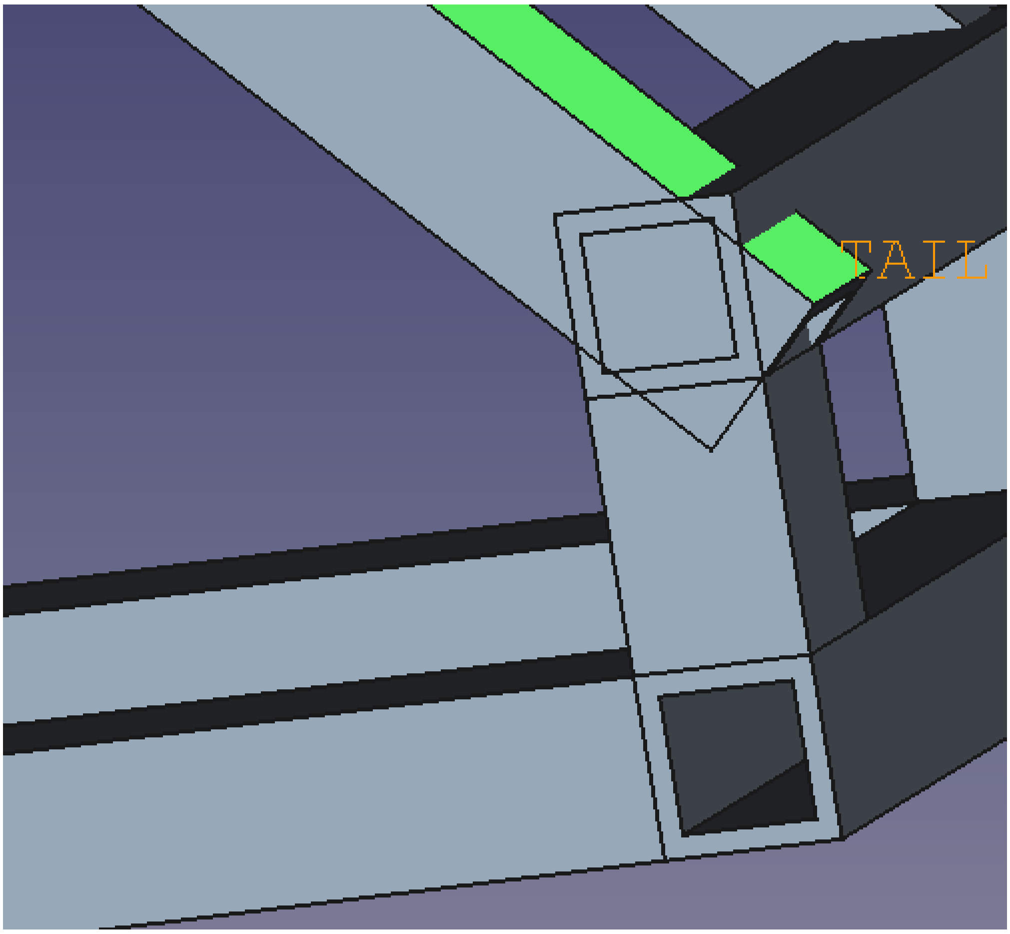
You can, of course, use any face of any of the IPE80 parts to set targets again to trim or extend other sections to.

Interesting Intersections
So far, we have made frames with frame members at 90 degrees to each other. Of course, it's possible to create frame structures around sketches or paths with other angles in them. However, we need to use some workarounds if we want to create accurate and neatly trimmed intersections. As an example, let's create a rectangular object with a large 45-degree plane on one end to create a frame around.
In a new project, open the 'Part Designer' workbench and create a new sketch in the XZ plane. Create a sketch similar to this.



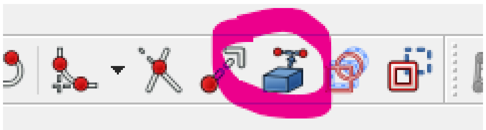
In this example, it's probably useful to fully constrain your sketch so that we know accurately where frame elements will be placed.
Next, pad the sketch to 250 mm and extrude the sketch into a large frameable object.

Moving to the 'Dodo' workbench, once again open the 'FrameBranch Manager' dialog and create a frame on our object in the usual way – we opted again to use the RH 40 × 40 × 5 square box profile. When you initially create the frame, you'll notice that the angled sections need to be extended to fully meet the vertical and horizontal sections. If you then select targets and Trim/Extend, you'll see that the angled section now extends through the other sections with some excess material.

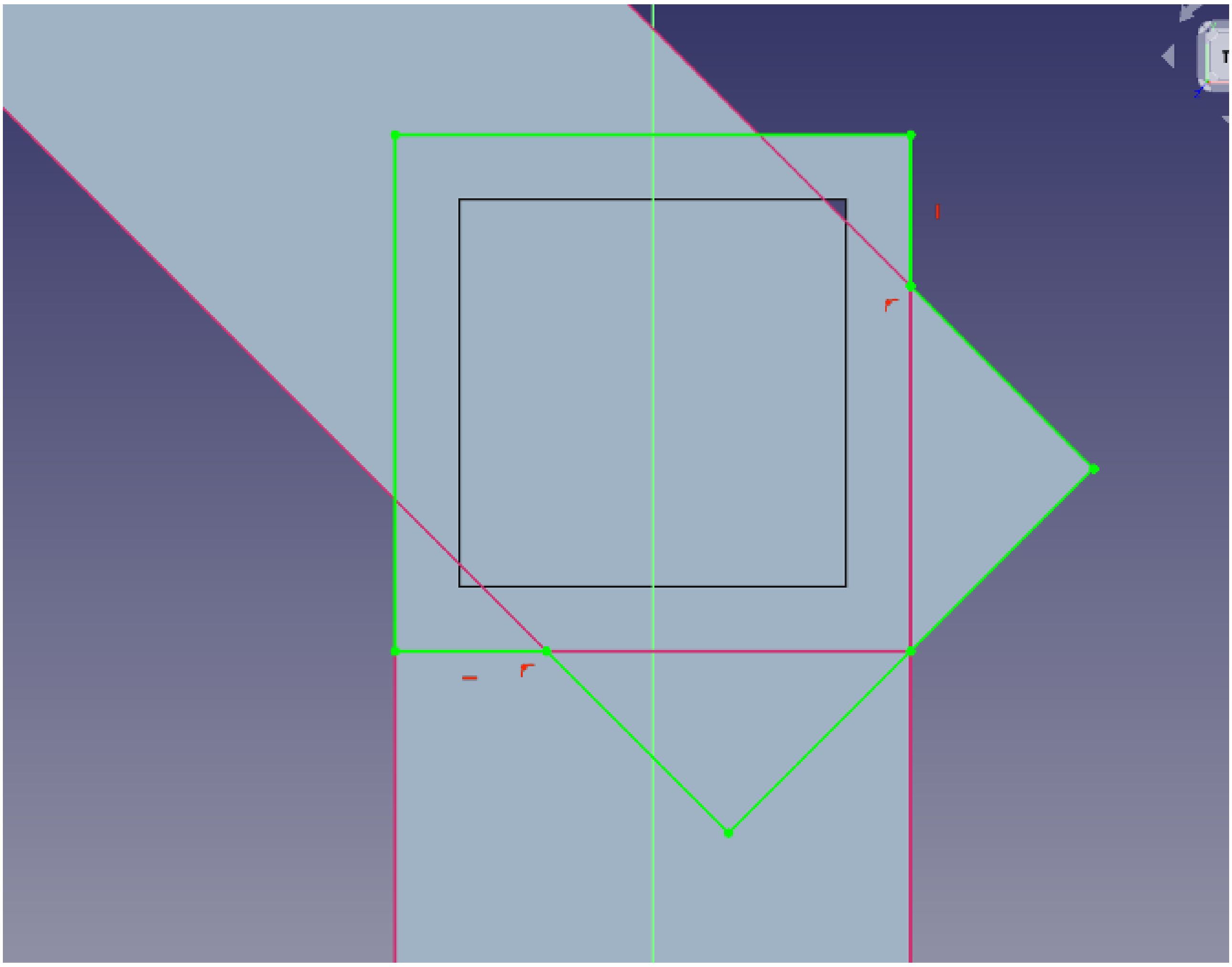


Removing Overlaps
Continuing to 'Trim/Extend', we can get the horizontal and vertical frame members to adjoin correctly, as seen in figure above. Now we are ready for our workaround solution for the angled parts. Basically, the idea is to create parts that cover all the material we want to remove and then, using the 'Part' workbench, we can use 'Boolean cut' operations to remove the unwanted overlap sections. There are, as ever in FreeCAD, numerous ways we could achieve this. We could, for example, create simple solid cuboid objects on the 'Part' workbench, move them into place, and then perform the cut operations.

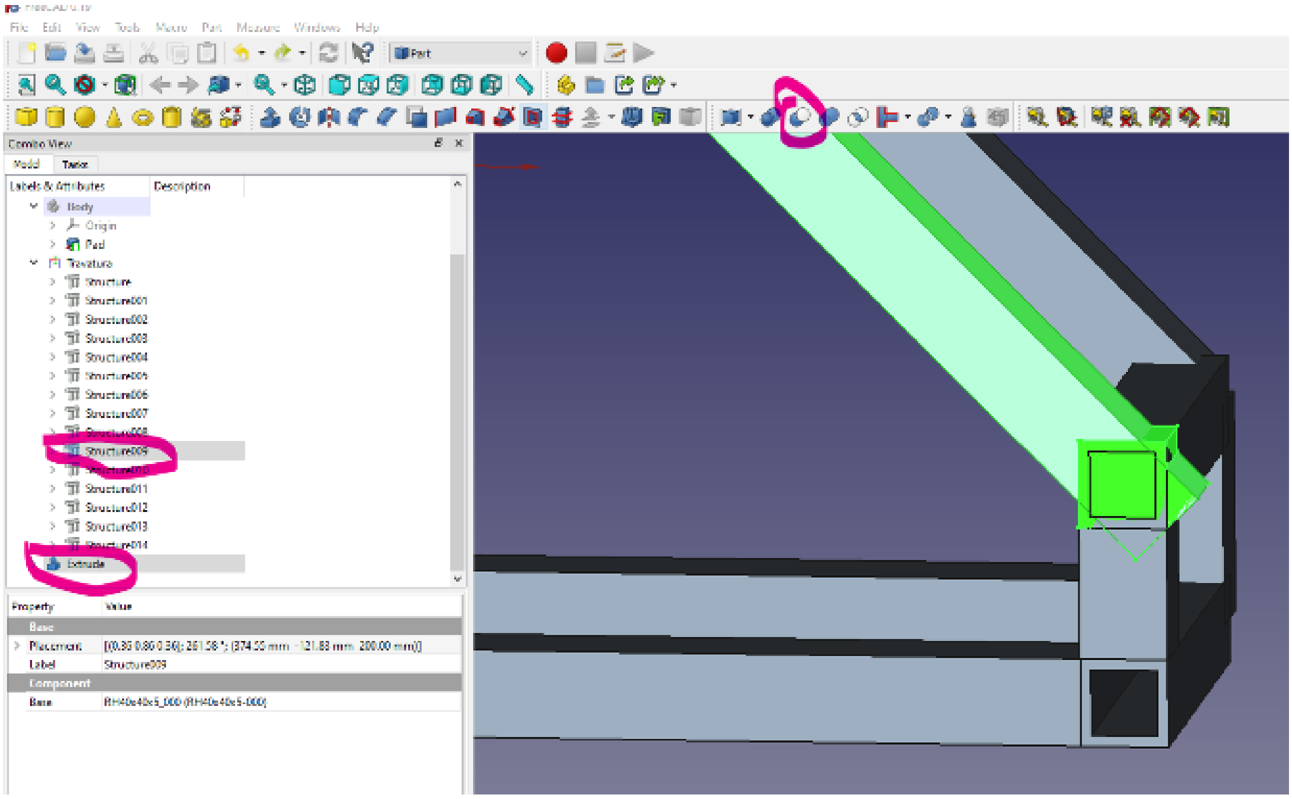
Another, perhaps cleaner, approach is to map a sketch onto the frame to create an extrusion that matches the material we need to remove and then perform the cut operation. Select the outside face of one of the short upright frame sections, then, on the 'Sketcher' workbench, create a sketch.
When prompted, choose 'FlatFace' as the sketch attachment method. In the sketch, use the 'Create an edge linked to external geometry' tool to import any edges and vertices that will help you create your sketch.



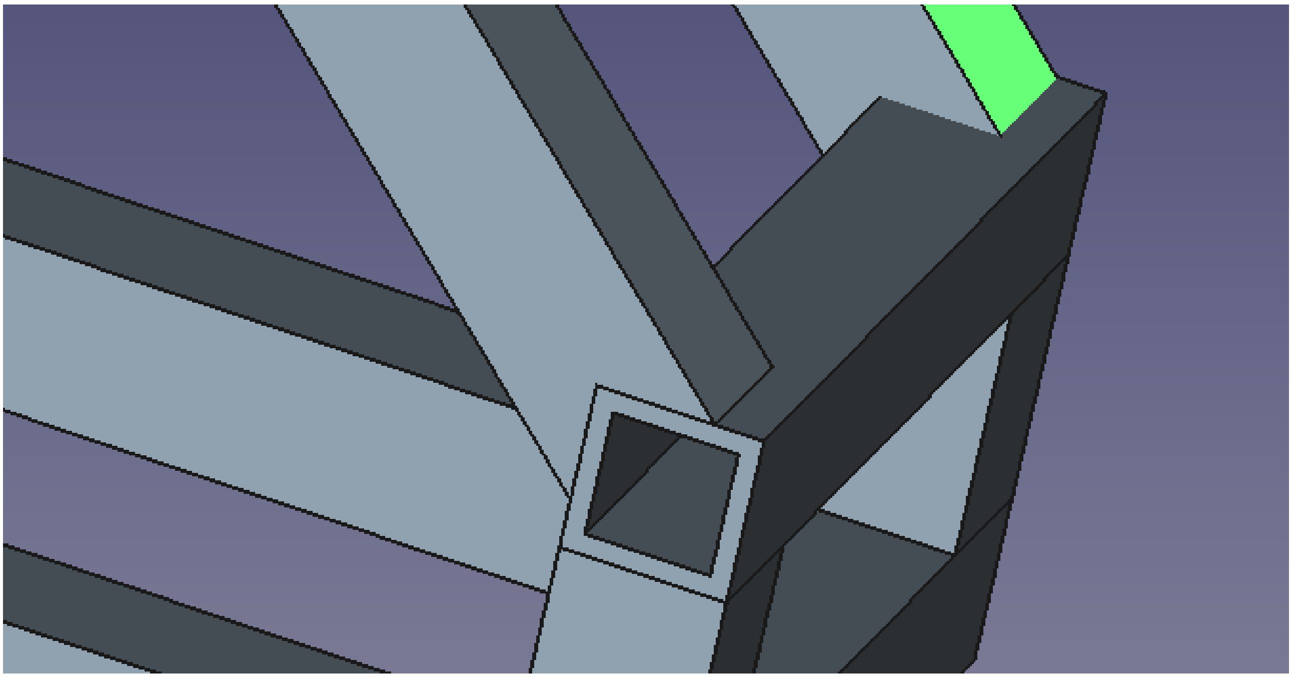
Then, use the Polyline tool to create a sketch that will remove all the material you need to remove from your target frame section.



