FreeCAD
FOR MAKERS

Create 3D prints, laser cuts, folded sheets, and more with free design software
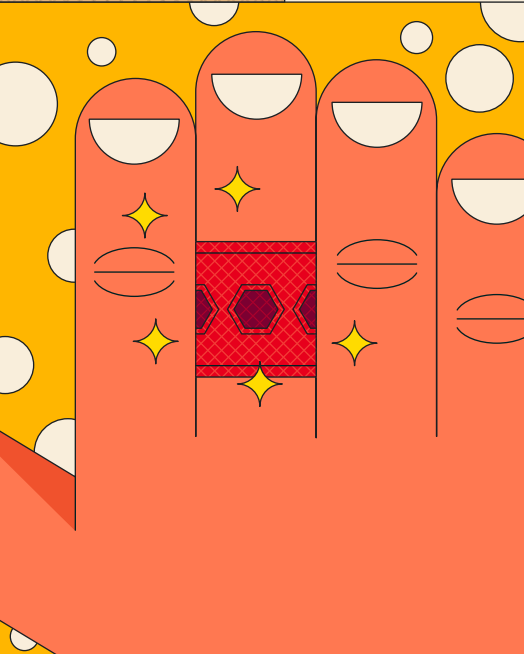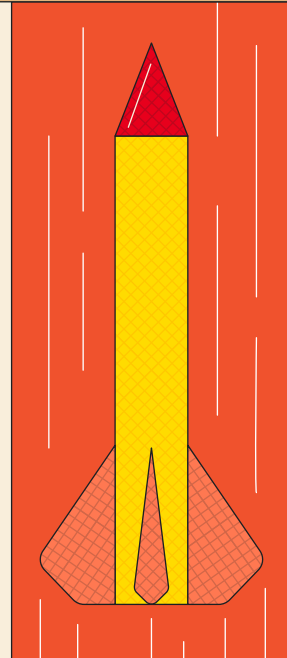
By **Jo Hinchliffe**

Foreword by
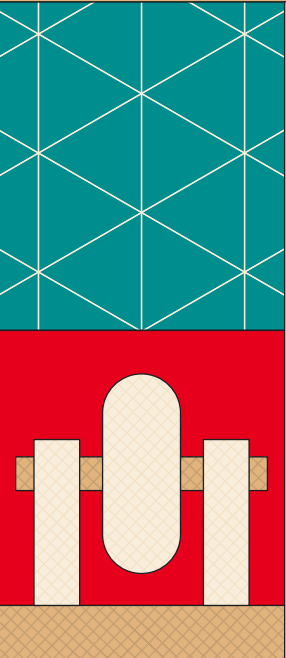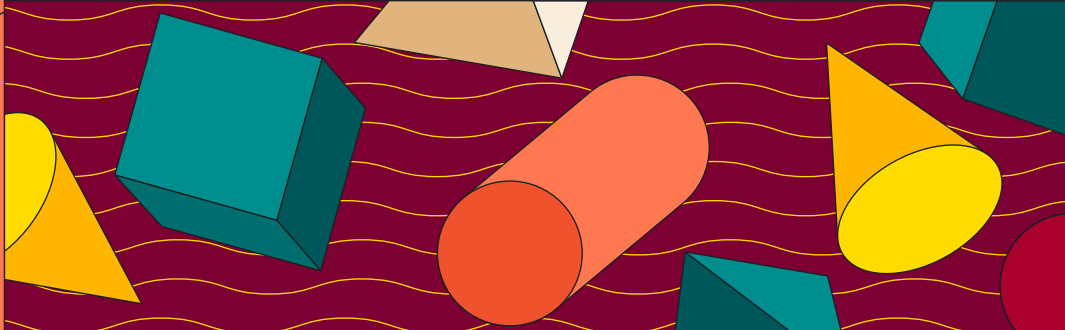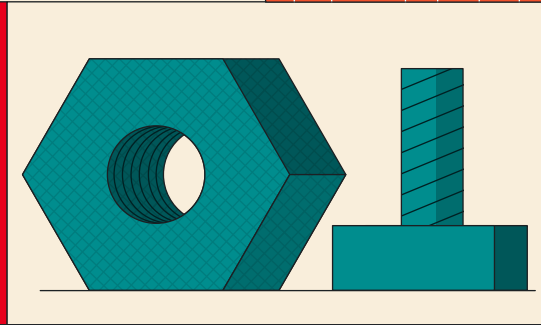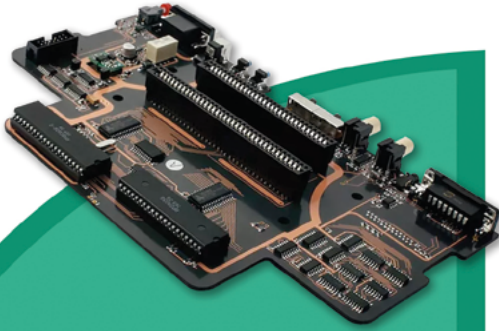**Yorik van Havre**
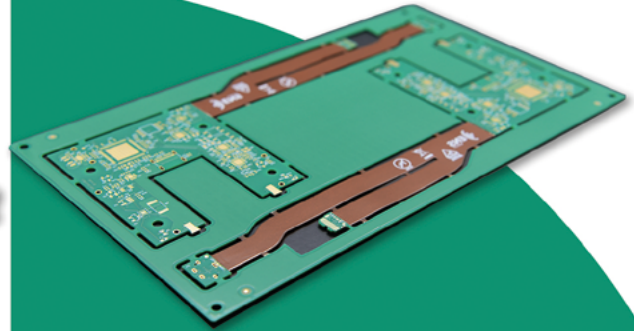
FreeCAD For Makers

Welcome to the HackSpace magazine guide to 3D design – which is really a guide to FreeCAD, the number one 3D modelling and design software. We could have chosen from a few pieces of software to base this guide on, but we picked FreeCAD for three good reasons.

It's not a clone. FreeCAD is designed from the ground up to meet the needs of users. Unlike some other free software applications, FreeCAD isn't playing catch-up with some paid-for application: it's its own thing.

It's free software. Time spent learning FreeCAD will never be wasted because no company can come along at a later date and change the licence agreement or start charging you to use it.

It's flexible. Because FreeCAD is free software, its users can add to it, writing extensions to the software to suit their needs. These add-ons, known as workbenches, cover a range of uses, including PCB design, pipes, timber-framing, architectural design, and loads more. So, whatever you're interested in building, you'll find a FreeCAD workbench for it.

> Unlike some other free software applications, FreeCAD isn't playing catch-up with some paid-for application: **it's its own thing**

All the tutorials in this guide were first published in HackSpace magazine, the magazine for the modern maker. We regularly feature 3D printing, home electronics builds, woodworking, and other projects, alongside a load of technologies far too esoteric to list here. Download a copy of the magazine for free from **hsmag.cc** and see for yourself. If you like it, subscribe.

Finally, a word on the author: Jo Hinchliffe is a prolific maker and tinkerer and full-time tool enthusiast. He's given talks at the European Space Agency (ESA), helped encourage and promote all kinds of creative communities, designed and produced open-source rockets – and particularly relevant for our purposes, he's extremely good at using simple words to describe complicated things.

ANDREW GREGORY
Editor andrew.gregory@raspberrypi.com

Got a comment, question, or thought about HackSpace magazine?

get in touch at
hsmag.cc/hello

GET IN TOUCH

 hackspace@
raspberrypi.com

 hackspacemag

 hackspacemag

ONLINE

 hsmag.cc

Available on the App Store

GET IT ON Google Play

Foreword

Yorik van Havre

FreeCAD developer

F**reeCAD has come a long way in 20 years.** It was relatively unknown during its first decade, developing slowly and steadily, then it got embraced by the 3D-printing community around 2010 and is now putting its feet firmly in the professional world. I am very fond of this grey area that exists in and around FreeCAD, where all kinds of users, professional and hobbyist alike, meet and share tools and knowledge. It is a tool made by hackers for hackers. Like what happens in hackerspaces, the distinction between what is professional and what is non-professional ceases to be meaningful.

Jo Hinchliffe's tutorials are just like that, professional yet not particularly aimed at a specific category of users. They will take you everywhere through FreeCAD and further. Even if you're here just to 3D-print a new handle for your favourite coffee jar, there is no reason for not doing it the same way as if you were launching your own coffee jar production line. That's what FreeCAD is designed for, and that's what we all hope you will like and, possibly, get hooked and be part of this big ecosystem where what we produce helps others. ❑

> **Jo Hinchliffe's tutorials are just like that, professional yet not particularly aimed at a specific category of users**

Contents

30

48

08

60

FreeCAD for beginners

Get started with FreeCAD, using some of the tools to design a set of small wheels ready to be 3D-printed

F**irst, go to freecadweb.org and download a copy of FreeCAD for your operating system.** 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 (**Figure 1**). 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 (**Figure 2**). 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.

Figure 1
FreeCAD is an excellent open-source CAD environment developed by a dedicated group of motivated volunteers