Next, close the sketch and move to the 'Part' workbench and extrude the sketch you just created. Extrude it in such a way that it moves through the frame element you wish to cut material from. For me that is -40 mm.
You can only perform a cut operation using two parts, so the next step is to select one of the angled frame members from inside the Travatura folder in the file tree and then select the extrusion we just made. Click the 'Make a cut of two shapes' tool.

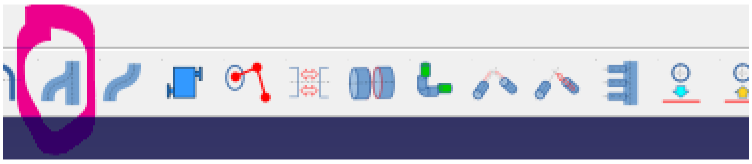


This should perform the correct cut on one of the angled frame members. Next, select the other frame member you wish to cut excess material from and then select our extrusion object, which now can be found inside the ’Cut’ object folder that was just created in the file tree.

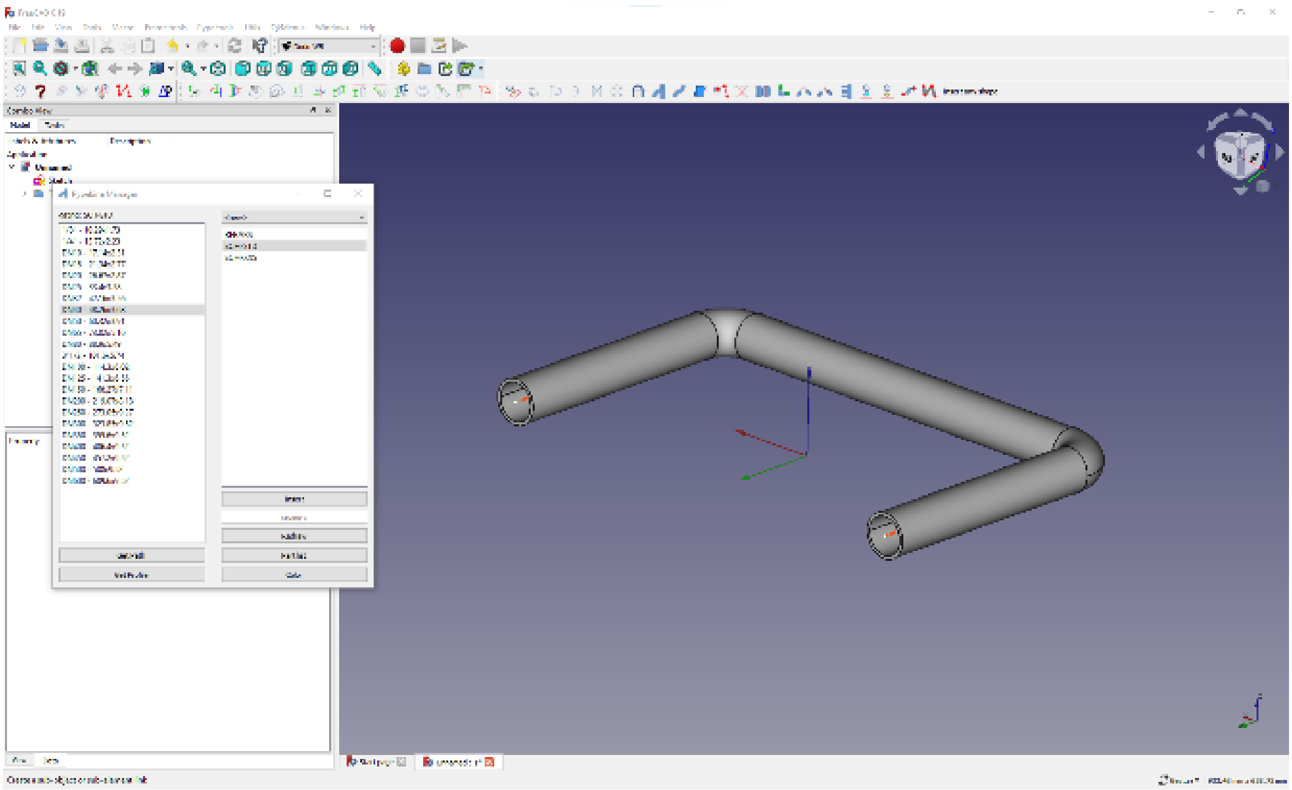


You can see that this has worked perfectly and has given us the frame member arrangement that we wanted to achieve.

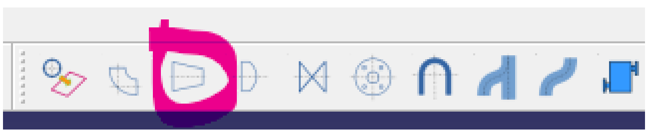
The ‘Dodo’ workbench doesn’t just have excellent tools for creating framed designs, it also has a collection of tools for making pipework designs easier to realise. Like many aspects of FreeCAD, it’s incredibly capable, but we can start with some simple examples that you can build on as you explore the tools.
In a new project, let’s start by making a sketch in the XY plane. We created a simple square U shape using the ‘Polyline’ tool. We made it a reasonable size, 300 mm x 500mm, because, if it’s too small, when we add pipes following the sketch, it might fail if the pipe corners can’t fit in the dimensions of the sketch. It’s worth noting that pipe and frame objects on the ‘Dodo’ workbench can be applied to not just sketches but also wires created using ‘Draft’ tools, or you can simply add pipe objects onto other existing pipe objects.
With a sketch created, move to the ‘Dodo’ workbench and click the ’Open PypeLine Manager’ tool icon.



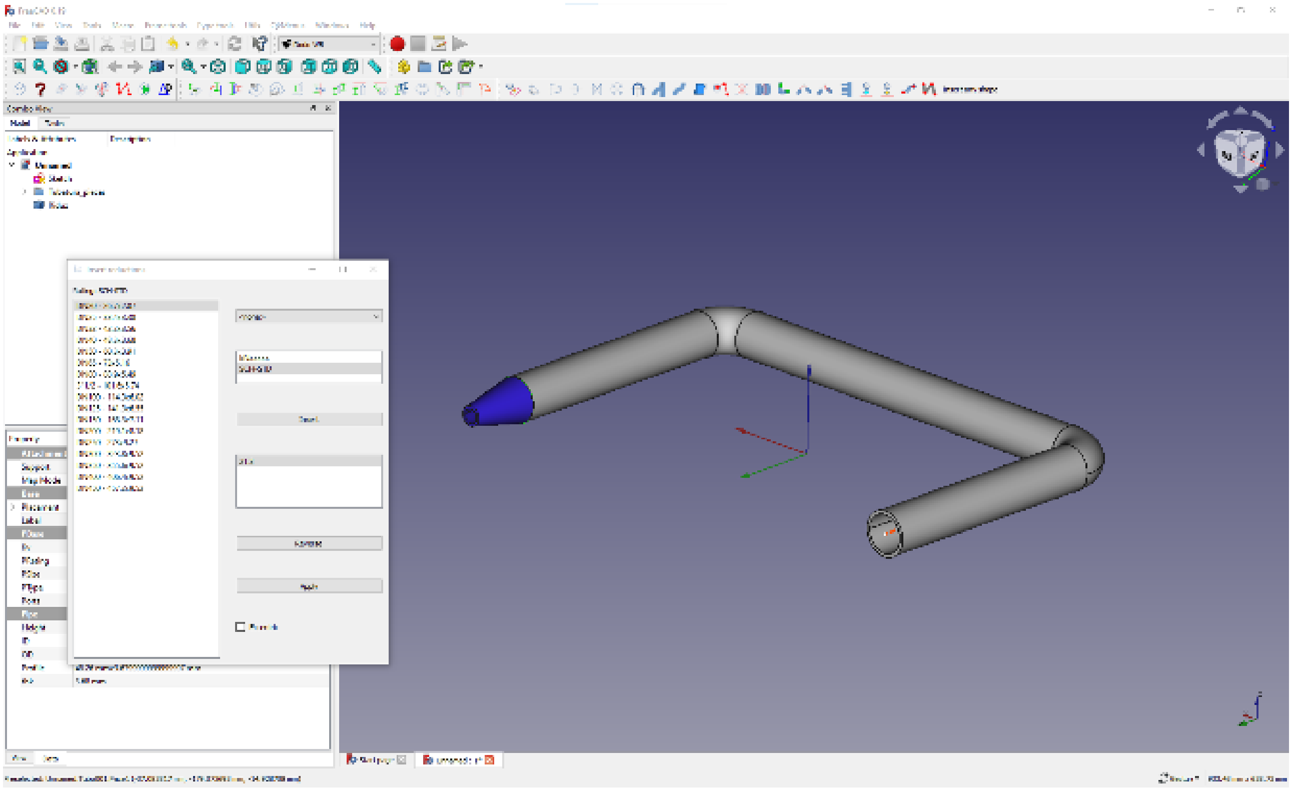
In the ‘PypeLine Manager’ dialog box, there’s a range of standard pipe types to select from – we chose the ’DN40’ pipe from the ’SCH-STD’ group.
This is a common pipe dimension with a 40 mm internal diameter. Next, click any part of the sketch in the preview window and then click the ’Insert’ button in the PypeLine Manager dialog. This should automatically create a pipe that follows the complete sketch path with the 90-degree curved sections added.



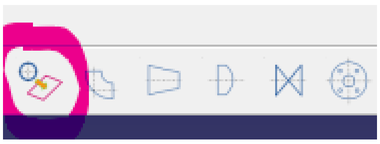
We mentioned earlier that you can add pipe objects and other peripherals by simply attaching them to an existing pipe. There are numerous tools that help us do this. Selecting the outer edge of one end of our U-bend pipe construction, click the ’Insert a Reduction’ tool.



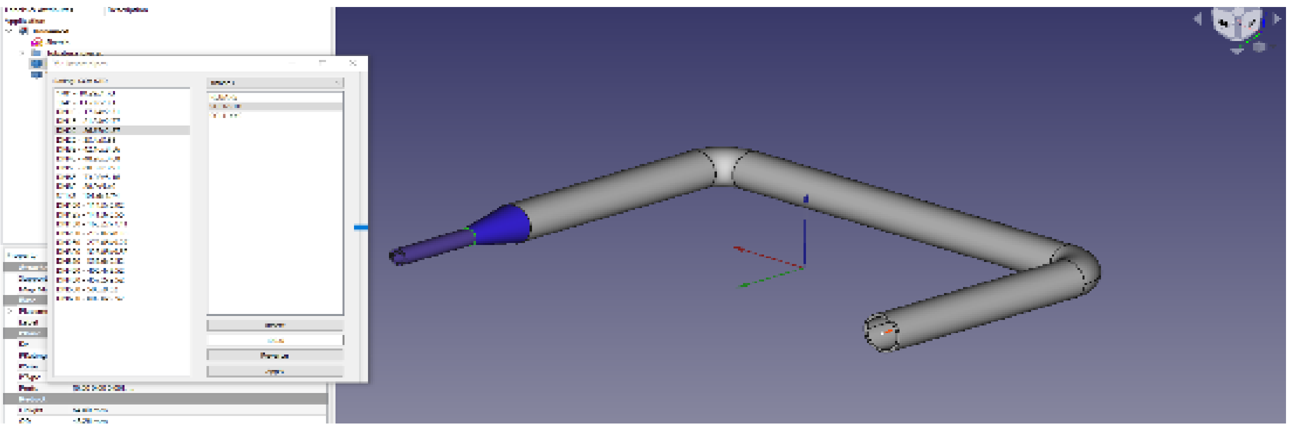
The larger end of the reduction is automatically defined by the diameter of the selected adjoining pipe, and so we simply use the 'Insert Reductions' dialog to select the smaller end of the reduction. You can see we opted for the 'DN20' size, and then clicked the 'Insert' button to add the reduction.



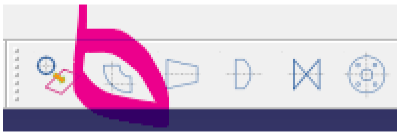
It's as straightforward to add a simple length of straight pipe to an existing design. Again, select the edge of the thin end of the reduction we just added and click the 'Insert a Tube' tool.



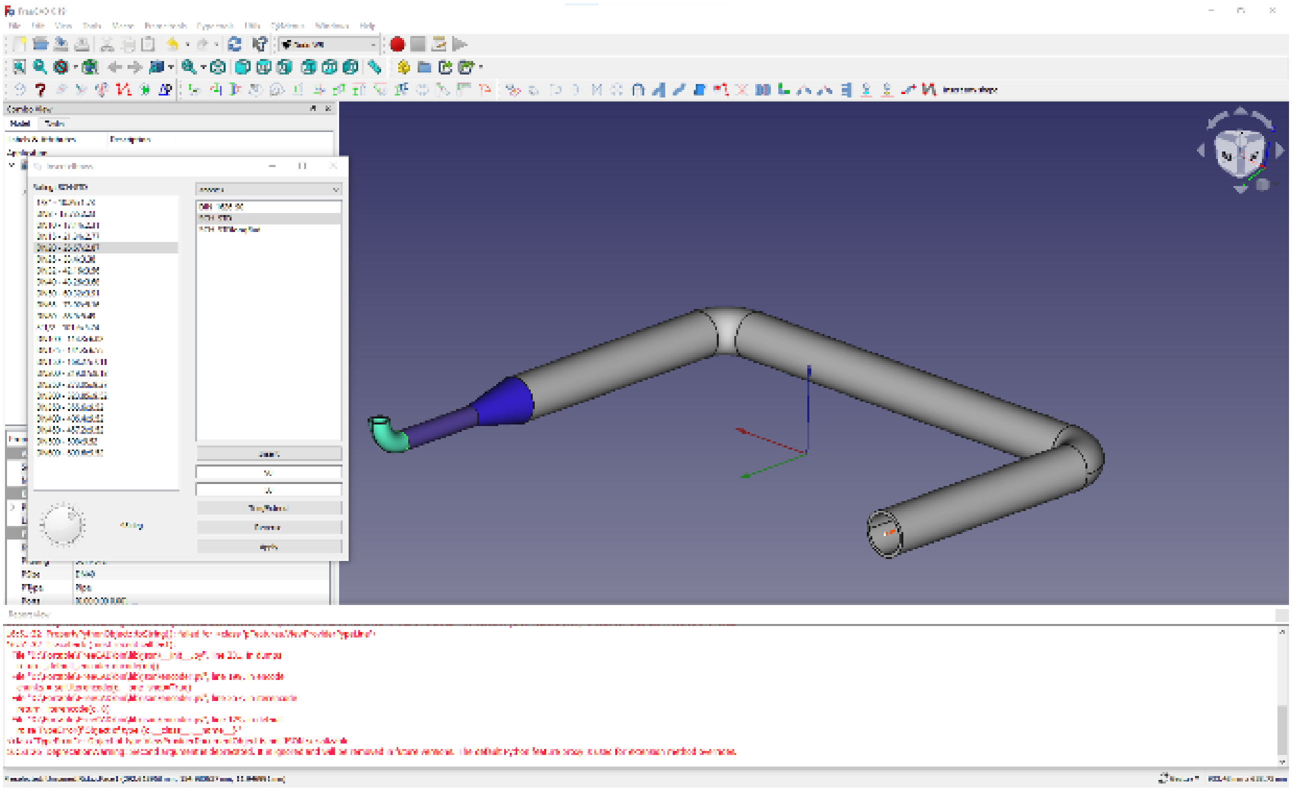
In the dialog box, select the tube type that matches the diameter of the reduction (DN20) and then type a dimension into the length input box to set the length of the tube you require.



Finally, click the Insert button to insert the tube. Similarly, we can add a curved section or 'elbow' into the design using the 'Insert a Curve' tool.

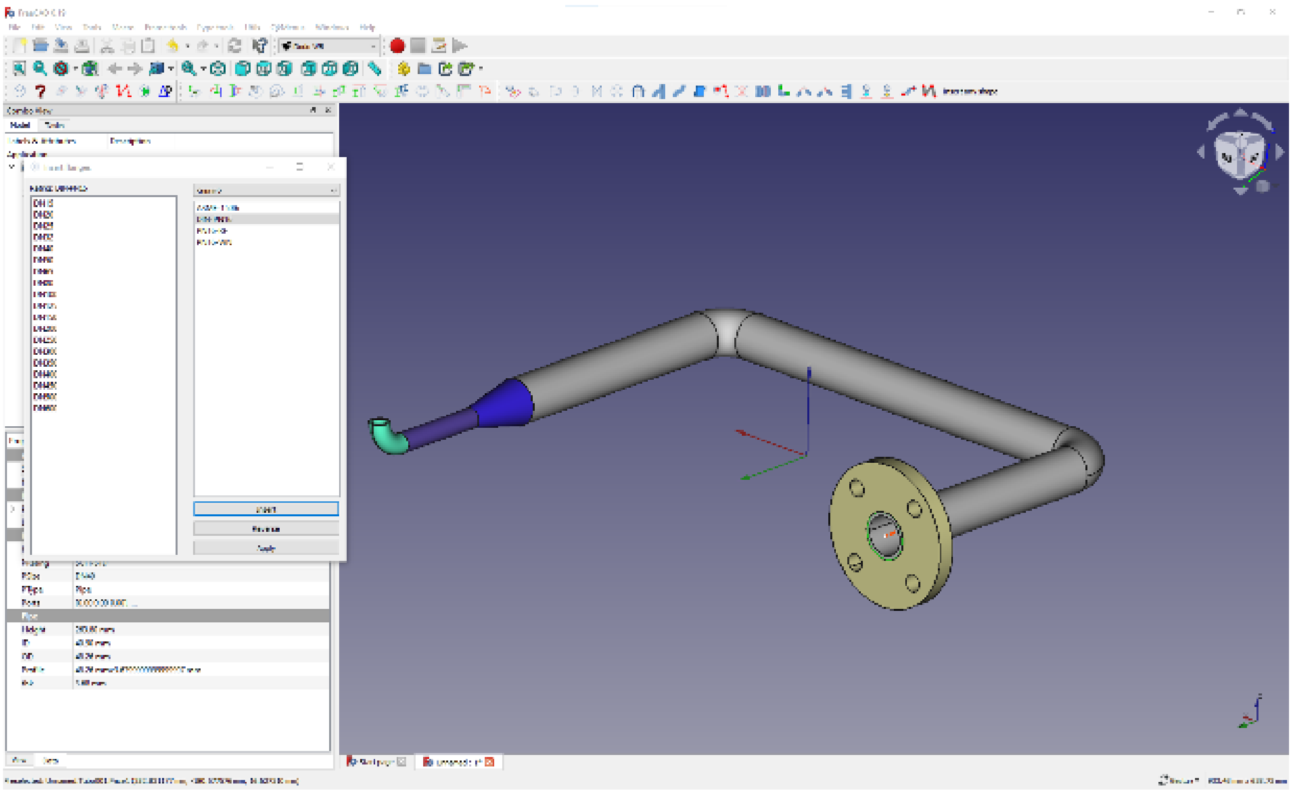


When inserting a curve, you select a target edge as before and then select a matching tube profile – again, we chose DN20. You then need to add a bend angle and a bend radius into the input boxes. Starting with values of 90 for each gives you a good result – you can experiment with other values. A small issue with this is that it's easy to input values that can't actually exist and therefore won't work! Once you have your values and options selected, click the Insert button to insert your curve. Finally, with the 'Insert Elbows' dialog, you have a dial wheel that you can adjust to change the angle of the curved elbow you have just added.



Often pipework uses flanges as joining mechanisms between pipe sections, and as such, the 'Dodo' workbench has a 'Flange' tool. At the other end of our pipe experiment, we selected the outer edge of the pipe end and clicked the 'Insert a Flange' tool.

There are various flange standards in the dialog box – we selected a DN40 type to fit our existing pipe edge. We then clicked Insert to add the flange.



We’re sure by now that you have the general idea, but it’s worth exploring the other tools available to manipulate and create pipe systems. There are tools for adding valves of differing types, U bolts to secure pipes to other structures, tools to cap the end of pipes, and tools to create flowing branches in pipes. There really isn’t much you can imagine that you couldn’t design using these tools.

Finally, it’s also worth exploring how you could add to designs using the workbenches we have looked at previously. When adding flanges, for example, it would be quite straightforward to use the ’Fasteners’ workbench to create nuts and bolts to join flanges together. We looked at the Fasteners workbench as part of the tutorial working with assemblies.

Extrude A Sketch

You might have noticed that within the ‘Open FrameBranch Manager’ dialog, there is the option to select ’<by sketch>’ in the drop-down menu where you select the different profile families. This is a really useful feature that allows us to draw a custom profile for extrusion into frame elements as a sketch.

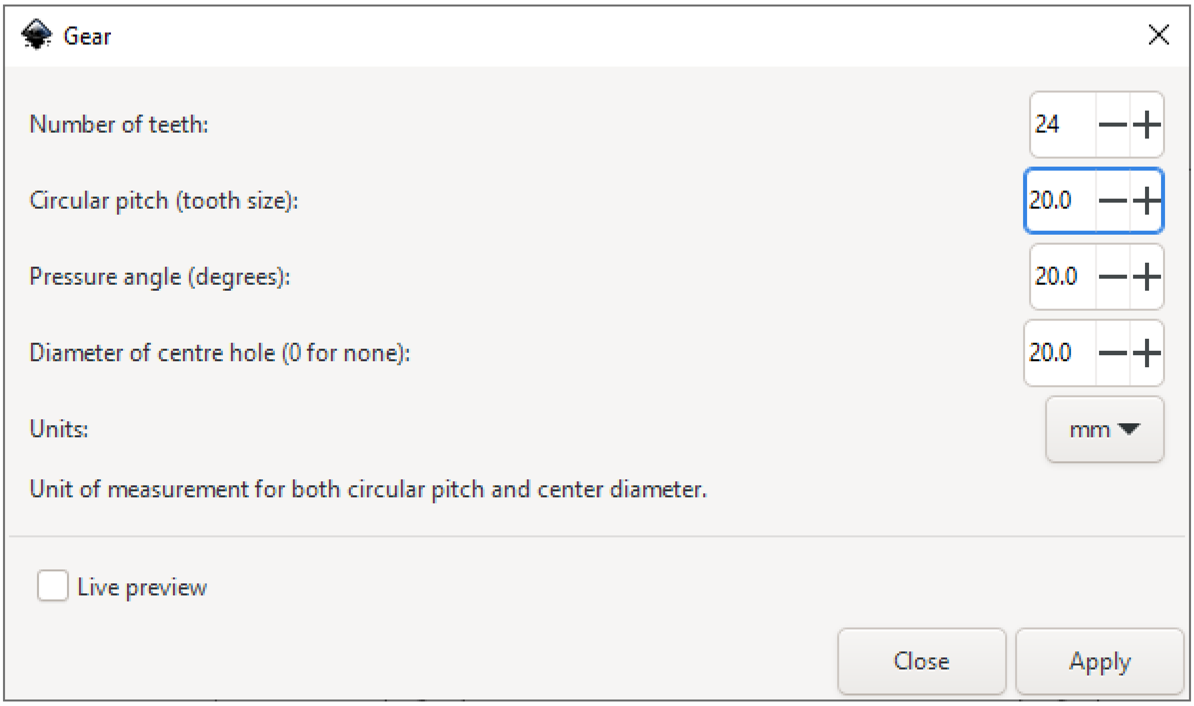
However, there is a particular rule that we need to follow when drawing a sketch for it to work correctly. The rule is that you need to draw your profile with the ‘Polyline’ tool, and you need to draw the polyline in an anticlockwise orientation. In a new project, create a sketch on the XY plane and draw a simple profile shape using the ‘Polyline’ tool. Remember, you can change the behaviour of the ‘Polyline tool’ using the M key on your keyboard, and it’s possible to create profiles with arcs and tangent lines as long as you create the profile moving in an anticlockwise direction in the sketch. For good practice, you should fully constrain the sketch. However, for a simple test sketch profile, it isn’t necessary. Once you have a profile sketch created, you can test it by first moving to the ‘Part’ workbench and creating a large cube to use as a target for a frame and then moving to the ‘Dodo’ workbench to create the frame. When selecting the profile for the frame, click the drop-down menu and select <by sketch>. You should see your sketch ’sketch’ listed in the dialog box. Select that, and then select an edge of the cube, and click OK as we have done before. You should see a frame created using your sketch profile. If you’ve accidentally created a sketch in which the polyline isn’t moving in an anticlockwise direction, you will probably still see some frame elements, but they won’t follow the edges of the cube and will extrude in random directions.

Getting Going with Gears

Gears are all around us. These clever interlinking drive systems underpin many mechanical wonders, but they have always been quite difficult to make. With modern maker tools like 3D printing and laser cutting, they are now commonly available and customisable for you to explore and use in your projects.

Gears come in many different sizes and flavours, but they are often used for the same small number of tasks. These uses include increasing or decreasing the speed of a system, increasing or decreasing the torque or power of a system, or changing the direction of rotation or direction of thrust. It's fair to say that you can deep-dive gear geometry and theory and find there is no end of stuff to learn. In this tutorial, we'll skim the surface so that you will know enough to get started.

Inkscape is a powerful free and open-source vector graphics application, and it's packed with excellent tools to aid all manner of making. Bundled into Inkscape are a collection of extensions that are incredibly useful add-on features includingt a gear generator. Clicking 'Extensions > Render > Gear > Gear' you'll see a pop-up window into which you can input data to create a gear.

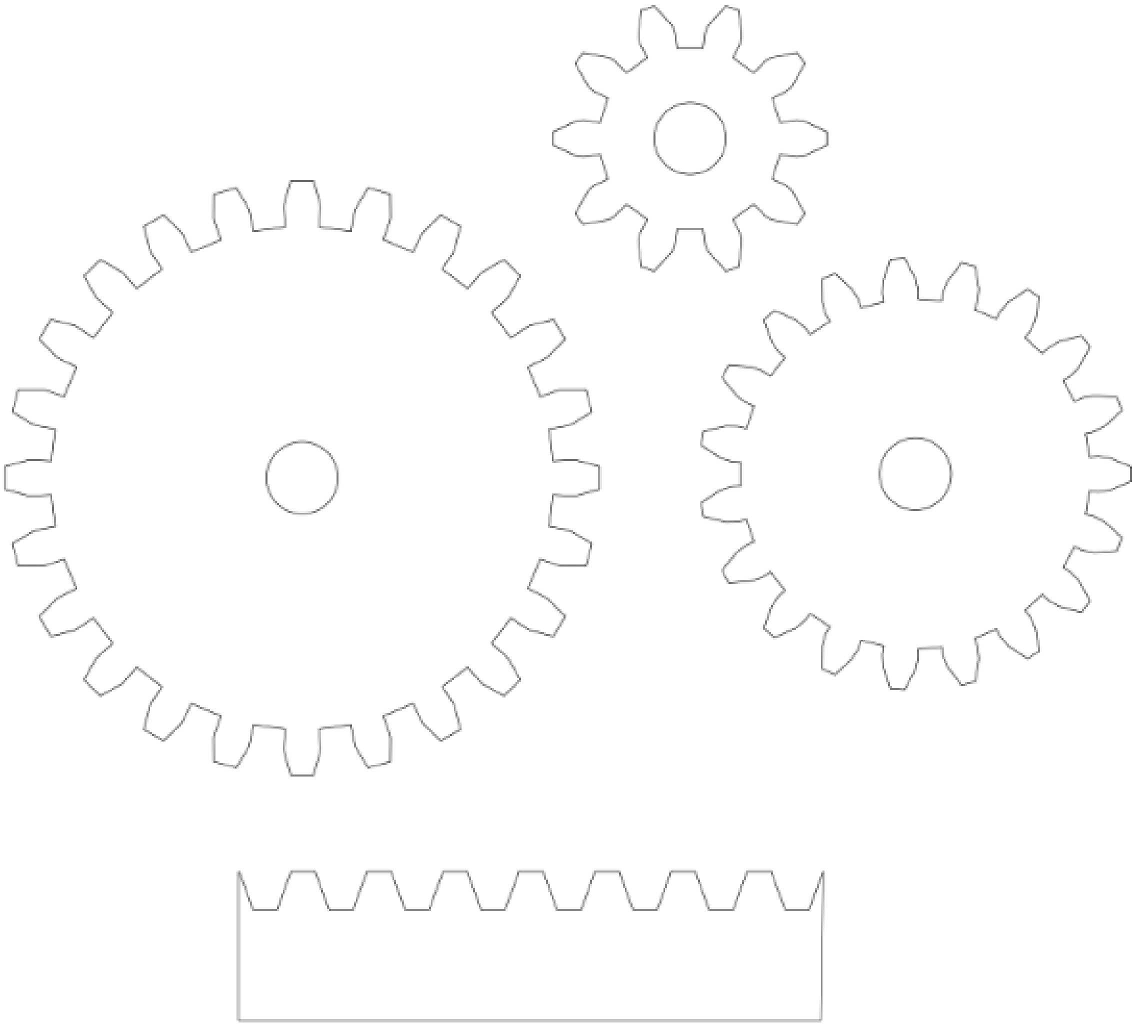


First, check that the Units option is set to 'mm', then you can experiment with 'Number of teeth' and the size of the tool using 'Circular pitch (tooth size)'. The pop-up also neatly has an option to include a hole in the centre of the gear, as you'll often want to mount it on some kind of shaft. We'll talk about the 'pressure' or 'contact' angle elsewhere in this chapter, but for now it's a good idea to leave it set to the default 20 degrees.

Once you have some settings in the gear generator pop-up, you can click 'Apply' and it should draw a gear perfectly on your canvas. The gear generator pop-up doesn't close after applying and creating a new gear. This means you can repeatedly make more gears with different values.