FIRST DESIGN

To begin, click 'create new' and the document viewer window will create an empty project in a new tab. Next, we are going 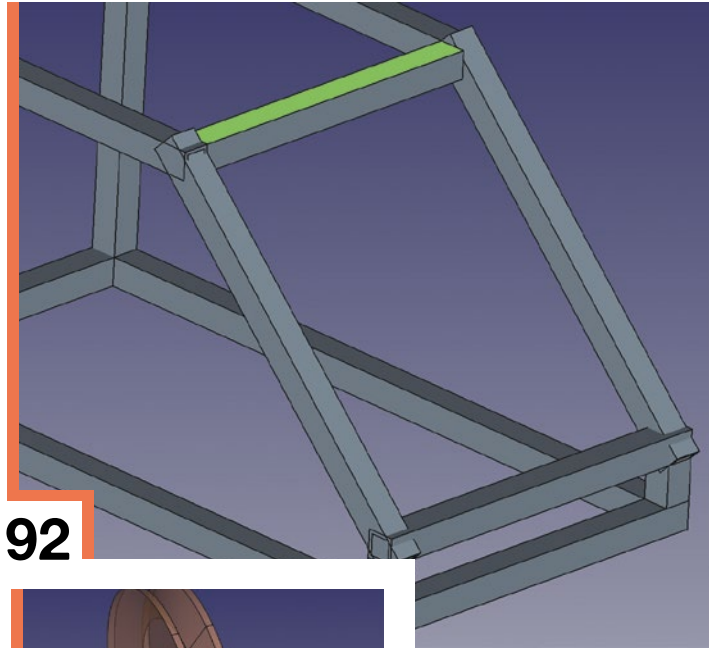to switch from the 'start' workbench to the 'part' workbench using the workbench drop-down menu. The part workbench is a good place to start to get oriented and to explore one of the simplest ways to make parts. Before we go too far, though, let's explore how we navigate, zooming and moving objects, 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 (**Figure 3**). Some of the navigational styles are based on other CAD environments, so if you are
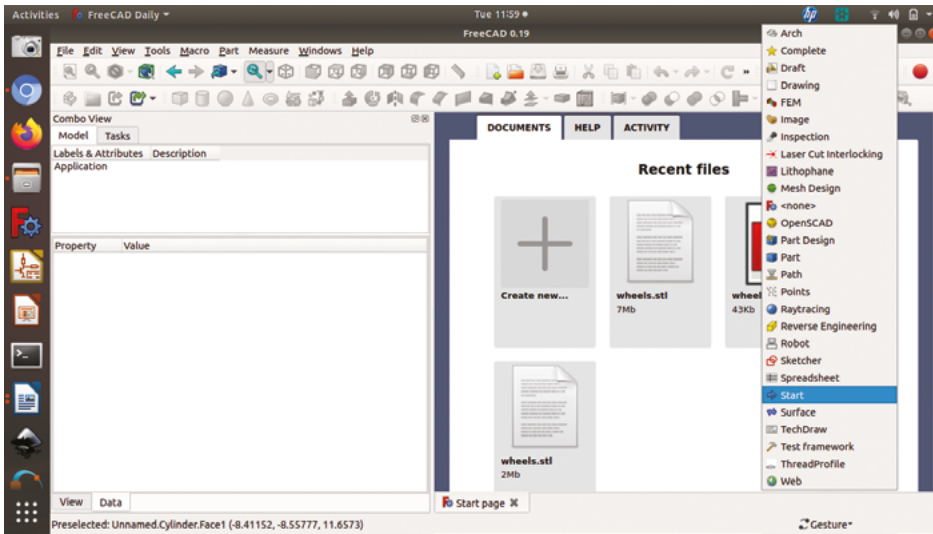
used to using, for example, Blender, then you might prefer that style. We use the 'gesture' style option that gives left-click for rotation in the document window, right-click for moving, and the centre button/wheel is for zooming. Of course, you need an object like the cylinder we created to allow you 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 we
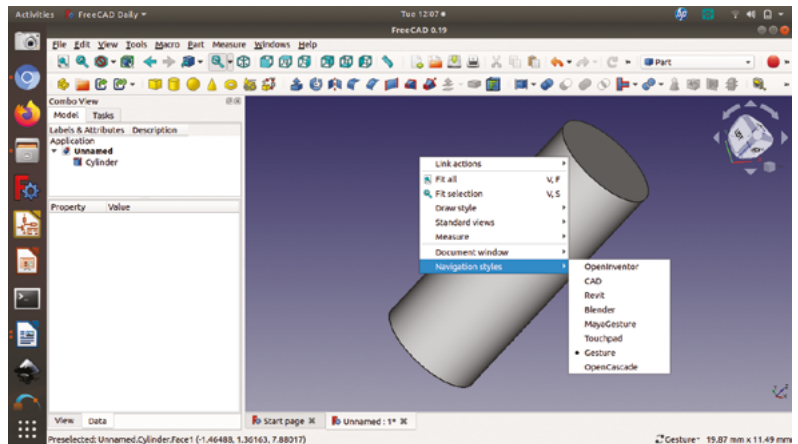
> **You need an object like the cylinder we created to allow** you to judge which navigational style suits you best

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 we created, we can see that on the left-hand side of the screen there is a file-tree-type view, and under the

FREECAD VERSION

For this series, we use FreeCAD version 0.19, but the same processes should work on a newer version, though as time passes, it's possible that some things will change.

We are using the app image version on Ubuntu 20.04; however, this should be the same for the 0.19 version on other operating systems such as other Linux distributions, Windows or Mac.

heading 'unnamed project', we can now see that 'cylinder' is listed, (**Figure 4**, overleaf). We can single-left-click on this cylinder label in the file view to select the object, or we 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 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 we 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 our other cylinder and is currently positioned inside it. If we 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 our first cylinder, which was 15 mm. If we click anywhere off the image of the cylinders in the viewer window, we should now be able to see the ends of the cylinder we created in the top and bottom of the original cylinder (**Figure 5**).

SLICING AND DICING

Similar to most drawing and CAD packages, we can combine objects in multiple ways to create new objects. You should be able to see a selection of tool

> **Similar to most drawing and CAD packages,** we can combine objects in multiple ways to create new objects

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'. We are going to use this tool to cut the

1.5 mm radius cylinder we just made out of our 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** and **Z** to undo and try again. The shortcuts may be different on your OS. 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 we wanted to change the radius of our hole to 2 mm from its current 1.5 mm, we 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 our hole visible in the viewer window. We 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 our cut with the new-sized cylinder. We can also change the position of the hole if needed by moving the object.

WHEELY GOOD

Our 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. We are going to perform another cut with this object, but we want to move it to the top of the wheel design. To do this, right-click on the cylinder object in the file
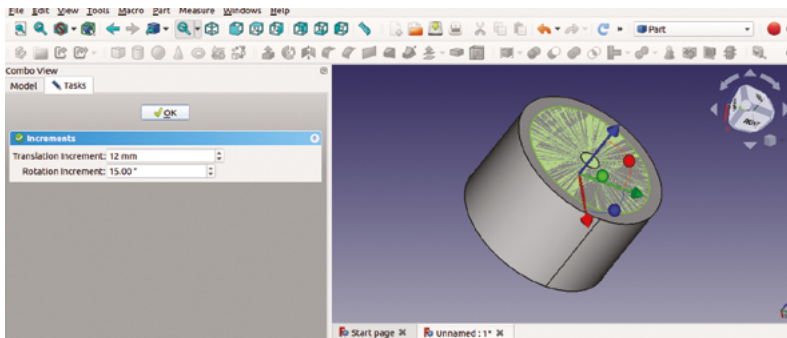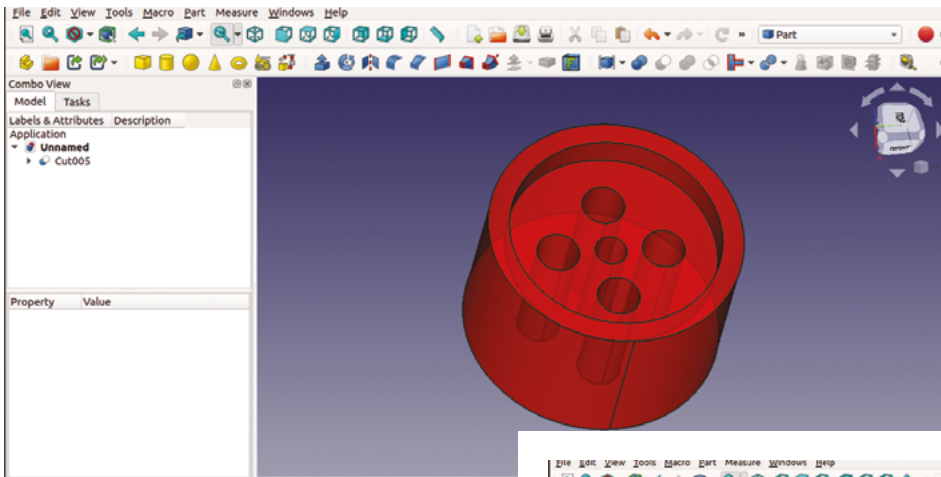
Figure 4 ◆
The combo view panel on the left-hand side includes a file tree window for selecting objects and parts. When an object is highlighted, its dialog box appears below

Figure 5 ◪
It can be hard to see objects when one is positioned inside another. Hovering over object names in the file tree will make them appear highlighted in the viewer window

Figure 6 ◈
When moving objects, or rotating them, we can set the increments to allow us to quickly position objects accurately

tree and then select 'transform' from the drop-down menu. You should now see a dialog box called 'increments', and in the viewer window, the cylinder will be highlighted with some red, green, and blue arrows attached. These arrows correspond to the x, y, and z axes, and left-clicking and dragging on either the red, green, or blue arrow points will move the item in that direction. It will move the object in multiples of the amount set in the 'translational increment' box, (**Figure 6**). This usually defaults to one millimetre, but we can set it to any step size we prefer. For example, in our case, if we set the translational increment to 12 mm, we can then drag the blue arrow by one step, bringing our 3 mm-high cylinder flush with the top of our design. Whilst we don't need it for our wheel, it's worth having a play with clicking and dragging the red, green, and blue spheres on the arced lines with the translation arrows. Clicking and dragging these rotates the object around the axis the colour represents. Again, you can set the rotational increment dialog to whatever number of degrees you like.

Next, perform a cut to remove the new cylinder from the design. We then continued to make our wheel look a little fancier by creating four more cylinders. We moved the four cylinders equally outwards by 5 mm from the centre of our wheel using the moving techniques we learnt earlier. You should end up with a simple wheel that looks like **Figure 7**.

FINISHING TOUCHES

Our wheel is looking wheel-like, but the part workbench has some simple tools we 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 our wheel design, that doesn't apply here. The part workbench has two nice tools for adding
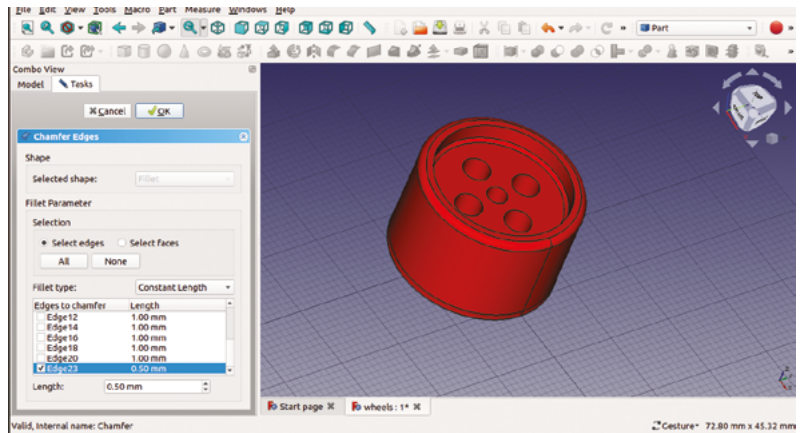
either fillets or chamfers to edges of objects. They appear as two tool icons: the fillet tool as a curved area between two flat surfaces, and the chamfer tool as an angled flat surface. In the viewer window, left-click to select the outside edge of the top of our wheel, and then click the fillet tool. In the dialog box, we can see a list of edges, including our 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. →

We'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 our wheel. Click the chamfer tool this time, and set the chamfer length to 0.5 mm and OK it (**Figure 8**, overleaf). You should see that a nice internal chamfer has been created. Finally, we added a matching fillet to the bottom outside edge of our wheel.

RENAME

Now we have our basic wheel design (it's called 'fillet' in the file hierarchy). Let's create some simple copies of our wheel to make a set of four.

Figure 10 ◈
Our finished set of wheels, ready for export

Figure 11 ◈
Our wheel design, exported and 3D-printed

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 we double-click it, it won't open up the fillet dialog, but will help us track when we make multiple items.

CREATE COPIES

To create our copies, single-left-click 'wheel' in the file tree view to select our wheel and then go to the 'part' drop-down menu; select 'create a copy', then 'create a simple copy' (**Figure 9**, overleaf). Clicking this option will create a new 'wheel001'. If we double-click on wheel001, we can see that it can now be moved as a separate item. As we made a 'simple copy', our new wheel doesn't contain the hierarchy of operations and objects our original has and also is not dynamically linked to the original object. This means that if we make further changes to our 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.

THE NEXT STEP

Let's make copies and lay them out as a set of four (**Figure 10**). We're going to 3D-print these wheels, and we could just export one instance of the wheel and then duplicate it in our slicer software – but, again, it's just as easy to do this in FreeCAD. Having made our four copies of the wheel, let's save our work and then, click 'file – export'. We can export a mesh file of the wheels ready for slicing for 3D printing. If we don't specify the type of mesh file by adding a file suffix, it will default to an STL file. Having exported our STL file, we could open it in our favoured slicer and 3D-print them (**Figure 11**). However, FreeCAD isn't only for 3D printing, and in in the rest of this guide, we'll look more widely at what FreeCAD can do. ▫

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.