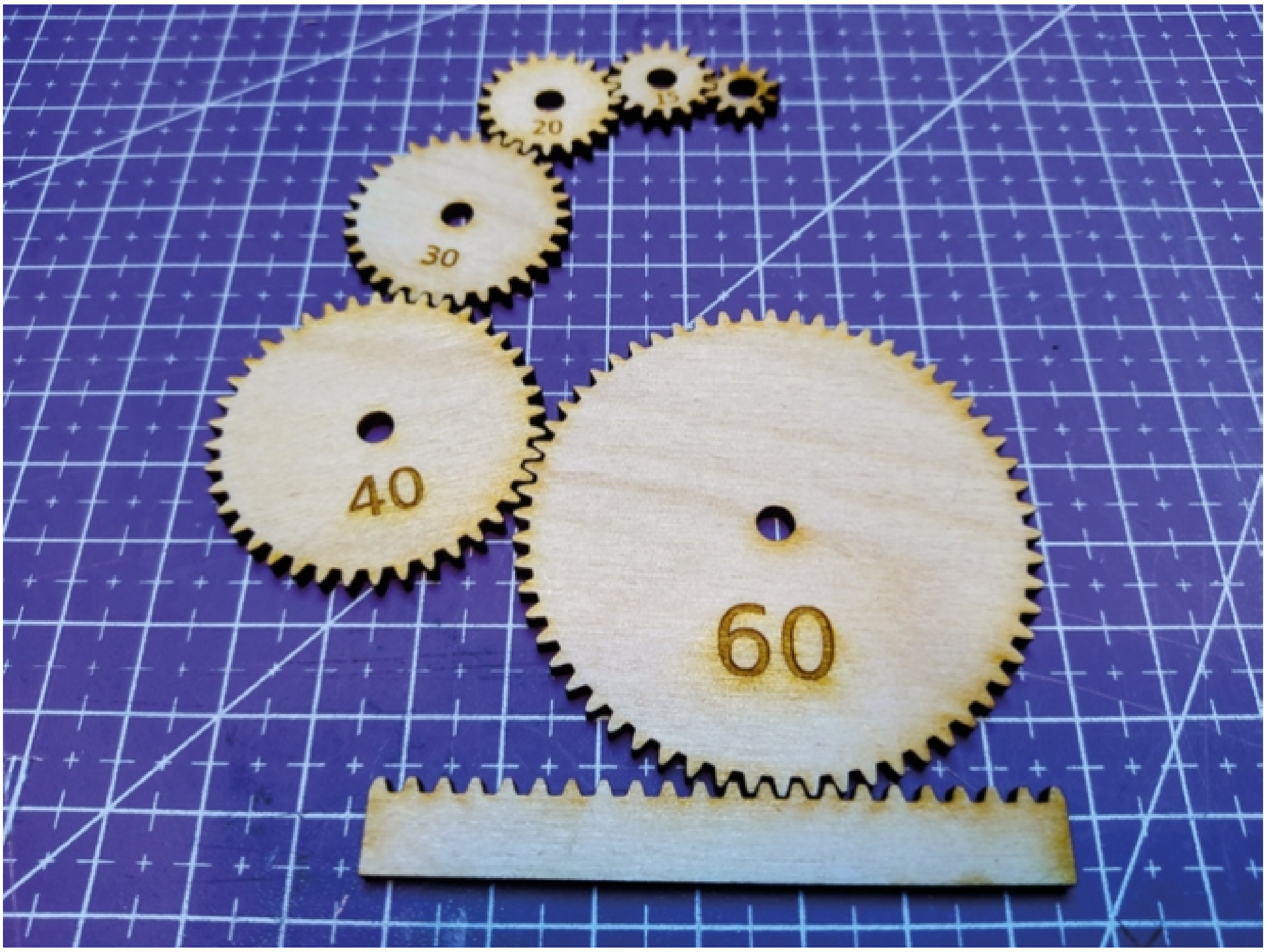
Note: If the first gear you render is quite small, it might appear behind the gear generator pop-up!

You can see that we have made a collection of gears, all of which have the same tooth size so should be compatible.



You can also see that we have made a ’rack gear’ section. Rack gears are where a small traditional round gear (pinion) engages with a long flat-toothed gear (rack). These rack-and-pinion gear systems can be used to create horizontal movement, with the pinion gear remaining in position and the assembly attached to the rack moving side to side in a linear fashion. If you click ‘Extensions > Render > Gear > Rack gear’ you’ll see a pop-up dialog to create a section of rack gear teeth. In our installed version of Inkscape, we found that we needed to set the tool spacing to double the amount of our tooth height when creating a standard gear to create a matching rack section. So, for the gears that we generated with 20 mm teeth, we needed to set the tooth spacing to ’40’. Again, leave the ’contact angle’ set at 20 degrees and set the length of rack you want to generate. Click ‘Apply’ and you should see the length of rack gear teeth appear. It only generates the teeth, so you can use the regular drawing tools to turn the rack line into an object for your rack gear project.

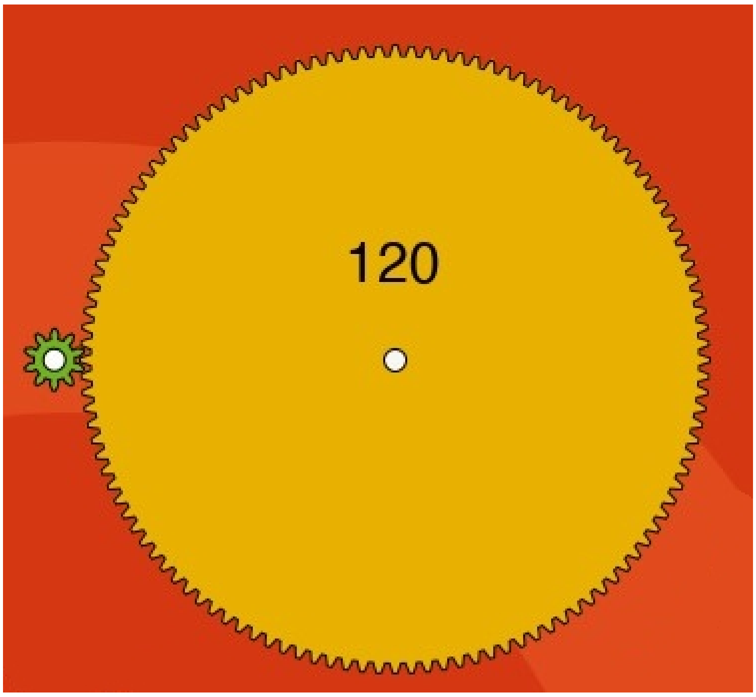
The great thing about using a laser cutter to cut gears generated in Inkscape is that you can also add details onto the gears that can be engraved. In our set of gears, we have engraved the number of teeth to each gear so that we can easily identify which gear is which.



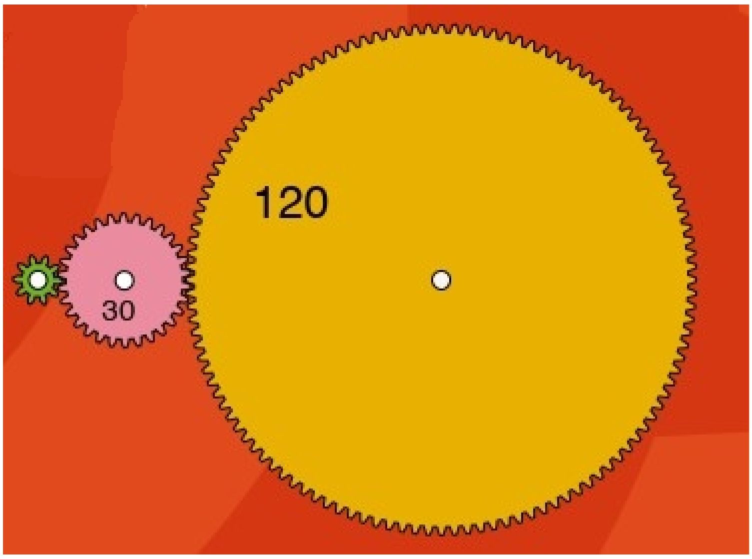
Finally, to complete our gear set, we created a backplate that had two 4 mm wide slots which would allow us to bolt our gear wheels to the board and slide them into various positions. A useful concept to understand is gear ratios.

When talking about gear ratios, it’s common to consider which gear is ’driven’ and which is the ’driver’. The driver gear is the gear connected to something that makes it turn, for instance, a motor, engine, rubber band, hand crank, windmill, ... Sometimes, instead of ’driver’ and ’driven’, you might hear the terms ’input shaft’ and ’output shaft’ – again, with the input shaft being the mechanism connecting the driver gear to a rotational source. The gear ratio, when we consider two gears, is defined as the number of teeth on the driver gear connected to the input shaft divided by the number of teeth on the driven gear connected to the output shaft. So, as in picture above, we can see that the driver gear is a smaller 10-tooth gear, and the driven output gear is larger with 40 teeth. This creates a ratio of 4:1. Therefore, the smaller gear has to turn four times for the larger gear to make a single revolution.
With gears connected similarly to our example, there is also a change in revolution frequency between the input and the output gear, and so the larger gear will rotate more slowly than the smaller gear. Again, this is proportional to the gear ratio, so if, for example, the smaller input gear is driven at 80 revolutions per minute (rpm), the output gear will rotate at 20 rpm.
There is also a change in torque when gears are combined. You may understand this instinctively if you have ridden a multigear bicycle – changing the front gear to the small cog and the rear-wheel gear to the large cog enables you to ascend a steep hill more easily. Using our example of the 40-tooth and 10-tooth gear, if we imagine that we have some kind of crank or motor attached to the smaller gear that is inputting a torque of 1 newton, then using the above equation we can see that the output torque would be 4 newtons.
You’ve probably noticed a small difference between our directly coupled gears and our bicycle discussion. In our example, the direction of rotation is opposite between the gears, so if the input gear is rotating clockwise, then the output gear is anti-clockwise. If we need gears to be directly coupled and want to maintain the direction of rotation at both the input and the output, we need to add an extra gear to the ’gear train’. Using our laser-cut gears, we can achieve this by adding an extra shaft and placing a middle gear. Another reason to add an extra gear to a gear train can be that you can reduce the size of the system when you require a large gear ratio.

For example, if we want a ratio of 120:10, which can be simplified to 12:1, we could use a 10-tooth gear and a 120-tooth gear.

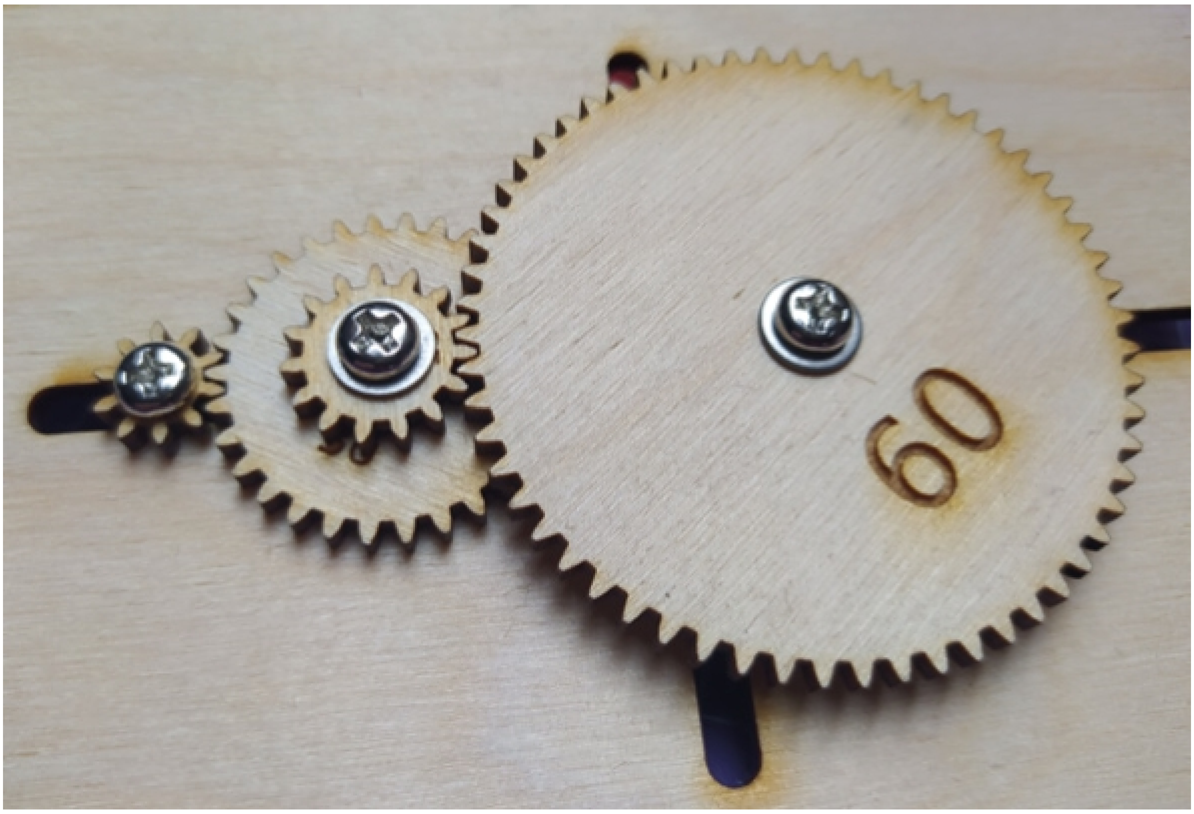


However, that 120-tooth gear will be physically quite large, and we might want to reduce the size. We could consider a 60:5 ratio, but, as a rule of thumb, gears with less than ten teeth begin to run not quite as well as their larger counterparts.
Crucially, adding an extra gear doesn't change the gear ratio overall, and while you can check this simply by looking at and rotating gears, you can also see why in the maths. If we stick with a 10-tooth input gear and a 120-tooth output gear but place a 30-tooth intermediary gear, we can calculate the overall gear ratio by multiplying the two ratios made by the three gears.



First we have the 10-tooth gear and the 30-tooth gear, making a 30:10 ratio. Then we have the 30-tooth gear and the 120-tooth gear, making a 120:30 ratio. To check the overall ratio, we need to multiply these two ratios together which looks like this $(30/10)\times(120/30)$. This returns a value of 12, which could be written 12:1, the same as our original 120:10 ratio. While we have created a system where the input and output gear now rotate in the same direction, we have added a gear to our already large gear train, making it even bigger.

We wanted to reduce the size of the mechanism. We can achieve this by using a compound gear system, where the middle gear is actually two gear wheels joined together. Continuing with our 120:10 example, when we added the 30-tooth gear, we had a 120:30 ratio on the output side. We can simplify this ratio or fraction and identify smaller gear combinations that will give the same ratio. For example, the 120:30 ratio could simplify to 60:15 or 40:10. Using our first simplification means we could create a gear train where we attach a 15-tooth gear onto the 30-tooth gear of our 30:10 ratio pair, forming a compound gear, and then swap the 120-tooth gear for a 60-tooth gear. Again, if we check the maths, we have $(30/10)\times(60/15)$ which returns our expected result of 12.



As you can see, a 30:10 to 60:15 compound gear train is much more compact than our original idea of a 120:10 would be.

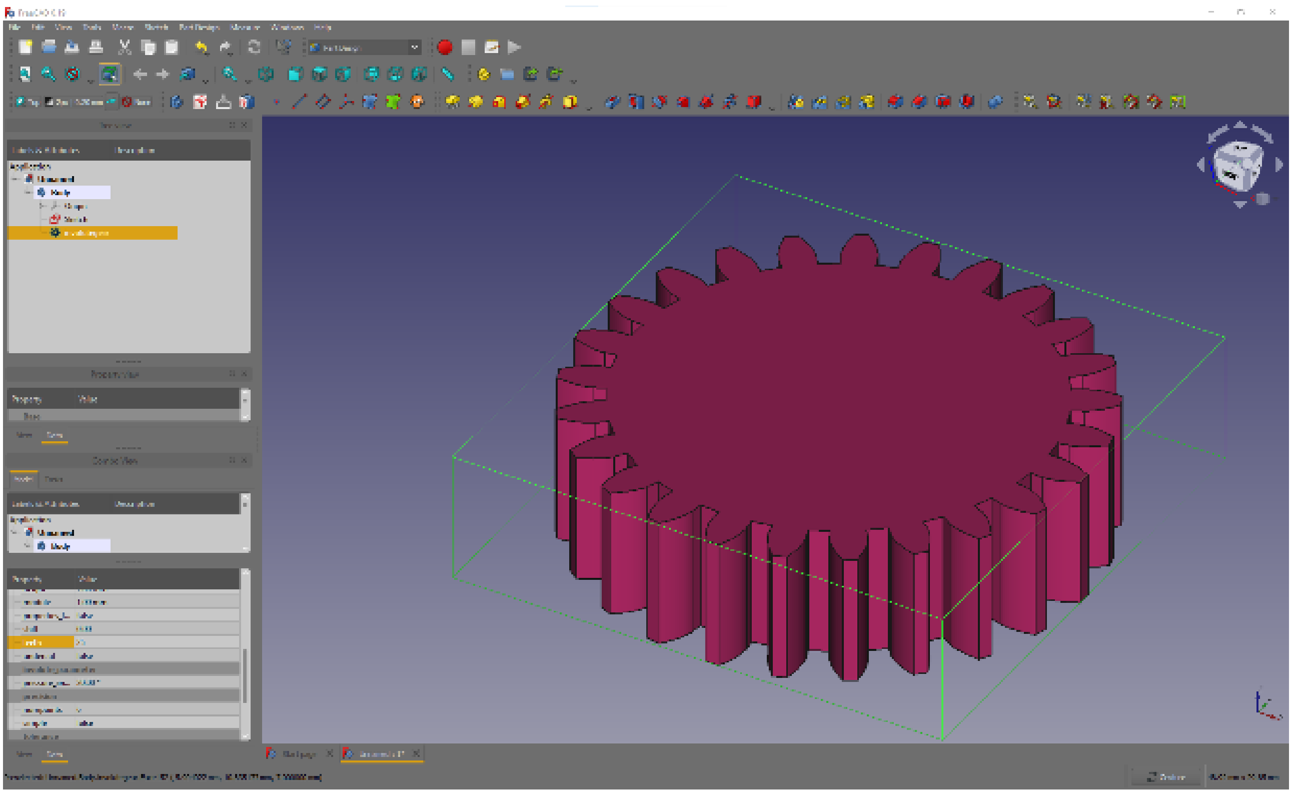
When you start to create gear trains, or you begin to consider using gears that aren't traditional flat involute gear wheels, you may consider 3D printing as a method to produce gears.

Similar to Inkscape, there are free and open-source pieces of software to help you generate 3D models of gears.
Brilliantly, FreeCAD also has an add-on workbench that makes designing gears really easy. Go to the Addon manager in the Tools drop-down and then search the list for 'FCGear'. Install this workbench and restart FreeCAD.
In a new project, move to the Gear workbench and you will see a collection of nine yellow icons that represent different types of gears you can create.

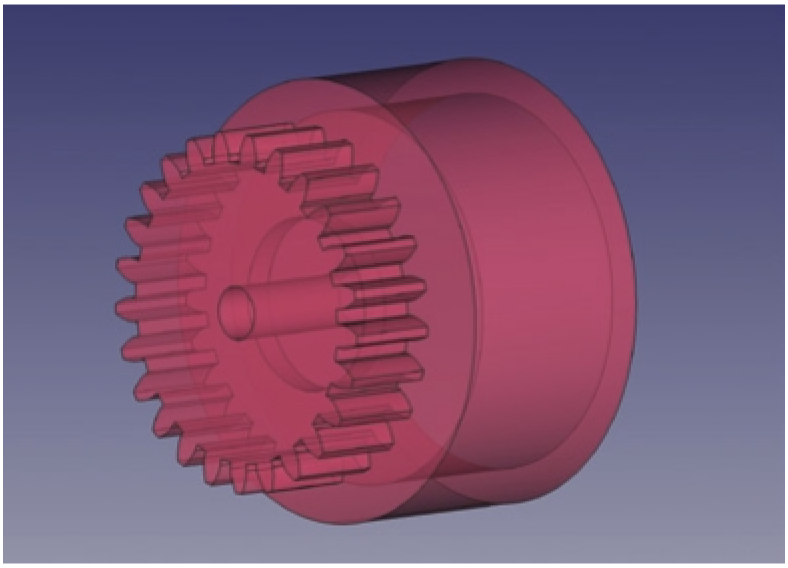
First, let's quickly make an involute gear wheel similar to the ones we created in Inkscape. It's incredibly simple to get started. Simply click the 'Create an involute gear' option and a gear will appear in the live preview and as an item in the file tree.

In the lower window of the combo view, you can see and edit the gear properties. Gear properties are numerous and can get complex quite quickly – many books have been written on the subjects of gears! To get an idea of the working parameters, it's worth looking through the FreeCAD documentation on the Gear workbench as it provides a good overview. A page specific to the involute gear properties can be found here:
https://wiki.freecadweb.org/FCGear_InvoluteGear

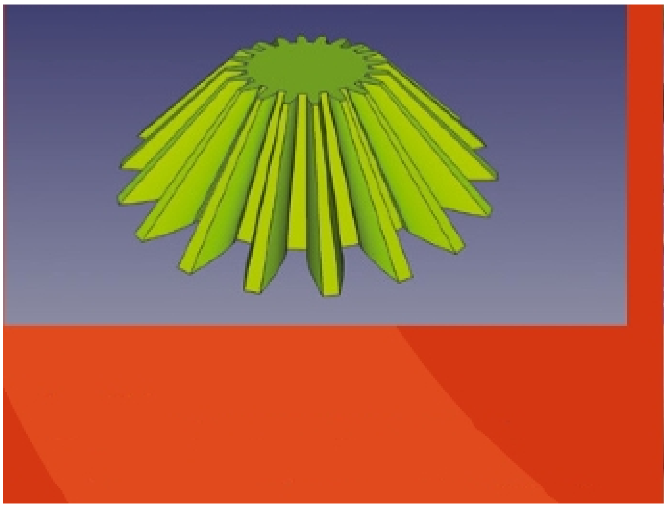
To create a variety of involute gear wheels, we can apply some changes. Scrolling down the properties, we can change the number of teeth using the 'teeth' option. Changing the number of teeth automatically calculates the new diameters and dimensions of the gear. We can change the height of the gear using the 'height' option to make a thicker or thinner gear. Lastly, we have found that it can be useful when 3D-printing gears to increase the 'clearance' property from the default 0.25. We found increasing the clearance to 0.4 made our 3D-printed gears mesh a little more easily.

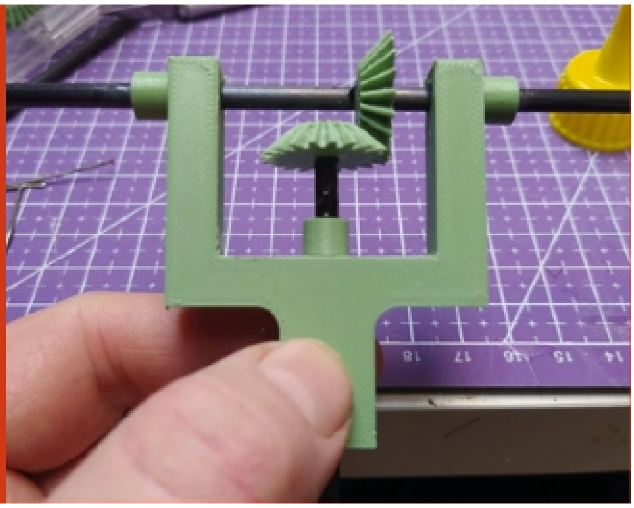


We have made a 25-tooth gear that is 7 mm tall. Notice that we have created this inside an active body that we set up on the Part Design workbench.
This means we can add to our gear using the Part Design workbench tools. In simple ways, we can add a hole for a shaft through the gear, but 3D design and printing allow us to experiment with attaching gears to other parts of designs.
For example, we have connected our 25-tooth gear directly to a simple wheel hub design. Printing this is relatively straightforward, especially if we compare he process to creating this part in metal using lathes and milling machines and indexing tools.



The Gear workbench allows us, with only basic gear theory, to explore other families of gears. The bevel gear, or a pair of bevel gears, is a great example of a gear system that can create a change in the direction of work or thrust. It’s easy to create.
Again, in a new project, simply click the ’Create a bevel gear’ tool. Similar to the involute gear, we can make changes to the design using the object properties. We have created a 5 mm tall, 20-tooth bevel gear with a clearance of 0.4. We again created this inside a body so that we could add a sketch to pocket a hole through the centre of the gear to receive a shaft.

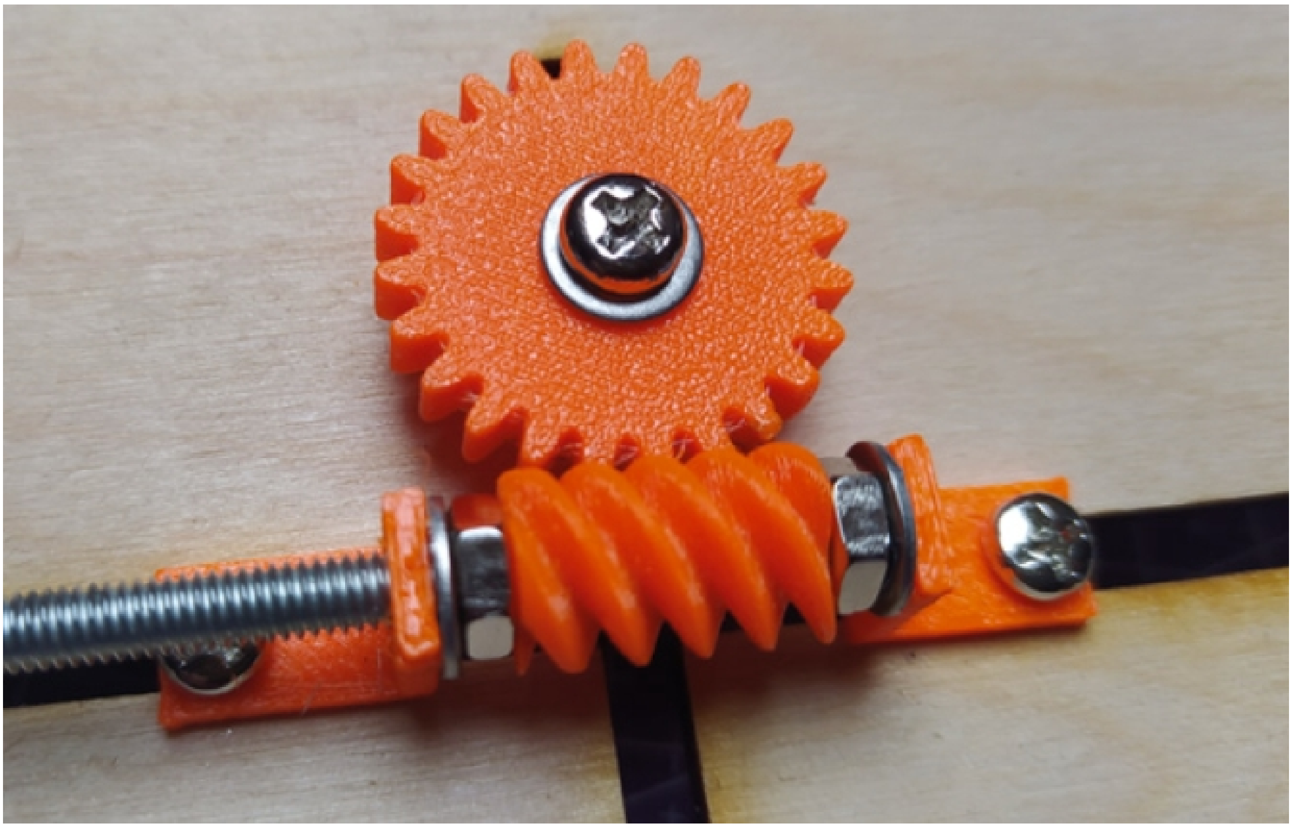




Printing these small bevel gears is quite the torture test for your 3D printer. We printed them in PETG, and as the gear narrows to the upper surface, it’s tricky to tune the print to not get small amounts of stringing in between the teeth. However, with a little light work with a needle file after the print finishes, it is possible to get a well-meshing pair. As an example of using two bevel gears to change the direction of thrust, we made a quick rubber-band-powered vehicle using some lightweight carbon fibre rods and some further 3D-printed housings.

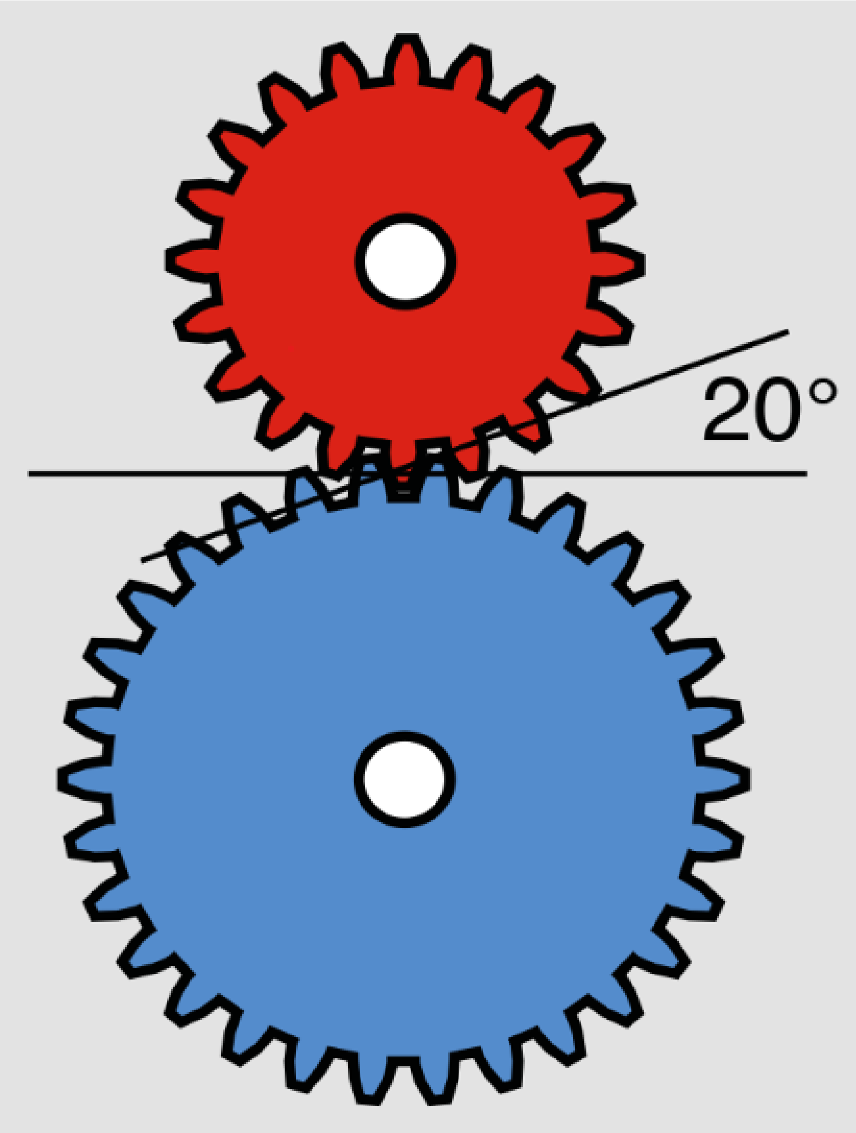
Worm gears are another interesting example of changing the direction of thrust when driving another gear. A more correct pairing for a worm gear is a ’worm wheel’, which appears like a regular involute gear albeit with a curve cut into the gear teeth. A worm wheel is a tricky gear to generate, but you can create reasonable worm gear and wheel examples using a regular involute gear wheel. Worm gears are a type of helical gear where each tooth is a helix wrapping around the length of the gear. If we click the ’Create a worm gear’ icon, we will create the default worm gear. The default worm gear has three teeth, or, as it’s hard when looking at a worm gear to discern the separate teeth, they get referred to as ’starts’.