Wireframe

Join us as we lift the lid
on video games

Visit **wfmag.cc** to learn more

FreeCAD: using the part design workbench

Our next project: we're going to design a small bird feeder
using the 'part design' and 'sketcher' workbenches

**n part 1, we began to look at FreeCAD
using the 'part' workbench to design a
small wheel.** We used simple primitive
shapes, cubes, and cylinders, and we
added, subtracted, and joined them together
to create our design. In this project we are
going to make a small bird feeder, but will use
different methods where we will create a 'body' using
the 'part design' and 'sketcher' workbenches. The
techniques are a little more complex, but give us
much more control over our designs.

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 should then see 'create sketch'; click
on this, and you'll then be presented with a choice of
which plane to base our 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 we sketched onto the printer bed, we are
then 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 which
is 7th from the left on the top row in **Figure 1**. We are
going to draw three connected lines in a U-shape
roughly laid out, as shown in **Figure 2**. 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 we 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 our
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
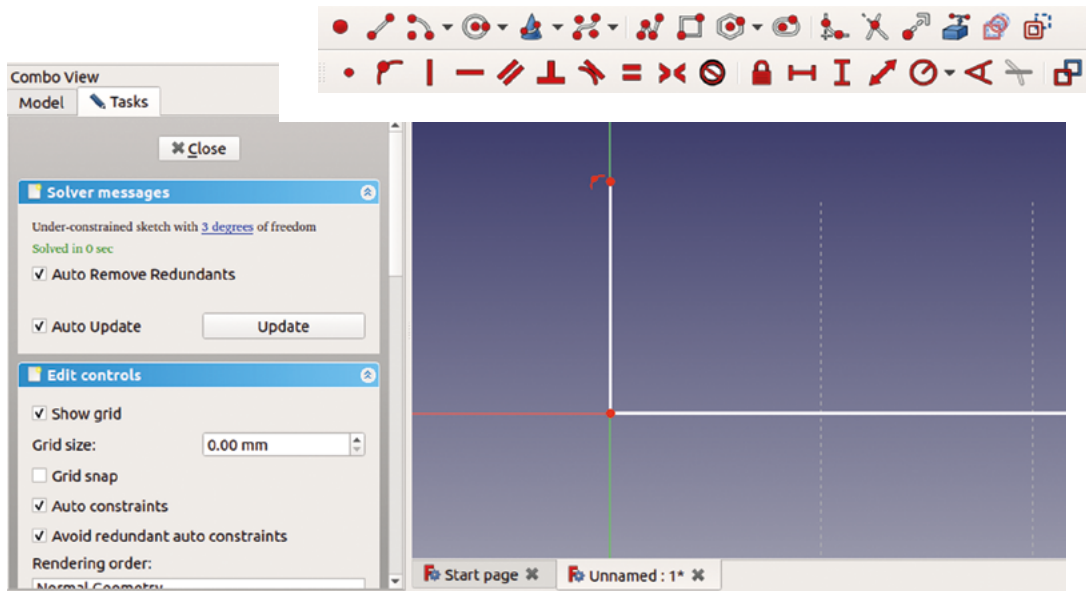
Figure 1
The icons on the top line are mainly the tools for drawing lines, circles, rectangles, and more. The icons on the bottom line are the tools to add different types of constraints

Figure 2
Our first three lines drawn roughly and unconstrained

Figure 3
Our first three lines constrained

information about a sketch item's positions and dimensions. This is an incredibly useful approach, meaning we can return to sketches at any point in our 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 we have just done. Starting on a datum line and including the origin point on the XY plane also means that we don't need to constrain these lines' positional co-ordinates, but we do need to constrain the lengths of the line. Later we 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.
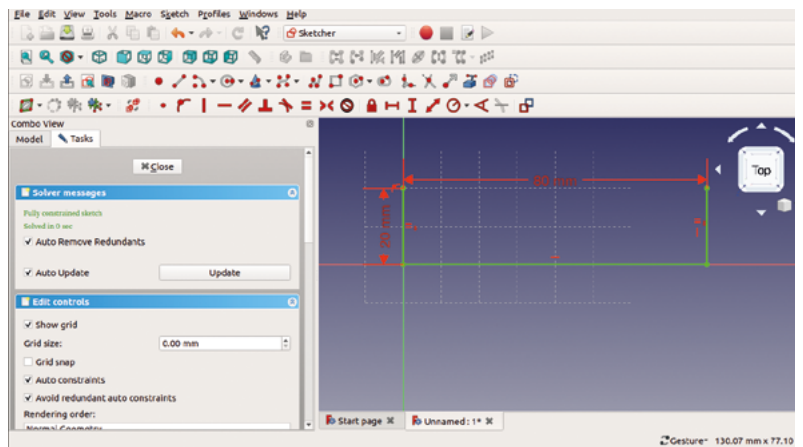
WHAT IS A BODY?

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 we 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 project, we are using sketches and performing operations on the sketches such as 'padding' where we extrude a sketch, and 'pocketing' where we 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 in part 1.

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 our lines, we are going 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 (8th from left, top row, in **Figure 1**). 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 one of the →
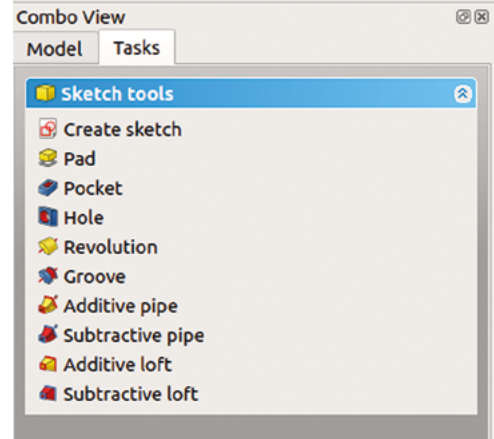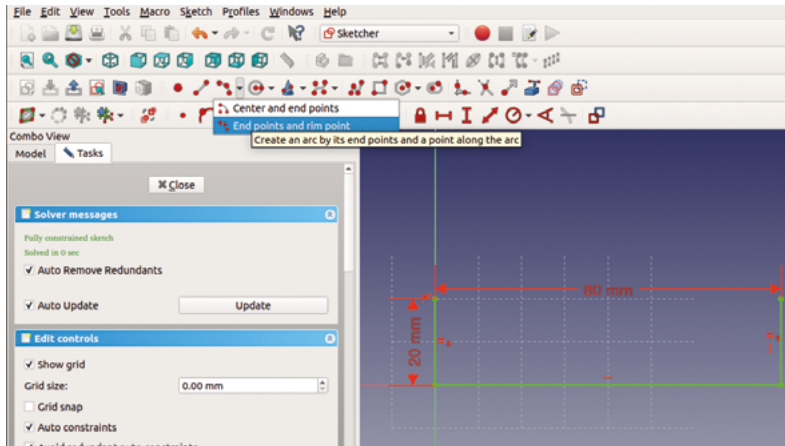
FreeCAD, Using the Part Design Workbench

vertical lines, and then click the 'fix the vertical distance' tool icon (13th from left, top row, in **Figure 1**). 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.

> **The first job is to make a pocket** for our bird feed to be contained in

Next, select the horizontal line and click the 'fix a horizontal distance' tool (12th from left, bottom row, in **Figure 2**) – 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 (**Figure 3**).

Figure 4 ◆
Some drawing tools have drop-down menus and options for their use, such as the 'create' and 'arc' tools

Figure 5 ▨
The part design workbench task menu tab

Figure 6 ◆
Selecting a face and clicking the 'create sketch' tool to launch a sketch attached to that face
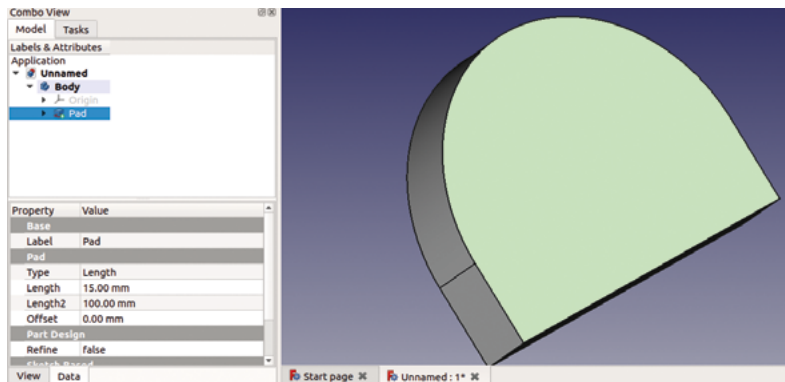
A BIT SKETCHY

We are going to add more to the sketch, though. Let's click the 'create an arc' tool icon, but click its drop-down menu and scroll to the 'end points and rim point' option (**Figure 4**). 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 – we 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 (15th from left, bottom row, in **Figure 2**), and in the dialog box, set the radius to 40 mm. We should now have a fully constrained sketch, with all lines turned green.

This sketch is the main outline of our bird feeder, so click 'close' on the sketch solver panel in the combo view window, and we will 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 (**Figure 5**).

For the rest of this project, we are going to create more sketches, but these sketches are going to be drawn onto faces of the pad object we have just made. The first job is to make a pocket for our bird feed to be contained in. So, left-click in the viewer to select the upper face of our object (**Figure 6**). Then click the 'create sketch' tool icon which should now bring you back into the sketcher workbench with the face of the pad we selected in view. Click the 'create a circle' tool icon (4th from left, top row, in **Figure 2**). Left-click and drag and left-click again to draw a circle
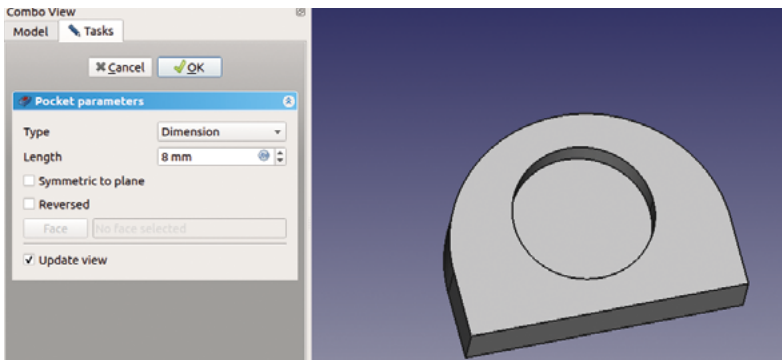
of any size anywhere in the sketcher viewer. Use the 'constrain an arc or circle' tool we used earlier to set the diameter to 20 mm. Next, click to select the centre point of your circle, and then click the zero point of the axis at the bottom of the left-hand vertical line. To position our 20 mm circle, click the 'horizontal distance constraint' tool we used earlier and set the distance to the midpoint of our 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 our circle as a pocket into our bird feeder design. We set the depth of our pocket to 8 mm (**Figure 7**).