This makes sense when you look at the flat end of the worm gear and you can see that there are three lobes which are the start of each tooth helix. In terms of gear ratio, they work in a similar way to involute gear wheels. A three-start worm gear coupled with a 24-tooth involute gear will create a 24:3 or 8:1 ratio.
Another interesting aspect of worm gears is that they are often referred to as 'self-locking'. If you set up a worm gear driving an involute gear wheel, you will find that if you try and turn the involute gear, it will lock and not move the worm gear; compared to other systems where you might need to leave a system powered or apply a mechanical brake, this can be a useful feature.



As ever, we have merely scratched the surface of both the FCGear workbench and gears generally. FCGear is capable of creating many variants of gear, and also timing pulleys, the type of which are commonplace on belt-driven 3D printers and other CNC machines. It's certainly worth clicking each of the 'Create a gear' icons on the Gear workbench and having a play with some of the parameters. There is a wealth of information online about gear geometry, construction, and theory of use, and while it's a fascinating subject, it can at times be a little overwhelming.

Pressure Angles
We've mentioned the 'pressure angle' or the 'contact angle' when talking about gears a couple of times, but what does it actually mean? In the image below, you can see two gears that are meshed together, and we can imagine one is driving the other. The forces exerted by the input gear are transmitted through the gear teeth along an angled line defined by the geometry of the gear. If we draw an imaginary horizontal line that is tangent to the gear circles, then the pressure angle is the angle between this line and the line displaying the force. Without delving deep into the complex maths and geometry underlying this, pressure angles are commonly 20 degrees, as this offers a good balance of efficiency in force transmission, minimised pressure on bearings or shafts, and generally produces least wear on a gear.



Quick Tip

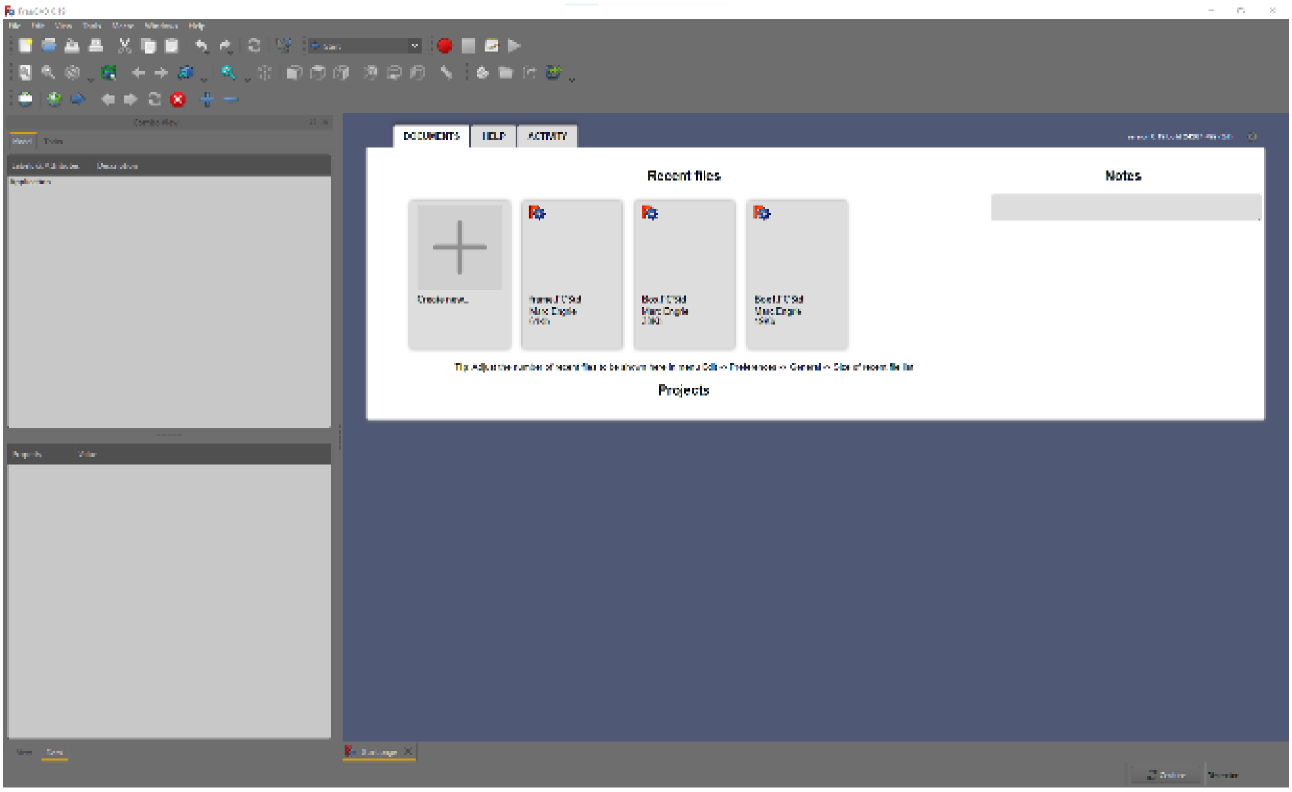
- Don't have access to a laser cutter? You could make larger gears by printing the gears onto paper, gluing them to wood, and using a bandsaw or fretsaw to cut them out

- Input torque × gear ratio = output torque. Sometimes referred to as 'torque at the wheel'.

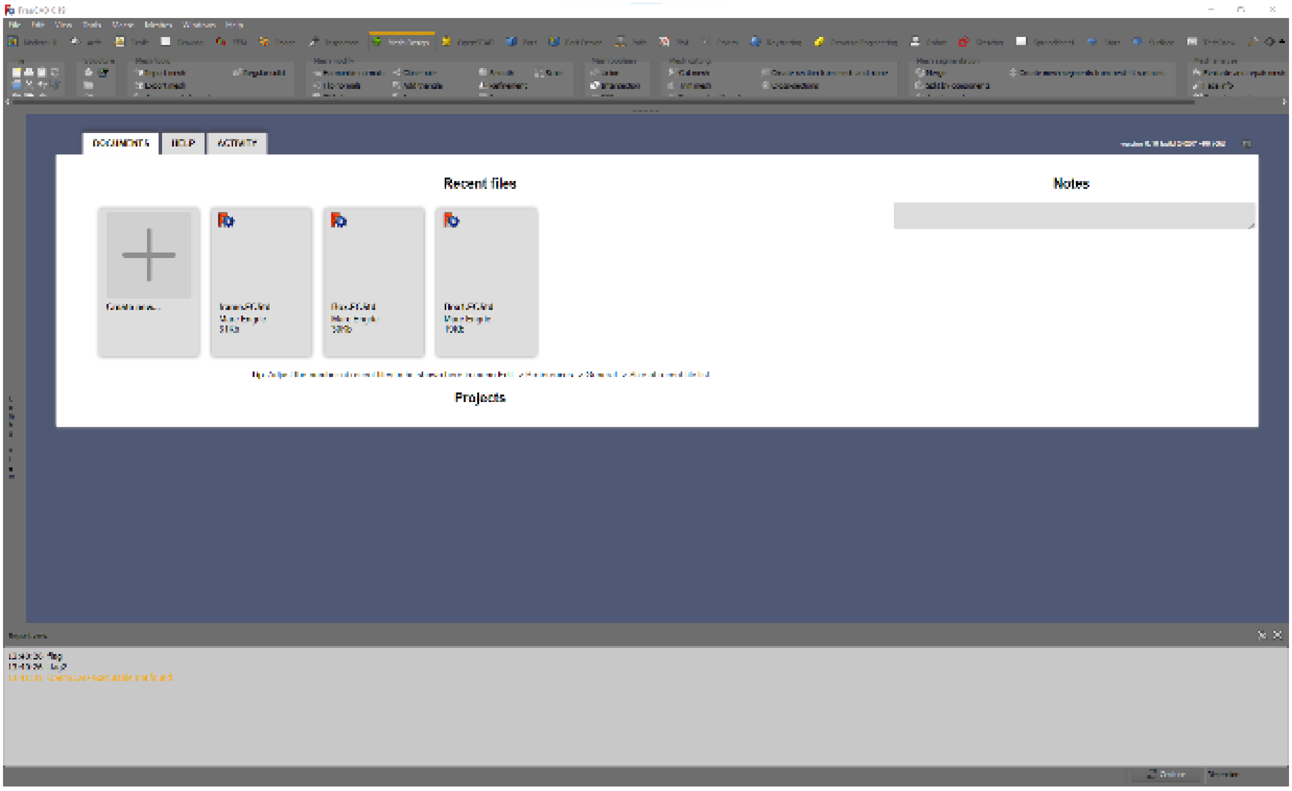
Finally

We thought it might be a good idea to look at some areas that you might like to explore now that you have a good grasp of the basics. You probably have some intermediate skills if you have followed through this series fully, so let's look at other areas of FreeCAD that you might explore.

For the sake of consistency throughout this series, we have used FreeCAD in its most default appearance – this makes it easier for people to recognise where tools are from the images etc. However, it’s possible to change up the theme and even change the user interface entirely. For simple changes of theme go to Edit > Preferences and then click the General tab. You should see options to change the ‘stylesheet’, as well as options to change the size of the toolbar icons.
Here we have changed the stylesheet to ‘Darker Orange’, and increased the tool size to medium.



For a more significant change, you might like to try a different user interface. This can be achieved by installing the 'ModernUI' workbench from the Tools > Addon Manager menu. You'll be prompted to restart FreeCAD and will be rewarded with a different UI where one of the major differences is that the workbenches are tabbed across the top of the screen, allowing for quick switching.



If you don't like the ModernUI, you can uninstall it in the usual way using the Addon manager but, additionally, you need to create and execute a macro to return to the original FreeCAD UI. This is straightforward to achieve – you select and copy the macro script text that is supplied in the description of the ModernUI workbench in the Addon manager. Then, from any workbench in FreeCAD, click the Macro drop-down. In turn, click Macros and, click Create. Give the macro a name, eg. restore and paste in the macro script copied from the Addon manager. Click the Macro drop-down and hit Execute macro window. After restarting FreeCAD, you'll have returned your installation to the original state.

Whilst on the subject, macros in FreeCAD can be incredibly useful. If you have regular tasks that you do repeatedly, or similar objects that you create across a range of projects, a macro may help to streamline your workflow. You don't have to script a macro by hand either, you can create macros using the record function. Essentially, once a macro is recording, it builds a script of all the actions you perform until you stop the macro recording. You can then save and name the macro, and execute the macro whenever needed via the macro menu. A simple way to try this is to create a new sketch and then start the macro recorder. Draw a couple of circles and constrain them, and then perhaps Pad them using part design, and then stop the macro. You can look over the script and execute it to repeat the same task. Often macros require a little editing and tweaking.

We’ve come to the opinion, whilst exploring FreeCAD, that it really is unique as a CAD environment and we love that it isn’t trying to be an open-source clone of another tool. This becomes really clear when you look at the range of workbench environments there are for FreeCAD. There are so many we haven’t explored as part of this series, but we wanted to highlight a few examples that you might like to go on to explore.
Perhaps the most obvious one we haven’t looked at is the Arch workbench which is built into FreeCAD, usually at the top of the workbench list. Arch is short for Architecture, and it’s chock-full of useful tools to help you design buildings and structures. Whilst you may well need to look through the wiki page or find an online tutorial, you can quite quickly get going with this fabulous workbench.
We can drawn a simple rectangle using the draft tools in the Arch workbench, and then used a tool to generate a wall from the wire outline. We’ve then used the built-in tools to simply click and add preset windows and doors in our simple structure. Check out some of the amazing structure designs in Arch on the FreeCAD forums to get properly inspired.

The Robot workbench is perhaps a great example of just how powerful a tool FreeCAD can be. Whilst the Robot workbench is still included in FreeCAD, it isn’t being particularly actively developed, but it’s a fun workbench to play with for a few minutes. Often, with many workbenches, the FreeCAD wiki has a succinct tutorial to work through to get you up and running with the basics of the bench. The Robot workbench is no exception, and working through the instructions at https://wiki.freecadweb.org/Robot_tutorial, you can import an industrial robot arm.

We used a Kuka IR500, and you can create trajectories to move the arm through, which then can be played like a script. If you happen to own or have access to a Kuka robot arm in real life, you can even export the subroutines you create to run on the machine.

The Path workbench is FreeCAD’s main CAM environment, and is where you can set up CNC routing and milling toolpaths to create parts on a wide variety of machines. We wrote about the Path workbench. Of course, FreeCAD has developed a lot since then and the Path workbench, with its amazing dedicated developers, is one of the most actively developed areas of FreeCAD. As such, it’s changed somewhat since our write-up, but there’s plenty of information about it online. It’s definitely worth exploring, and offers robust and powerful CAM approaches, and is capable of creating G-codes post-processed to suit a wide variety of machines.

The FEM workbench enables complex analysis of objects under loads, and this type of FEM/FEA approach often is quite expensive to perform in other proprietary CAD environments. It requires a couple of extra external packages to be installed/configured, and then models can be assigned material properties and then tested under loads. To get a fleeting appreciation of what this looks like, even without installing the extra packages, you can open one of the example projects bundled into FreeCAD on the FEM workbench, and see the type of strain/stress analysis and representation you can achieve.

When we looked at the Mesh workbench, we needed to have another CAD environment, OpenSCAD, installed on our machine to enable some of the Mesh workbench features. There’s also, however, a dedicated OpenSCAD workbench within FreeCAD. OpenSCAD differs from many CAD packages in that it is script-based. This means you ‘code’ your CAD models. For example, to make a cube in OpenSCAD, you might write a line that looks like ‘cube([10,10,10],true);’ which creates a 10 mm cube centred around the origin point. The addition of a dedicated OpenSCAD workbench means that if (as we did) you have used OpenSCAD and are moving to FreeCAD, any components or models you have made in OpenSCAD can still be used within your FreeCAD designs. The workbench also allows you to create new parts using OpenSCAD scripting directly. If you’ve solved a really hard geometry in OpenSCAD, this gives you not only the option of reusing that work, but also for that work to be enhanced using all of FreeCAD’s other tools.

Whilst there are more than enough workbenches to get to grips with listed in Addon manager, there are also workbenches out in the community that aren't included either in the main download or in the Addon manager. These are often in development and experimental in nature, but it can be fun if you find something in the community that relates to something you know about or are interested in.

Sometimes these workbenches are not included as development on them has ceased. However, often the open licences of these archived workbenches mean that someone could begin to work on them again. They also may still be functional nd of use. One excellent example is a workbench first released back in 2015: the Glider workbench enables the design of paragliding canopies and recently, in 2021, a forum member posted pictures of a paraglider in flight that had been designed with this innovative workbench.

Are you a developer? FreeCAD is always interested in attracting more developers to contribute to the software. The majority of the FreeCAD workbenches are written in Python. However, there are also some workbenches and areas of FreeCAD written in C++.
There are many areas to look at, or perhaps you have your own idea for a tool that you'd like to create for FreeCAD, or an issue that you think you can solve.
The first port of call is the forum, and in particular 'Developers corner', where it would be a great idea to introduce yourself. Read the pinned thread, 'Read this first if you want to write code for FreeCAD', to get the basic orientation of the FreeCAD developer community approach.

In this final part, we wanted to recap and expand upon other sources of help, should you need it, for FreeCAD. One of the best resources out there is the FreeCAD community forum – forum.freecadweb.org. We've mentioned it before, but it's worth repeating a couple of points about the forum. First of all, there's a really high chance that there is information that can help you with your particular situation already on the forum. It's vast, and a search query for your subject will often return a heap of golden knowledge. It's definitely worth searching before adding a new question or thread.
Secondly, in order for community members to be well-equipped to answer a question, it's really useful if you post basic information about your operating system, version of FreeCAD, etc. Wonderfully, FreeCAD has this built-in: if you go to Help > About FreeCAD from any workbench in FreeCAD, you will see your system information with a handy 'Copy to clipboard' button. Click the button and paste it into your opening post, followed by your question, and it may well make it much easier for someone to answer.
Sometimes you'll find that something in FreeCAD doesn't work as you expect it to. Again, a good bit of forum searching and wiki reading will often yield a solution, but sometimes something might be broken.
It's always worth, and always preferred, that you search the forum, and post on the forum, checking if others have had your experience and allowing other community members to help identify if it is indeed a bug. If the consensus is that it is a bug, then someone in the community will raise it as an issue in the correct way or will inform back in the thread that the issue is a previously identified issue. Often, the answer will be that an issue in the current stable release version of FreeCAD has been addressed in the developmental version of FreeCAD. On that subject, the developmental versions of FreeCAD are always available for people to try. We'd suggest, as we have throughout these articles, that using the latest stable version whilst you are learning is recommended, but as you understand more it can be fun to try the developmental versions, which often show a sneak preview of new, interesting features and developments. Also, as FreeCAD is open source, there are some forked alternate versions of FreeCAD – this helps to keep FreeCAD development innovative, as the forked versions often explore different features and approaches. One popular forked version of FreeCAD, that contributes a lot of development back into the main FreeCAD branch, is the LinkStage3 fork by Realthunder – with some of the basics under your belt, you might like to try it out.
So, as we come to the end of this series, it's perhaps worth considering where to look for other sources of information and tutorials on FreeCAD, as well as perhaps places to find some inspiration! First up for inspiration, new releases, and new feature announcements – and also a great place to see challenging, interesting work done in FreeCAD – is following the official @FreeCADNews account on Twitter. Watching that Twitter account, you'll see the wide range of

approaches people use with FreeCAD and also come across people doing CAD challenges using FreeCAD and more. They often post using the hashtag #madewithfreecad, and it's astonishing to see what people have made.

YouTube is absolutely crammed with tutorial videos, and pretty much every area of FreeCAD is covered. One particular YouTube channel, Joko Engineering, deserves particular mention for excellent tutorials that not only cover using FreeCAD tools, but also often explore underlying CAD concepts.

We've mentioned it a few times, but the official FreeCAD documentation is in the form of a wiki and is available at wiki.freecad.org/Getting_started. All the built-in workbenches and tools are documented, and there are numerous short tutorials on different workbenches. It's got heaps of information clearly laid out, and the wiki is already available in numerous languages. It also has wider reading on FreeCAD.

A nice wiki entry that gives a quick-read version of FreeCAD's origins and history can be found at wiki.freecadweb.org/History.

We hope this tutorial series has been useful, and we feel that if you worked through all 16 parts, you would be pretty competent at modelling in FreeCAD.

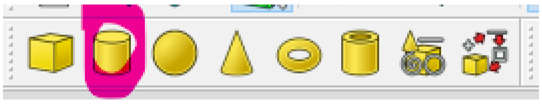
In our opinion, it's definitely a process that requires practice, and with each new model you'll find some area that requires a new approach to be explored and learnt. As such, it's a great idea to work with FreeCAD as much as possible. One thing we have seen a lot of in the communications around this series is that often CAD communities have competitions or ongoing challenges which can not only be good fun, but also provide opportunities for you to expand and learn more approaches.

We've really enjoyed writing this longer series, which has been a new approach for HackSpace magazine, and we hope that you have enjoyed it. Many thanks to those of you that have worked through all the tutorials, reaching out to us on social media to share your progress. We continue to look forward to see what you can create using the brilliant FreeCAD platform.

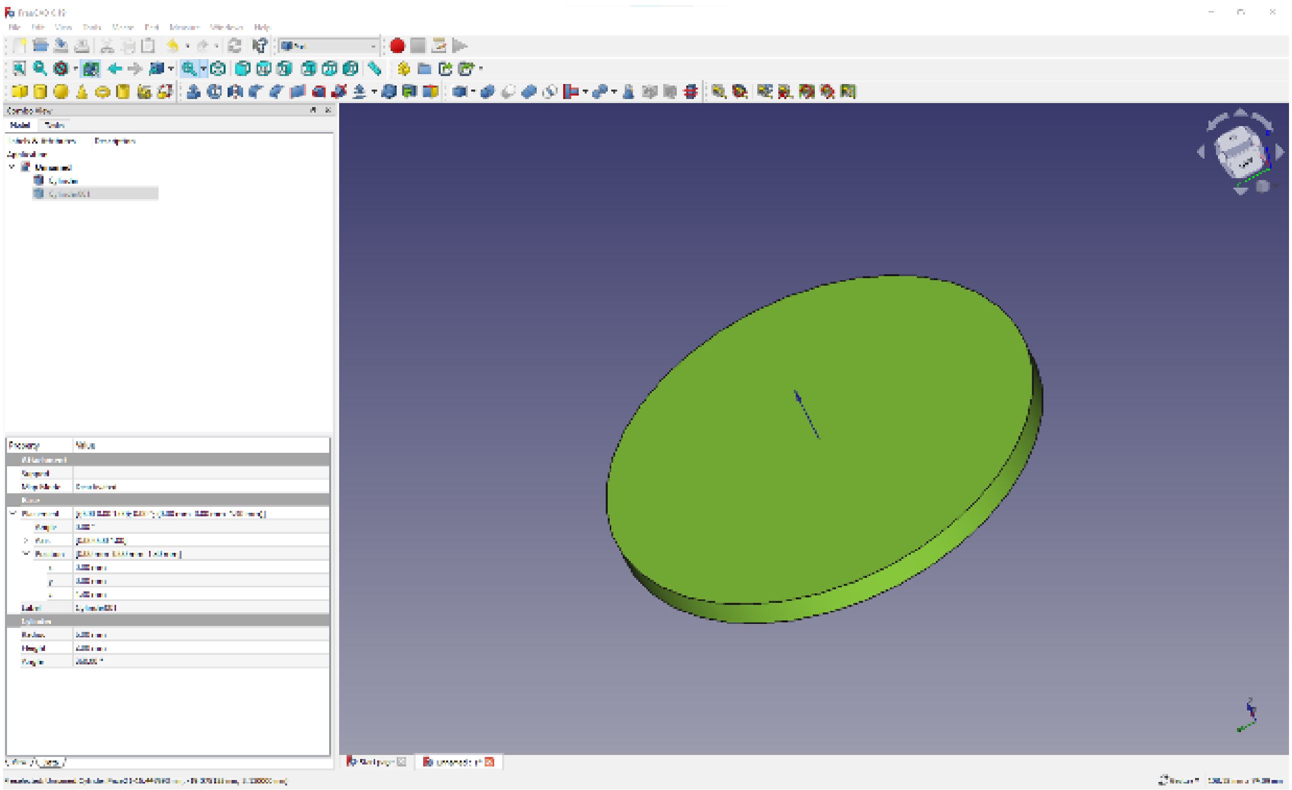
>>>>>>WORK IN PROGRESS<<<<<<

Let’s explore FreeCAD for creating a simple 3D model and some toolpaths for CNC routing

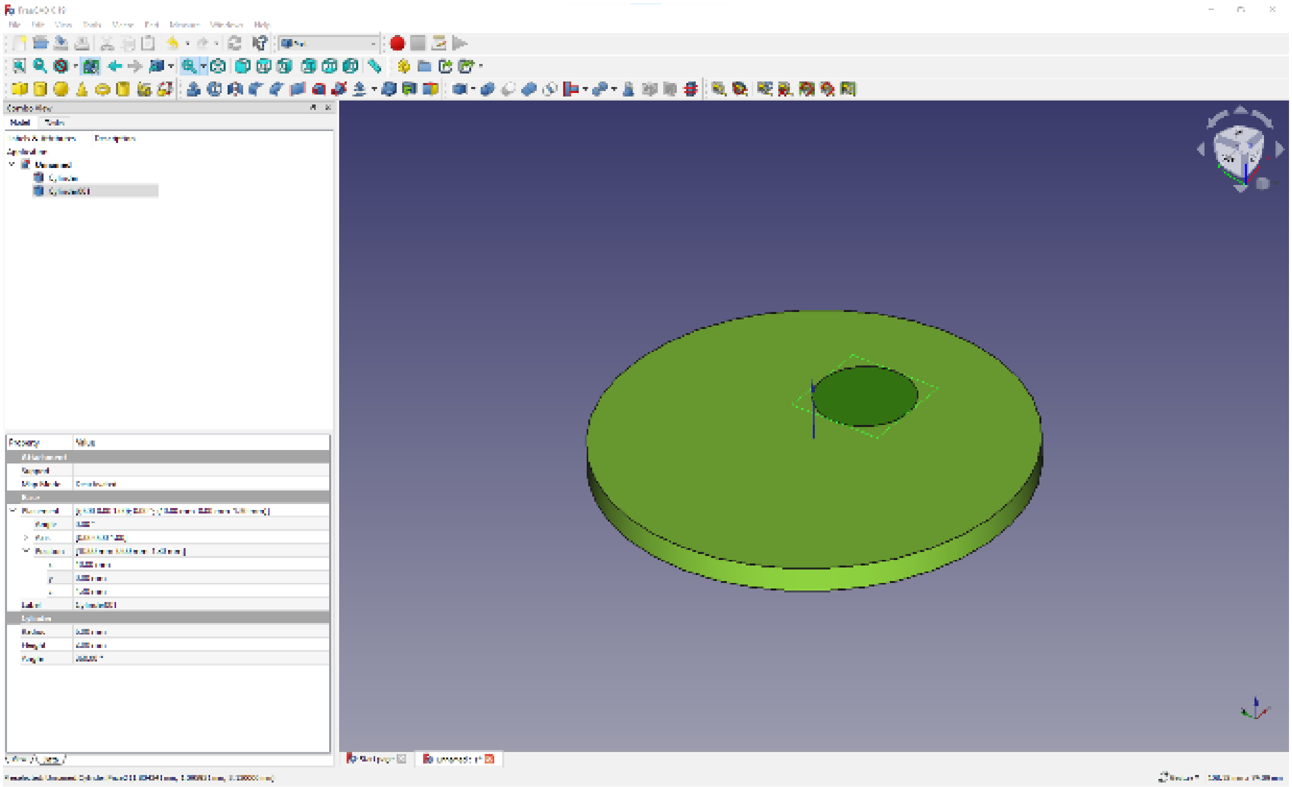
We’re going to look at how to use FreeCAD to create designs that you can mill using a CNC machine. This way, you can create parts out of wood or metal potentially much larger and stronger than you can with other automatic fabrication methods such as 3D printing or laser cutting.
To begin, switch to the ‘Part’ workbench. For this project, you are going to design a simple disc with a hole and a pocket cut into it – not the most amazing project, but you can learn a lot along the way.
First, select a cylinder.



Clicking the cylinder icon makes a cylinder appear on the right-hand side. You are going to eventually CNC-rout your flat disc from some 3.13 mm plywood – so let’s make this cylinder 3.13 mm tall. Change this dimension in the dialog box of the Combo View on the lower left. Then change the radius to 26 mm.



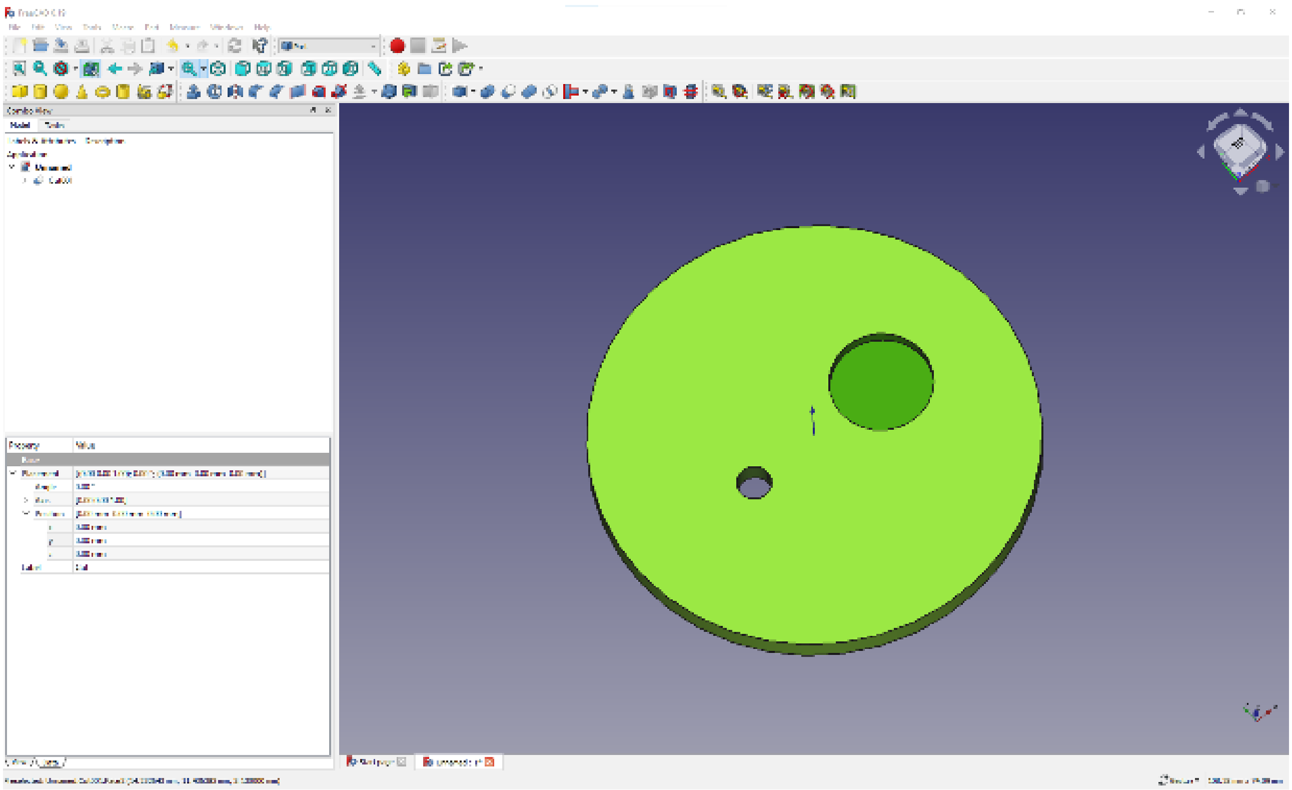
You are now going to repeat the above process to make another cylinder; this should appear in the file tree as ‘cylinder001’. Change the size of this cylinder to a radius of 6 mm and a height of 2 mm. It will now appear to have disappeared, as it is underneath your original cylinder. In the dialog box, expand the ‘placement’ drop-down and then the ‘position’ drop-down, and let’s change the position of this cylinder. Change the z coordinate to 1.3 mm. This raises the cylinder up 1.3 mm and sets the top of the cylinder flush with the top face of your original larger disc. Set the x axis position to 10 mm.