A STICKING POINT

To attach our bird feeder, we are going to create a pocket and a slot which we used to fit some rubber window suckers that we found in our parts drawer. The arrangement of the slotted pocket could allow the bird feeder to slot onto a couple of small screws. In the part design viewer, we rotated our bird feeder and selected the bottom of the design opposite to the pocket we had just created. As before with this selected, we clicked the 'create a sketch' tool. Next, select the 'create a rectangle' tool icon (8th from left, top row, in **Figure 2**). Draw two rectangles of any size anywhere in the sketcher window. Conveniently the rectangle drawing tool constrains the vertical and horizontal lines, so all we have to add is the line lengths and the positions. We made our rectangles 10 mm by 6 mm. Then we used the zero axis point to locate the closest rectangle to it, 10 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 we have done with other items). However, to position the other rectangle, we used a clever little function. By clicking the 'create an edge linked to an external geometry' tool (14th from left, top row, of **Figure 2**), we can select and import lines from the underlying
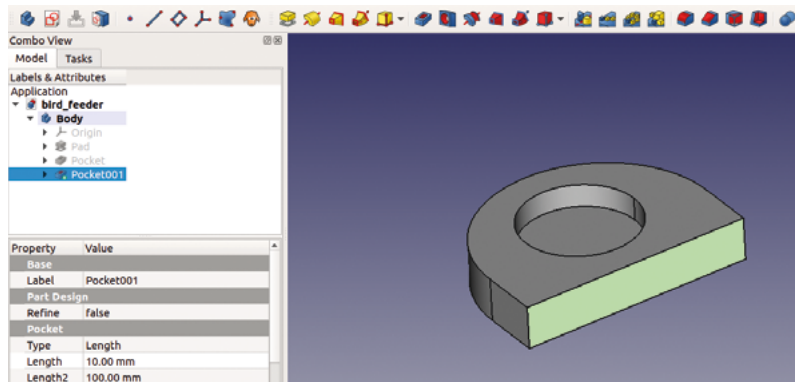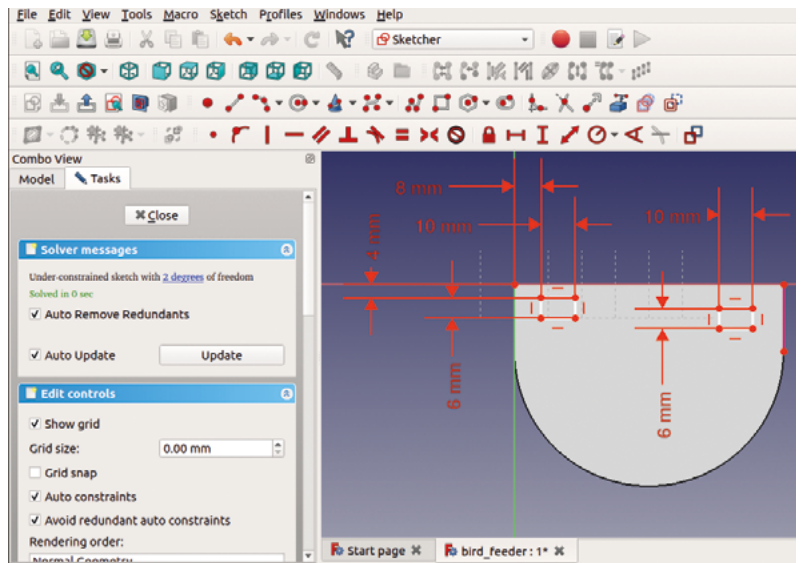
object which aren't in this sketch. We hovered over the 20 mm vertical line at the far edge of our bird feeder and selected it, once highlighted. This now creates an edge and points at either end to which we can attach objects or constraints. This means that instead of calculating the position of our other rectangle from the XY zero origin point, we can now constrain this by saying it is 10 mm and 4 mm from the opposite corner of the object; you can see where we connected that rectangle tool in **Figure 8**. →

17
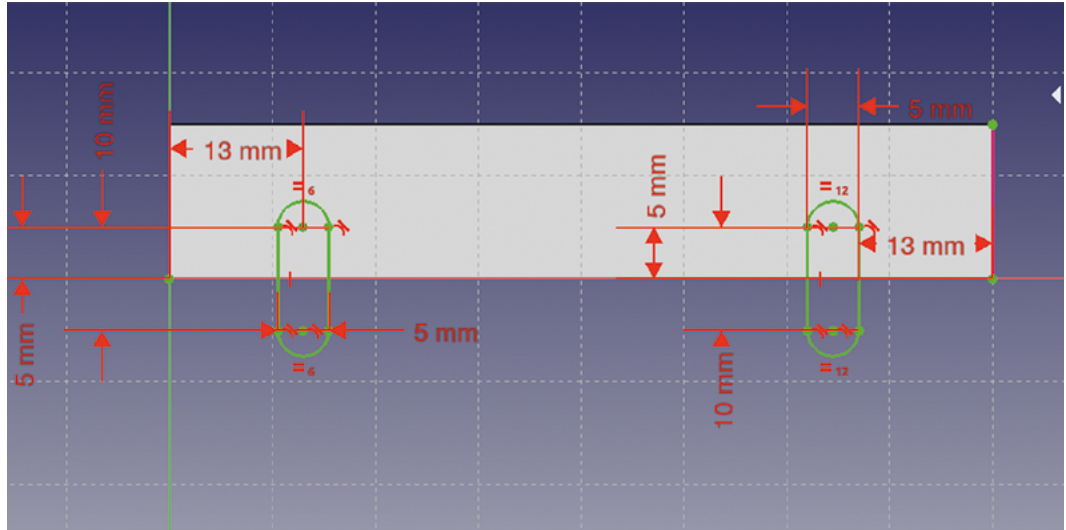
Having constrained our rectangles sketch, we closed the sketch and performed another pocket operation to a depth of 10 mm. Next, we need to cut slots that go into these pockets, so we selected the face of the back of the bird feeder object that we need to sketch on and created a new sketch (**Figure 9** overleaf).

FINISHING OFF

In this sketch we again imported and created an edge for the far side of the object and then used the 'create a slot in the sketch' tool (10th from left, top line, in **Figure 2**). 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 we opted to select two opposite points on either side of the slot and constrain the horizontal distance between them to 4 mm. We then constrained 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

> **We closed the sketch and performed another** pocket operation to a depth of 10 mm

need to constrain the arc, and so we only needed to position the slot sketches in the correct places to coincide with our rectangular pocket. We used 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 – you can see these values in **Figure 10**. We closed the sketch and again performed a pocket, setting the depth to 5 mm, to cut into our rectangular pocket, but not deeper into the object.

Finally, we 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 (**Figure 11**). We covered usage of the fillet and chamfer tools last month in the part workbench, but they work similarly on the part design workbench.

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. ◻

FreeCAD, parametric parts, and other approaches

Design a variety of objects using numerous different tools and approaches

n this tutorial, we're going to add to our skills by exploring a small range of tools and approaches and making a variety of small project items. We'll start by exploring the 'Add a Thickness' tool, look at using the Spreadsheet workbench, and working parametrically, we'll bring these skills altogether to make an automatic box with lid generator.

For a quick example of using the 'Make a Thick Solid' 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 **Figure 2**.

Close the sketch and then pad the sketch using the pad dialogue found on the tasks tab in the combo view window or by clicking the 'pad a selected sketch' tool icon.

We are going to now use the thickness tool to hollow out our hexagonal extrusion. Select the upper face of the hexagonal extrusion and then click the 'Make A Thick Solid' tool. You should see instantly that the part now appears as a hollowed version of itself. In the thickness tool dialog, you can see the selected face and change various parameters (**Figure 3**). 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 30 mm you now will have an object that is 30 mm plus the thicknesses. If you want to create an object that matches the external dimensions of the underlying geometry, click the 'make thickness inwards' tick-box. You can also swap between Arc and Intersection, which essentially
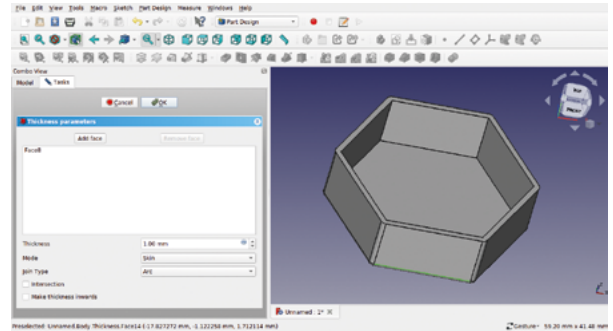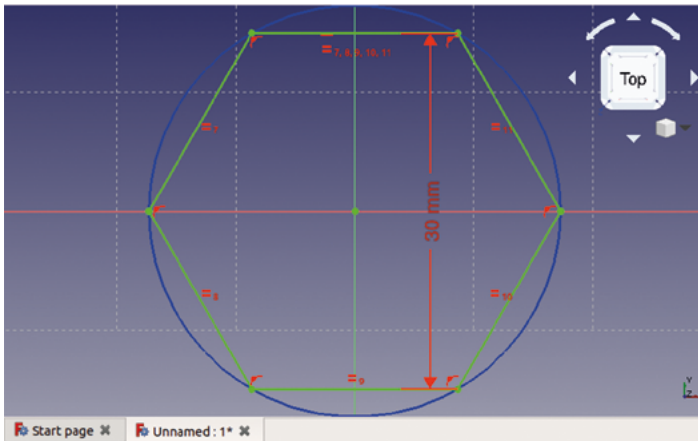
toggles between creating filleted edges or sharp edges on the thickness.

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.

LET'S GET PARAMETRIC!

We looked at making multiple 'simple copies' of parts using the 'part' menu on the part workbench in previous parts of the series. You can also use that approach to create simple copies of bodies that you have designed in part design. Sometimes we might want to make multiple similar parts, but with one or more dimensions changed – for example, if we were drawing a model of a 3D printer that used the common 2020 aluminium extrusion (**Figure 1**).

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. In the next part, we'll begin to look at 'Lofting' tools, which enable us to create hollow parts with curved walls and more.

One way we 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, we might choose to use the part design workbench to create a body for our 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 we can change the model geometry by adjusting parameters such as dimensions. Whilst only being one parameter, we can use this mini project to learn about using a spreadsheet to store and alter parameters for objects and sketches.

We downloaded an image with the dimensions of the 2020 extrusion and created a new project. On the part design workbench we selected to create a new body and then to create a sketch in the XY plane.

> **Parametric work in CAD is where we can change** the model geometry by adjusting parameters

We crudely drew a shape of the profile using the polyline tool and then worked through to constrain the sketch (**Figure 4**).