The next operation is to subtract the second smaller cylinder out of your first to leave a 2 mm deep pocket. To do this, hold the CTRL key down whilst selecting both the cylinders in the file tree. Next, find the 'Make a cut of two shapes' tool . Click the button, and the smaller cylinder should be cut out of the larger disc.

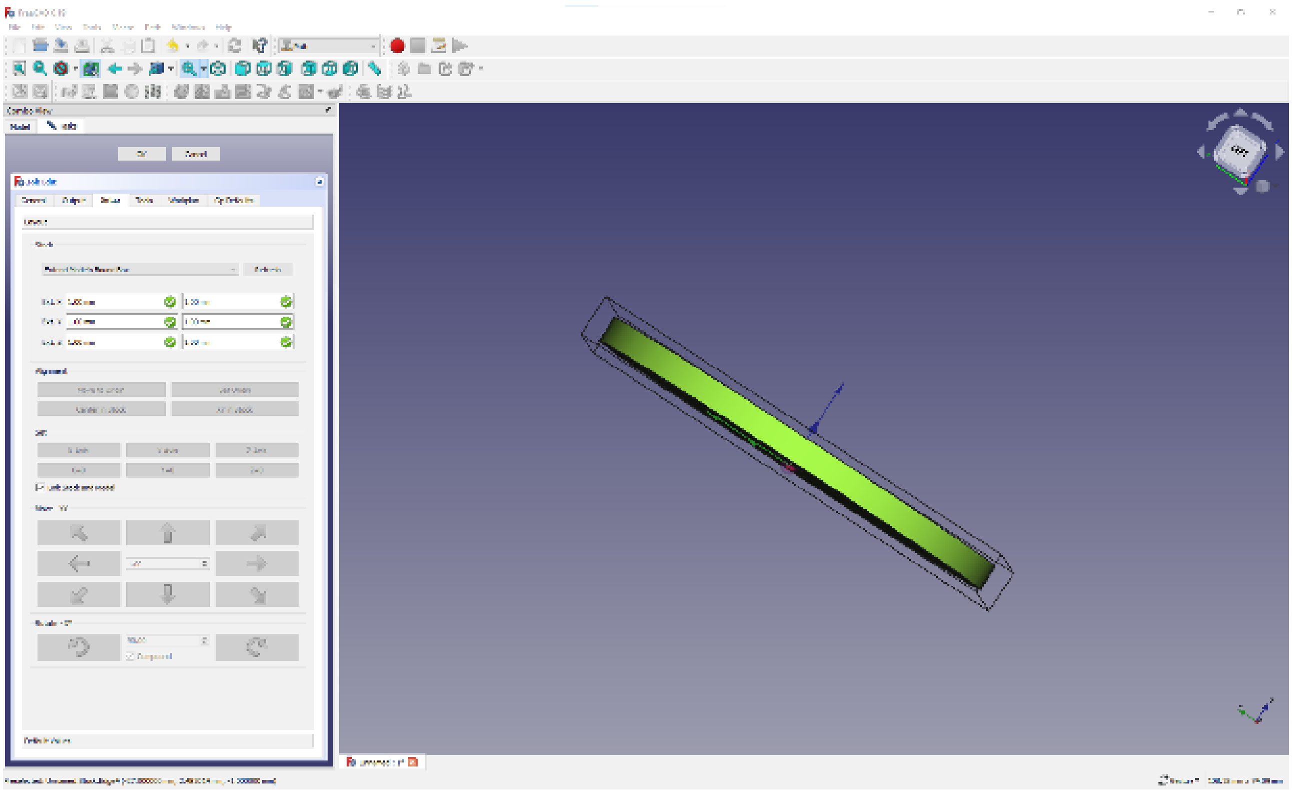
If at this point the larger disc disappears, press CTRL+Z to undo and reselect the two cylinders, but select them in the reverse order from what you did the first time and try again!

You repeated the above operation with a 2 mm radius, 10 mm tall cylinder to create the small hole through the piece, and you positioned this hole at -9 mm in the x axis.



Walking The Path
Toolpaths are the path that a cutting tool follows. This, together with some information about the size and shape of the cutting tool and the speed and feed rates the tool is travelling at, will all be combined into a G-code file which you can send to a CNC router. To begin this process in FreeCAD, you need to switch to the Path workbench.
Next, you need to select everything you have created (simply highlight the cut001 in the file tree view) and click the 'Create a path job object' tool icon. This button should be the only new icon that is available and not greyed out, appearing as a drill bit on the right-hand side of some green lines.

FreeCAD now will appear to have surrounded the disc you modelled with some lines, and a dialog box appears. FreeCAD has actually made a copy of the object, and has made the original object not visible.
To toggle visibility of items in FreeCAD, you simply hover over them in the file tree view and press the SPACE bar. It's important from now on that the original model is invisible, and that you work on the 'Job' model.
These lines, the bounding box, represent the stock material you are going to machine the disc out of.



As you are going to use 3.13 mm plywood that is the correct thickness, you don't need any material above the surface of the disc. Therefore, you will adjust the z axis dimension of the bounding box in the 'Setup' dialog box to reduce the stock dimension to zero by changing the 1.00 mm on the second Ext. Z axis column to 0.00 mm.
Clicking 'OK', you should then be able to see that the top line of the bounding box is level with the top of your model. You are also going to change where the zero or datum is on your model. Double-click 'Job' in the file tree and then carefully select the node point at the left-hand corner of the bounding box on the level of the top of the bounding box. Having selected the node, click the 'Set origin' button. The red, green, and blue datum arrows should move to this corner now. This can be seen as the origin of the tool travel lines.

Next, you are going to create a new tool to use, and create a tool controller. To start this, click the icon that looks like three drill bits called 'Tool Manager'. In the window that appears, click 'New tool', and then, in the tool editor window that appears, you are going to specify the dimensions of a cutting bit.
For all your CNC operations on this job, you will use the same tool – a 3 mm diameter, two-flute end mill. Give your tool a name such as '3 mm end mill' and set the 'H' to 15 mm (this is the available height of cutting surface on the tool, not the length of the entire bit). Click 'OK' to return to the 'Tool Library' window. Before closing this window, select the tool you just made and click the 'Create Tool Controller(s)' button (Figure 6). In the file tree, you should now have a tool controller that has the name of the tool you just created. Double-click on this and bring up a dialog box. In this Tool Control editor, you can set the feeds and speeds of this tool. You set ours to a conservative horizontal feed of 350 mm/min and a vertical feed of 100 mm/min.

The first toolpath you will generate is the path the tool takes to cut out the pocket for the small cylinder you placed, cut part-way through the larger disc. Select the bottom face of that hole and then click the 'Create a path pocket object from a face or faces' button. In the 'Pocket Shapes' dialog box, you then set the depth of cut on the 'Depths' tab – so you are starting from 0.00 mm and cutting down to -2.00 mm.
And you are conservatively again going to step down in increments of 0.5 mm, so it'll take four passes to cut this pocket. You also need to check the height clearances; on the Height tab, you set the safe height to 3.00 mm and the clearance height to 5.00 mm. Finally, on the 'Operation' tab, you set the cut mode to 'conventional', the pattern to 'spiral', and the step over

percentage (the amount the tool moves outwards on the spiral after each turn as a percentage of the tool diameter) to 40%. Click the 'Apply' button and you should see your first toolpath appear on the model (Figure 7). Repeat the above section, but create a toolpath using the inside face of the through-hole. Follow the same process, but set the tool to a depth all the way through the disc.
You need to cut out the shape of the overall disc from the stock. To do this, you are going to select the whole model and select a 'Profile based on face or faces' operation instead of the pocket cut operation you used earlier. Select this and set the depths and heights.
You will notice under the 'Operation' tab that you have the option to set the tool to the inside or the outside of the path – set this to outside and the direction to CW.
Click 'Apply'. If you cut this path in its current form, the workpiece would become loose as the machine cuts it out, and the work might be rapidly ejected! Adding 'tags' to the toolpath means that the machine leaves small tabs holding the part to the stock, and then you remove the part from the stock by hand. Make sure the profile path you just made is selected in the file tree view, and then click Path > Path Dressup > Tag Dressup from the main toolbar. In the resulting dialog box, you set the number of tags to 3, the width to 5.00 mm, and the height to 1.2 mm. Clicking 'Apply' adds the tags – if you zoom in, you can see the tool has moved around the tags, leaving material.

Finally, you need to export your toolpaths as G-code, ready to be used with your CNC router. Double-click the 'Job' entry in the File System view. In the dialog box, click the Output tab to see various options (Figure 8). Different CNC machines use slightly different forms of G-code, and some have post-processors to ensure the code is exported in the correct format. FreeCAD has numerous built in, including the one you want to use: 'grbl', because the CNC router we'll be using is controlled using the open-source Grbl system. You selected 'grbl' and added a location and a file name for your G-code file.
Grbl likes files with the suffix .nc, so you added this.
Click 'OK' on the dialog box and then use the 'Post process the selected job' option (next to the 'Create a path job object' button you used earlier) and save the processed G-code file.

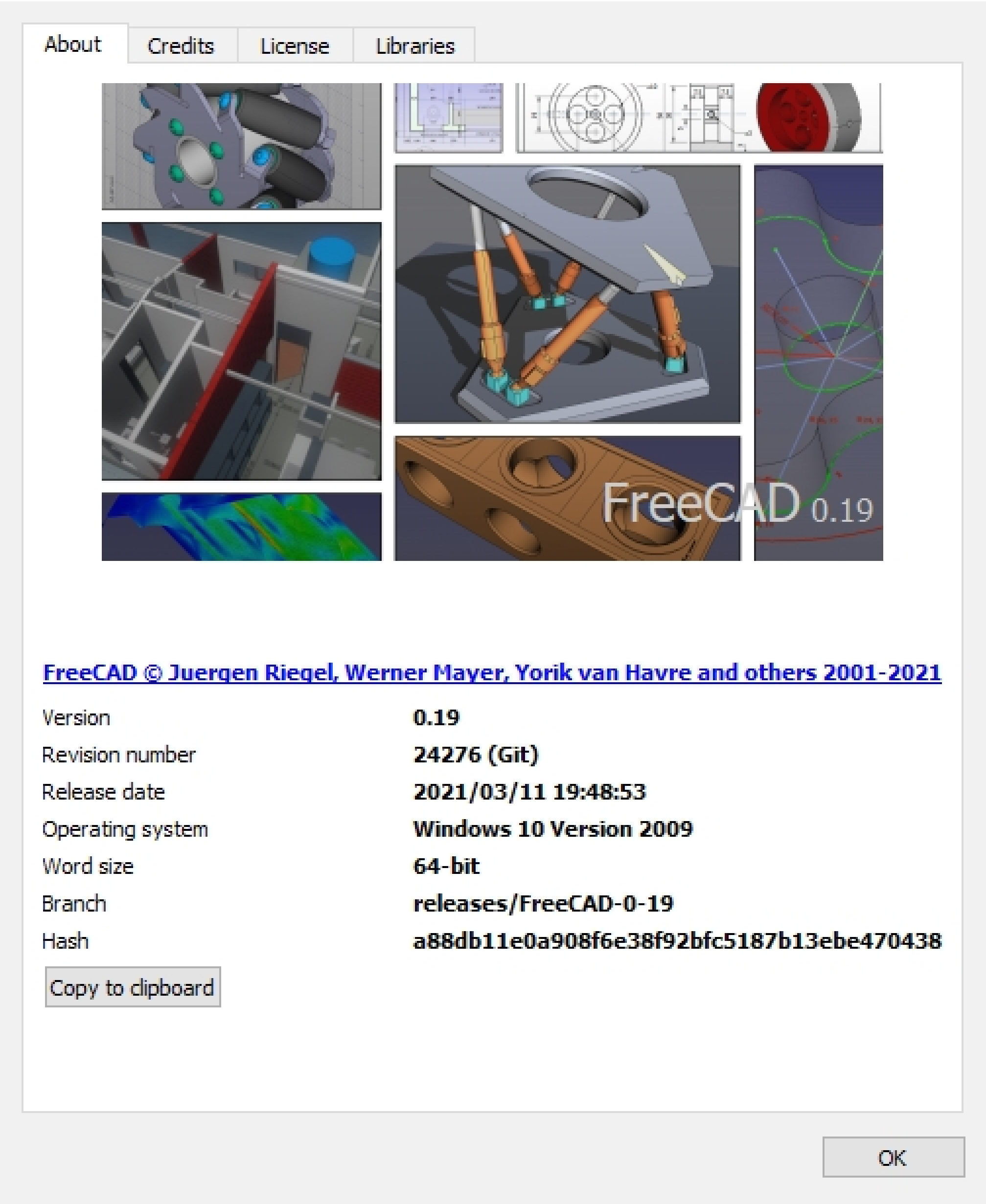
Quick Tips
For simplicity, we're using the same tool to do everything in this tutorial. However, you might want to use a different tool and different operations.
For example, you could drill the through-hole using a regular drill bit tool and a drilling operation.

When you have all your path operations set up, click the 'Simulate Path G-code on Stock' button to see a simulation of the tool performing the CNC routing. A useful check that everything is OK.

If your units don't appear in mm/min, click Edit > Preferences and then, under the 'Units' tab, select 'Metric small parts and CNC' from the drop-down 'User System' menu.

Community Forum

Asking for help with FreeCAD is something everyone who uses FreeCAD does or has done, and a great place to request assistance is on the FreeCAD community forum. The golden rule to remember is that everyone working on FreeCAD is a volunteer, so you need to do your best to make life easy, and be nice! Before you post, do a search on the forum for the topic you want to post about – the forum has been running for a long time, and chances are someone else has raised a similar discussion. You find that you barely need to post on the forum when you search first. You have also discovered all kinds of interesting approaches and techniques. Secondly, it’s really important when asking for some assistance that you describe which version of FreeCAD you are using and on which operating system. FreeCAD makes this really easy to do: just click Help > About FreeCAD, and you will see all the details of your installation listed.



There’s a ’Copy to clipboard’ button; if you click it, you can then paste those details into your forum post before going on to ask your question. You should also do this if you are answering another thread or comment with additional information relevant to the post. While you are on the forum, remember every now and again to say thanks to the developers for voluntarily creating such great tools for us to use.