Having constrained the sketch, we checked how it looked by performing a 'pad' from the tasks menu, and we then added the external corner radius fillet and the chamfers at the edges of the extrusion slots. We 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 we 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 →

QUICK TIP

Often in this article, we have described tools using the text that appears when you hover the mouse over the tool icon. This is a great way to find the tools or explore a new workbench.
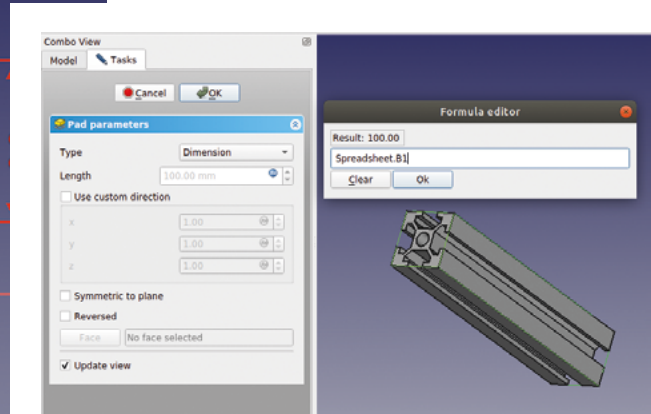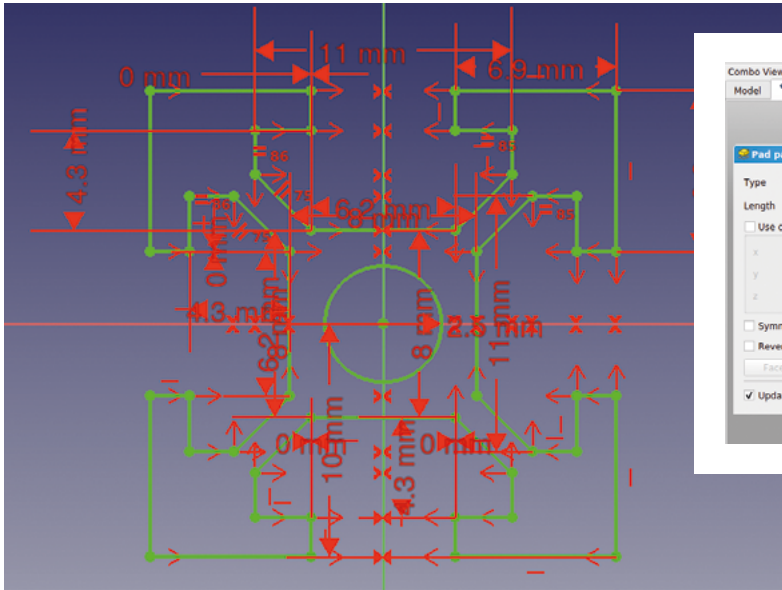
Figure 4 ◆
A basic sketch of
the 2020 profile. We
added chamfers and
fillets once the sketch
was padded

Figure 5 ◆
The spreadsheet
workbench, with our
spreadsheet opened
in its own tab in the
preview window

Figure 6 ▣
Clicking the circular
blue box in the corner
of any input box takes
you to the formula
editor, where you can
link the value to a
spreadsheet, amongst
other functions

'create a new spreadsheet' icon. You should now see a spreadsheet open in the preview window in a separate tab (**Figure 5**). 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, we are going to write the label 'Length' in cell A1, and then in cell B1 we can type a length we want our extrusion to be. When we 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 our case, this is millimetres and will be the case for our spreadsheet

values, so we don't need to define our input units in the spreadsheet.

Set cell B1 to 100 and 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 (**Figure 6**). 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, we need to input the cell location of the cell from which we want it to take its value. If you type 'B', it again

> **This autosuggestion is useful when we come to projects** with numerous cells of data in a spreadsheet
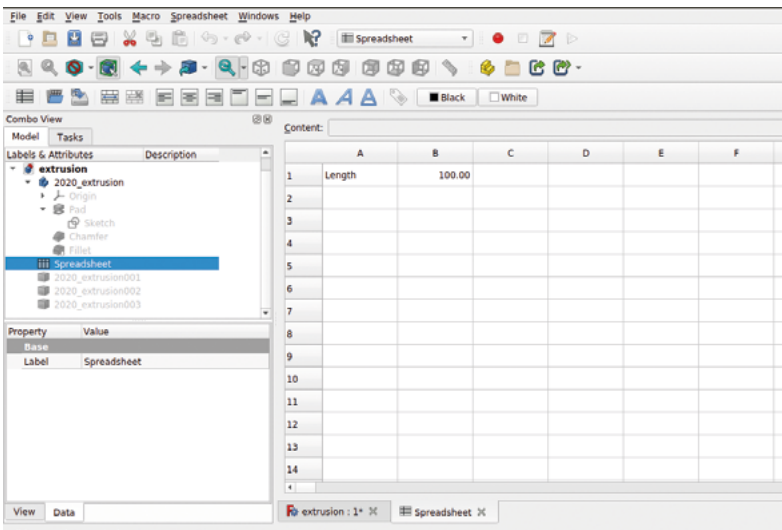
should automatically suggest 'B1' as a value. This autosuggestion is useful when we come to projects with numerous cells of data in a spreadsheet, as it will only suggest cells that have data in them. Click the 'B1' so that the value in the formula editor reads 'Spreadsheet.B1', as in **Figure 6**.

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, as in **Figure 1**.

Having explored using both spreadsheets and the thickness tool, let's combine our new skills and create an automatic parametric box generator. This simple

Content: =B1 * 2 + B6 * 2 + B2 — Alias:

	A	B	C	D	E	F	G	H	I
1	thickness	2.00							
2	box_length	40.00	lid_length	45.00					
3	box_width	45.00	lid_width	50.00					
4	box_height	40.00							
5	Lid_depth	12.00							
6	clearance	0.50							
7									
8									
9									
10									
11									
12									
13									
14									

Figure 7
The spreadsheet set up for our box generator

Figure 8
Using the spreadsheet and the thickness tool, we have a parametric hollow box ready to make a lid for

project makes it easy to create a box and lid of any size that we 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 our box, we are going to need four parameters: the box length, width, height, and the wall thickness. Create labels for these parameters in cells, and in adjacent cells input some initial values (**Figure 7**).

Next, jump to the part design workbench and create a body and a sketch in the XY plane. We 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 upper left-hand node, and the vertical Y axis line. Repeat this for the upper right and lower right nodes, and the X axis line. 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 our case was B2. Repeat this for the vertical line to set the box width, 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 in the spreadsheet. Next, we will click the upper surface of our box and use the thickness tool we explored earlier to hollow out our box. To set the thickness, we are going to link the thickness value in our spreadsheet. All being well, you should now have a hollow box in the preview window (**Figure 8**).

To make a lid, we are going to do the same process as we did to make the box section, but we are going to use some simple formulae so that the lid is generated to fit whatever dimensions are of the box
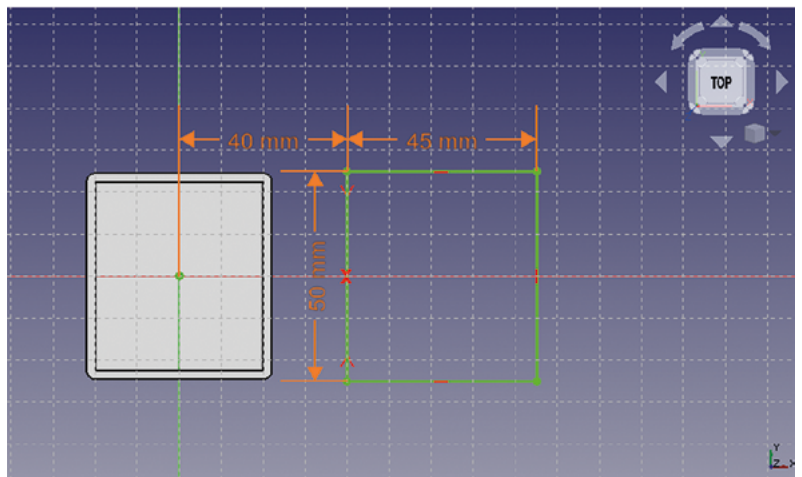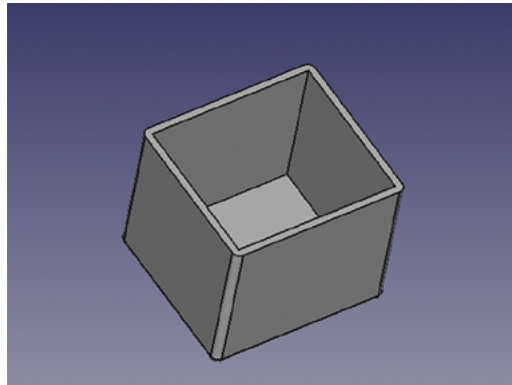
created. On the part design workbench, click to create a new sketch on the XY plane. We don't need to import any geometry from our first sketch, but let's draw another rectangle and place it along the x axis so it isn't on top of our box base. To positionally constrain this lid rectangle, select the upper left corner node and the sketch origin point at 0,0 and set a horizontal distance constraint. For the value of this constraint, we have linked the spreadsheet cell that holds the box_length value. This means that the lid →

Figure 9
Laying out the sketch for the box lid, making sure it will always be positioned away from our box

23
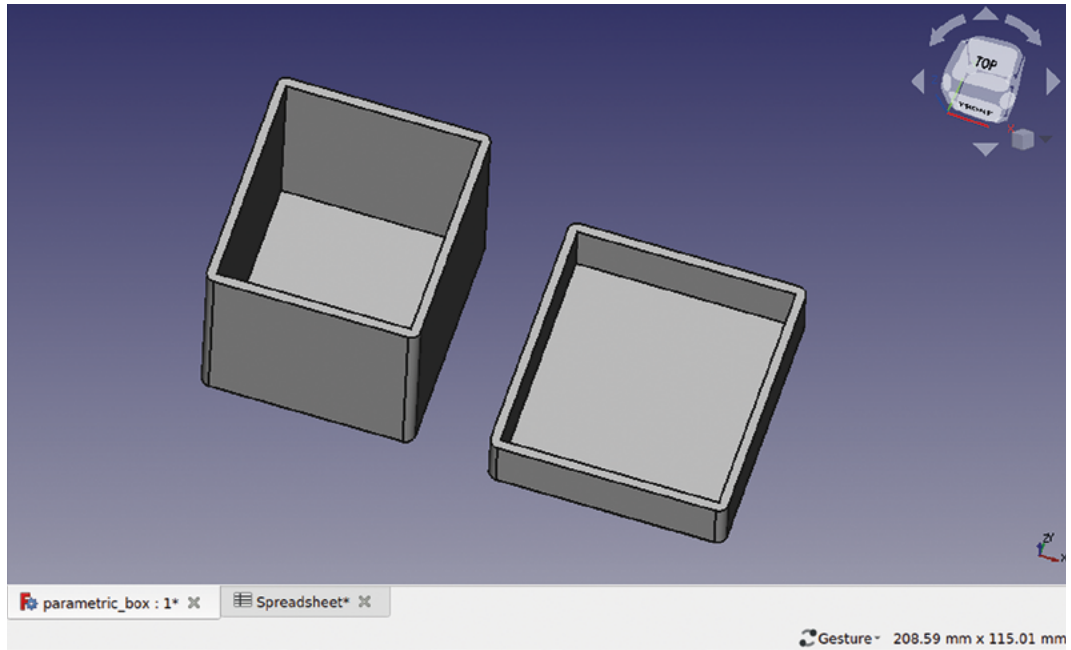
Figure 11
The box now complete with a fitting lid laid out on the XY plane

Figure 10
Adding a small formula to a cell to automatically generate the dimension needed for the lid length

will always be away from the box with a gap in between them (**Figure 9**).

Next, select the upper right and lower right nodes and the x axis line, and create a symmetry constraint. To create the length and width of the lid, we are going to create two new labels in the spreadsheet and create two new cells to hold this value. However, we are going to insert a small formula in the cells to generate these values. The value for the length of the lid will be equal to the box length value plus twice the value of the wall thickness. It will also need a clearance to make the lid fit over the box. We have created a label and a cell called 'clearance', and put in a value of half a millimetre which we will add to each side of the lid. So, to create the value of the lid_length in that cell, we input the formula to multiply the thickness and the clearance by two and add them to the box length. To add this formula, highlight the target cell and type it into the formula bar (**Figure 10**).

To finish the lid, we added the formula for the box width, using the same formula for length, except using the box width dimension. We also created

an input value for the lid height. We constrained the length and width of the lid sketch by adding constraints linked to our calculated dimensions in the spreadsheet. We 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 (**Figure 11**).

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.

FreeCAD: let's get curvy!

Add bendy transitions to your parts for a more elegant look
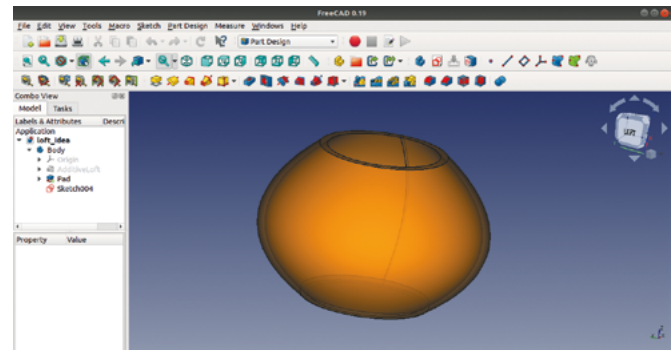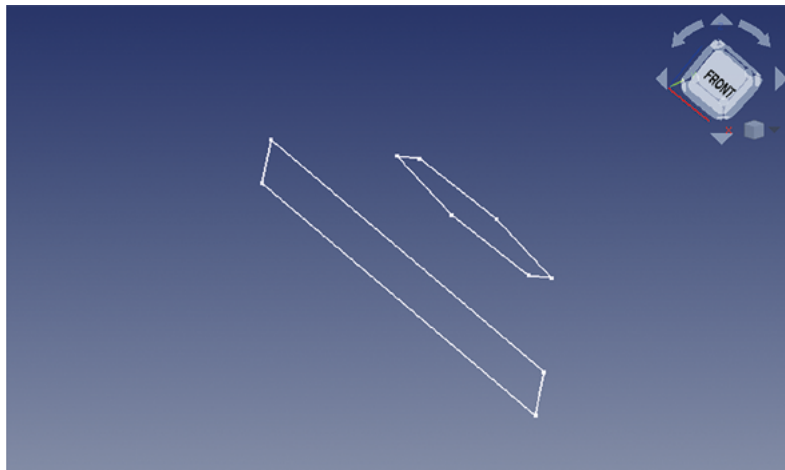
First up is the Additive Loft function, which is a method that creates straight or curved transitions between sketches – it's a great way of creating complex flowing shapes. To start with, we will make a nonsense object just to see how it works, and then we will create a small ornamental bowl that can 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, we 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 'New sketch' button, and select the XY plane for our 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, we 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, we can't move the basic position of the hexagon sketch, but we 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 we went for 20mm. If you are in an isometric view in the part design workbench, you should see the hexagon sketch rise above the rectangle sketch (**Figure 1**).

Next, select the original rectangle sketch in the file tree view, and then click the yellow and red 'loft a
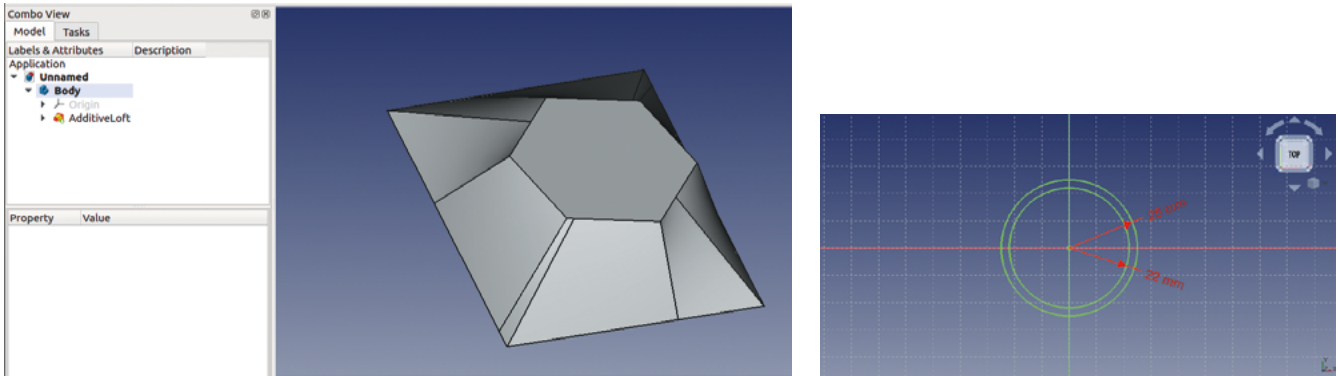
selected profile through other profile sections' tool. In the combo view panel, you should now see a window – click the 'Add section' button. Having clicked that, select the hexagon sketch in the preview window. As you do this, you should see a preview of the lofted part connected between the rectangle 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 our simple example, but we will use this in our 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 we 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 – the bowl pictured in **Figure 2** – and show how we 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 our object. We made our 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 (**Figure 4**).

Close the sketch and then create another sketch in the XY plane. Create a larger pair of circles, but with the same wall thickness – we 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 we drew.

With the second sketch highlighted, move these circles up the Z-axis using the same method we used

earlier with our hexagon. Move it up by 25 mm. Next, do the same with the third sketch, moving it up the Z-axis 60 mm (**Figure 5**). 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, as shown in **Figure 2**.

Adding one final sketch of a circle matching the outer diameter of the lowest sketch in our bowl means that we can pad/extrude to add a base to our ornamental bowl.

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 our skill set. →
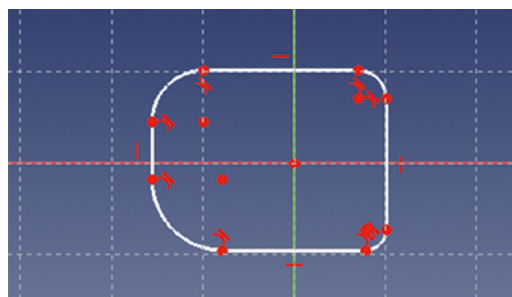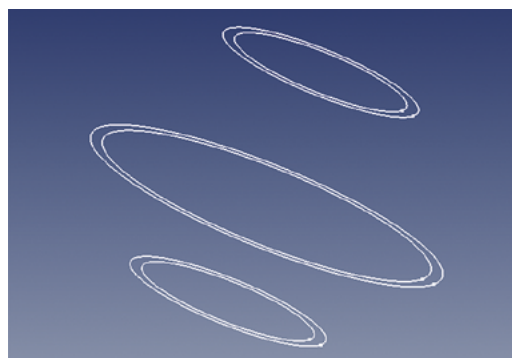
Figure 3
Our completed loft experiment

Figure 4
In this approach, the two circles in each sketch represent the wall thickness of our bowl design

Figure 5
The three sketches for our lofted bowl laid positioned on the Z-axis

Figure 6
The first sketch which forms the profile for our curvy extrusion using the sweep function
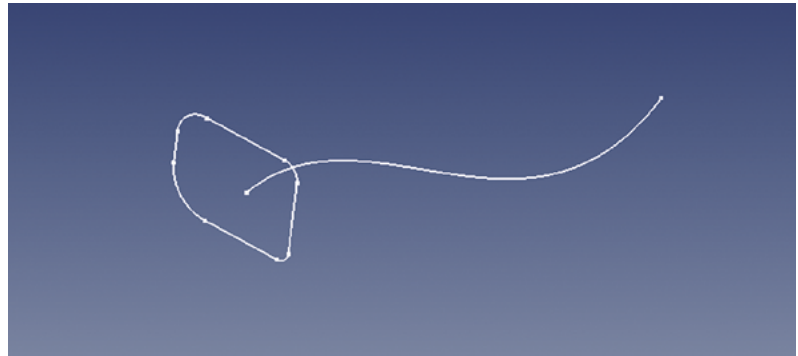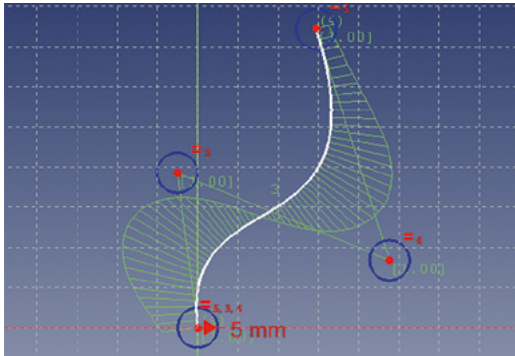
Starting on the part design workbench, create a new project, create a body, and create a sketch in the XY plane. In the blank sketch, make a small closed shape around the origin point. We drew a square and then added some fillets in the corners by clicking the 'create a fillet between two lines or at 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 our item we extrude by sweeping it along a path (**Figure 6** overleaf). 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, we are going to create the path along which our 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 we created (**Figure 7**).

Close the sketch, and in the preview window, you can rotate your view a little and see that the curved edge we created starts at the origin point, which is in the middle of our profile sketch (**Figure 8**). To sweep the profile along the curved path, highlight the first sketch we 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, we need to select the path or object along which to sweep. We can do this in a couple of ways. We can either click the 'Object' button under the 'path to sweep along', or we can click the 'Add Edges' button. Having clicked either of these buttons, we can then select the curved path in the preview window. You should now see a brown preview of the profile swept along our path. Click 'OK' to finish the sweep. You should now have a nice swept object in the preview window (**Figure 9**).

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 we will

Figure 7 ◆
A sketch in the XZ plane using the 'create a B-spline' tool

Figure 8 ⬛
The profile sketch, and the path to sweep the profile along

Figure 9 ◆
The completed object with the profile sketch swept along the target path sketch
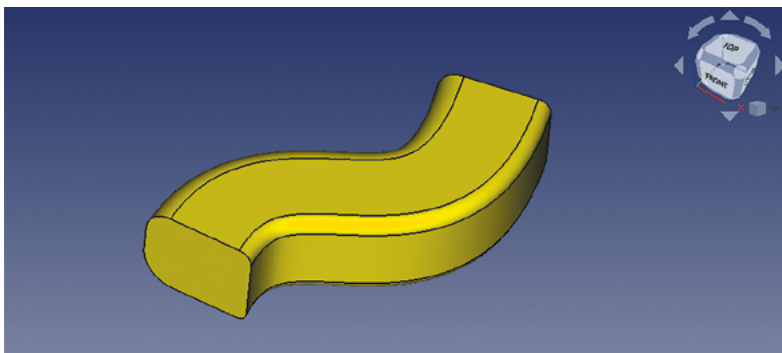
MAKING PIPES

The part design workbench sweep tool creates single solid body objects, but we can apply some techniques we learnt in the last chapter, where we used the 'Create a thickness tool'. As a quick recap, select the face of one end of our 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.
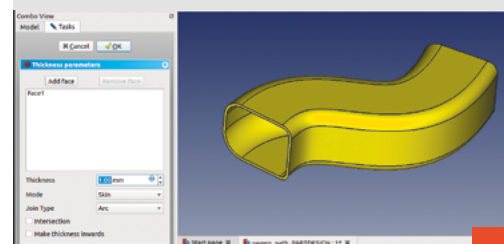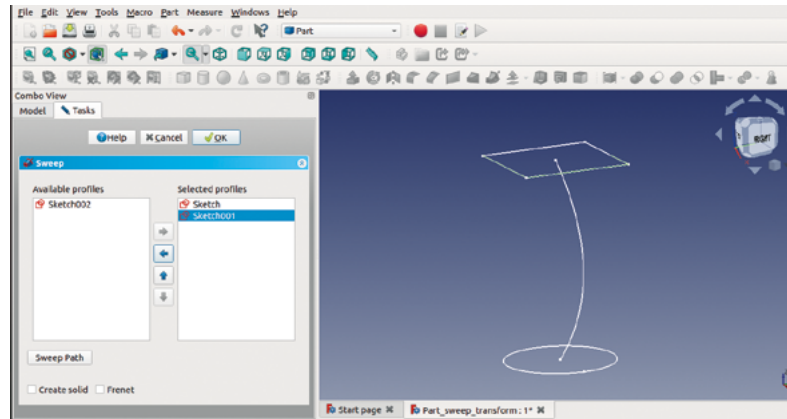
explore on the part workbench shortly. The other thing of note on the part design workbench is that there is a blue and red icon that is the same as the sweep tool we used, except that it's for subtractive sweeps. This means you could draw a solid object and then cut out and remove a swept path through it.

Finally, let's look at the sweep utilities on the part workbench, which are laid out a little differently. We 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 and go directly to the Sketcher workbench. 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 – which should be a similar size to the circle we drew in the first sketch – around the origin point. Close this sketch, but highlight it in the combo view panel so that the dialogue box options for the sketch appear below it. We are going to move the position of the sketch so that it shifts vertically 70mm up the Z-axis. In the sketch dialogue box under the 'Base' heading, open the drop-down menu labelled 'Placement' then, in turn, open the drop-down menu labelled 'Position'. In this menu, change the Z-axis value to 70mm.

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 **Figure 10**.

Moving to the part workbench, click the 'utility to sweep' tool. In the dialogue box, you should see the list
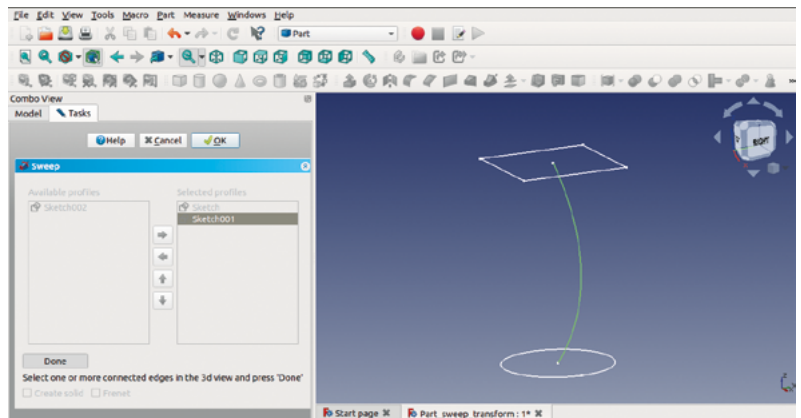
of sketches on the left-hand side. In turn, select the first sketch containing the circle, and click the right-facing arrow that becomes highlighted to move the sketch into the 'selected profiles' column. Repeat this for the second sketch – the sketch that contains the square (**Figure 11**). Next, click 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 we created earlier (**Figure 12**). Next, click 'Done' and then click the 'OK' button. You should now see the circle profile has been swept towards the square on our 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. ◻

QUICK TIP

When you are drawing the third sketch, the circles will appear exactly on top of the first sketch. To make it easier, make the original sketch invisible by highlighting it in the file tree, toggling visibility by pressing the **SPACE** bar.

FreeCAD: technical drawings

Explore the TechDraw workbench and create technical drawings of our FreeCAD projects

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 (**Figure 1**) 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, we need a 3D model to use. Let's quickly draw an item using simple techniques we learnt in

the first parts of this guide, 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 to **Figure 2**. We 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 we 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 on the sketch to create our part. Finally, before we 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 – see **Figure 3**.

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 – **Figure 4**, overleaf.

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 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,
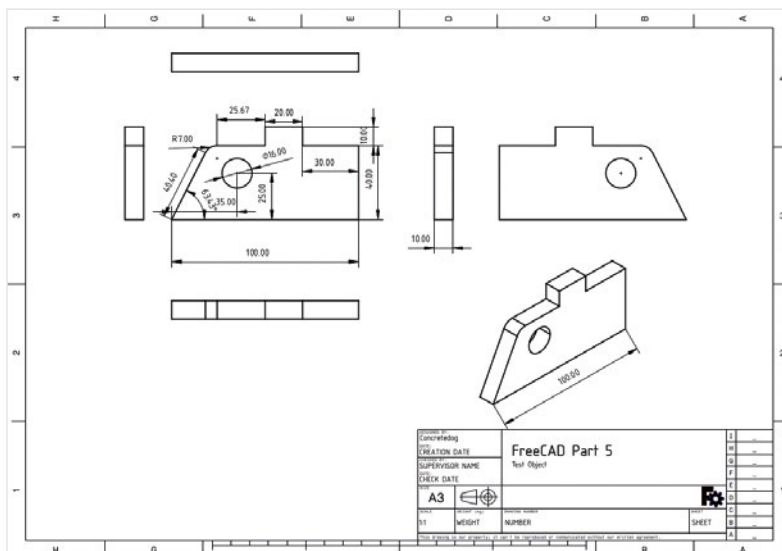
Figure 1 ◈
FreeCAD allows the easy creation of technical drawings of our 3D parts
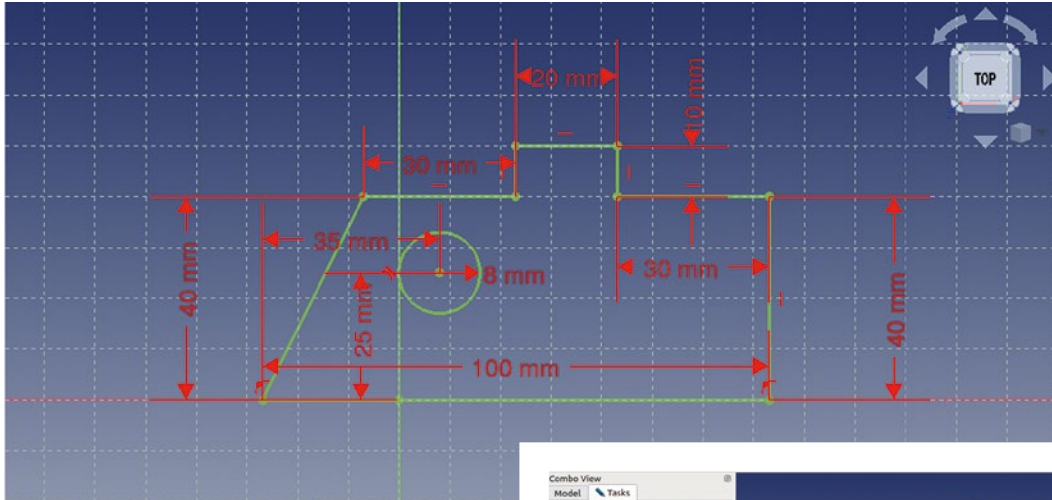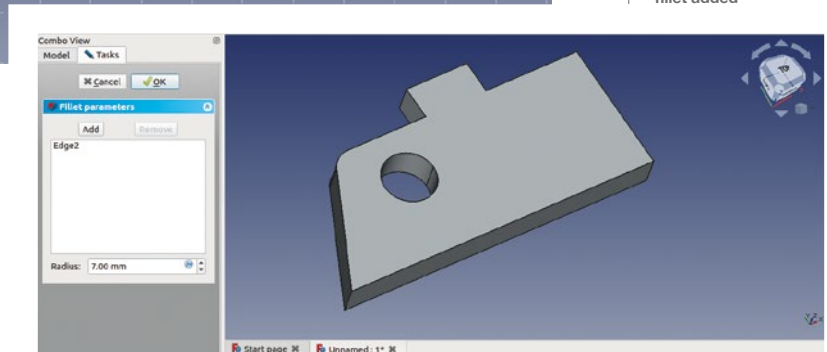
you might not notice, but it can affect dimensions in technical drawings added later. On the 'page' tab, you should have a dashed line around the 'view' – these are referred to as the 'View Frames'. Clicking the frame and dragging allows you to move the view around the technical drawing page. Highlighting 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 View' tool to add the view to the page,

> **Our simple test piece design doesn't need masses of views** to be able to show all of its attributes and dimensions

(**Figure 5**, overleaf). 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'.

Our simple test piece design doesn't need masses of views to be able to show all of its attributes and dimensions, but we would definitely need another view adding to show the thickness of the part. Later, we will look at automatically generating multiple views of a part which adhere to technical drawing →

GETTING SET UP

Before we add any views to our new page, we 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.
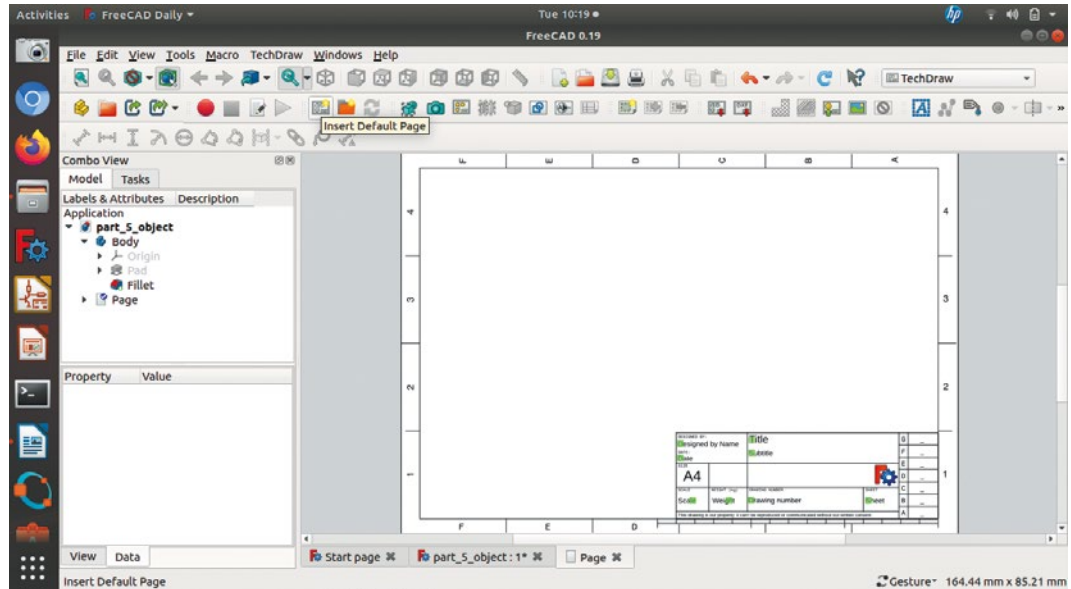
Figure 4 ◆
Starting a first
technical drawing
by inserting a
blank page

ANNOTATIONS

Most technical drawings will have some details added
as text; in our default template, we 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 the
view frame either by clicking it in the preview, or
selecting 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. Note that it doesn't use text wrapping,
and you need to click the **RETURN** key to create a new
line of text. In the image above, we have filled in the
default drawing details with the editable text boxes,
but also added the lines about the part tolerance and
surface finish by adding an annotation box. Notice
we 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.

conventions, but for now, let's just add another view.
Do this manually as we did with our top view. Return
to the 3D model preview, and move the model so we
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 our object in the front view
on the technical drawing page. Then click the yellow

> **You can click and drag this
> dimension to different**
> areas to organise where
> the labels are

'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 we don't have a single complete
line from the bottom to the top of the object, so we
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.
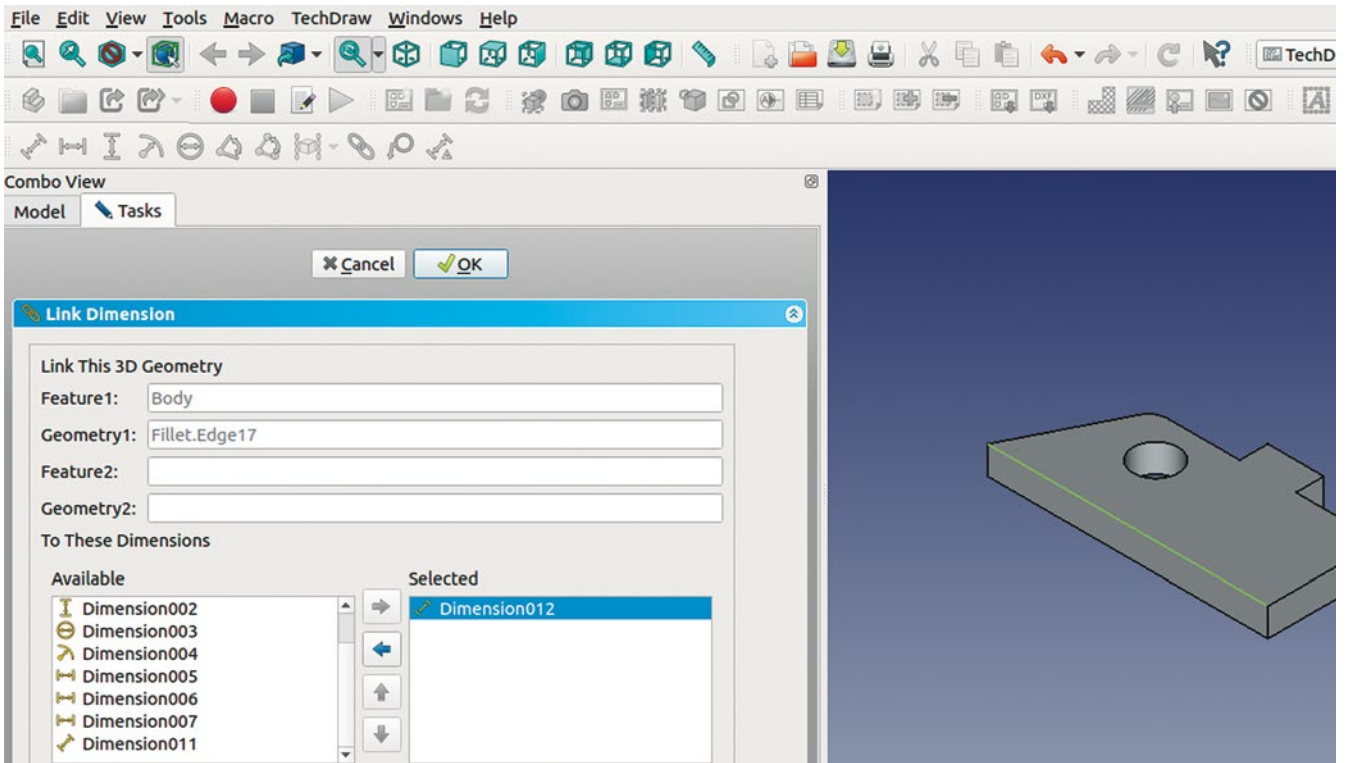
Slightly differently than the Sketcher workbench, when you want to select multiple vertices, you need to hold the **CTRL** key down. Select the right-hand vertex on the bottom edge of our object and the highest vertex on the right of the small bit that sticks up at the top of the design. Click the 'Insert 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 our hole is similar to adding a radius constraint when we drew the circle in Sketcher. We simply select the circle and then click the yellow 'Insert Diameter Dimension' and the diameter of our hole is displayed. To indicate the co-ordinate position of the hole centre, we can select the centre mark and apply horizontal and vertical dimensions relative to another vertex. We 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 our fillet arc. To add this, simply click the arc line and then click the 'Insert Radius Dimension' tool – **Figure 6**. →
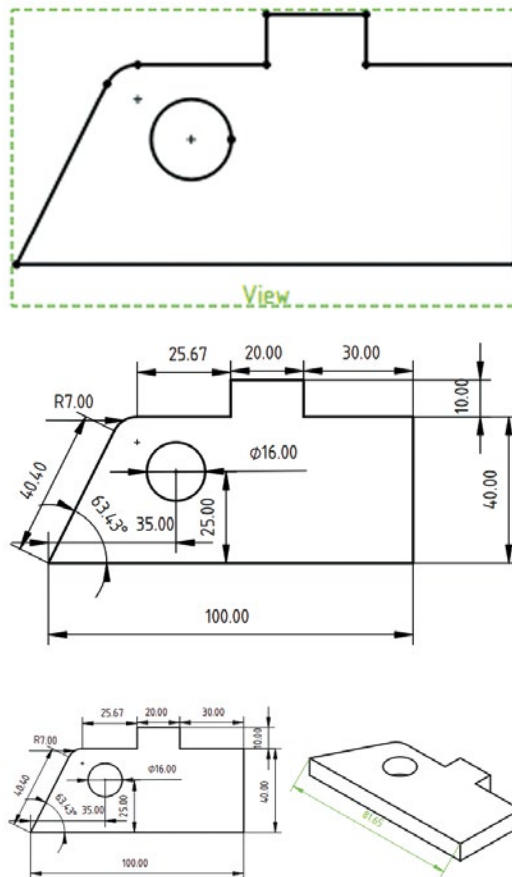
Figure 8 ◈
Linking a drawing dimension back to the model is useful in angled isometric views

Figure 5 ◈
Our first view of our object inserted into the new drawing

Figure 6 ◪
Our view of our object with most dimensions added

Figure 7 ◪
Adding a line length to an isometric view can create an incorrect dimension, as seen here
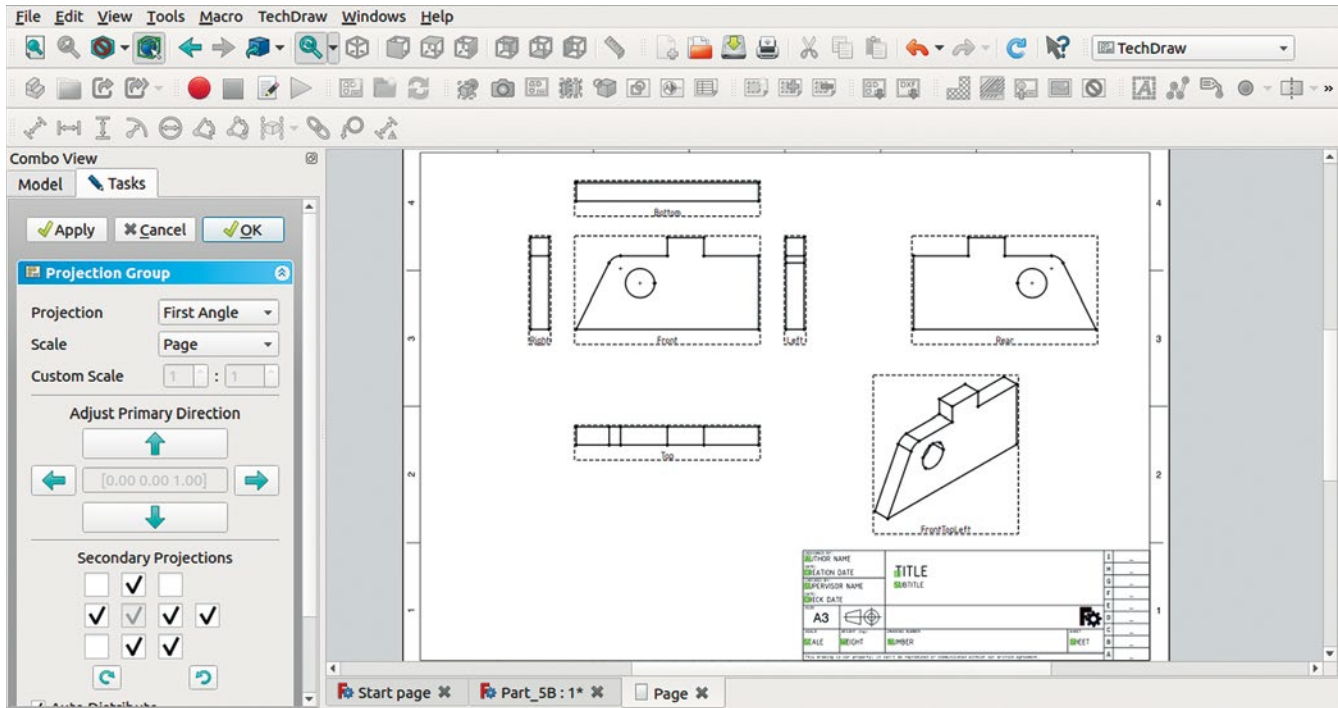
To add a dimension showing the angle of the left-hand side line, relative to the base of the object, select both these lines and click the yellow 'Insert Angle Dimension' tool icon. Whilst it's not often necessary, sometimes we might want to add the actual length of a line rather than a vertical or horizontal 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

> **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 we return to our 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 our 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 our object and click the 'Insert Length Dimension' tool. If you compare the dimension you have just added to the very first horizontal dimension we added on our first view of the object, you'll notice that they are different. This is because the TechDraw dimensions default to being

WHICH ANGLE?

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: **hsmag.cc/Angle**.
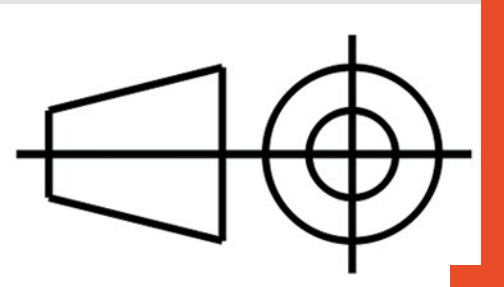
based off the view drawing rather than the actual 3D object, so the actual length of the angled line in the isometric view drawing is shorter (**Figure 7**, overleaf).

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 we just added to the isometric view, and look in the file tree view to identify the name of this dimension. In our 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 we noted earlier and then click the right-facing arrow to bring that dimension name over to the right-hand column – **Figure 8**, overleaf. 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.

Having played with the TechDraw tools a little, let's delete our practice drawing and start again, looking at a couple of things that might make the process quicker now we know how some of the tools work. Select the page we were working on in the file tree and delete it. To create a new page for our 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, we can see any changes we 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 our template indicates the drawing will be in first angle projection. For this example, we 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 our drawing. Select the pattern we have selected in **Figure 9** 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 our isometric view, which we can move anywhere in the page.

As ever with FreeCAD, there's heaps more we 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. ❑

QUICK TIP

To export and print your technical drawing at any time, simply right-click anywhere in the drawing to get a range of output option.

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 our 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 our 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 we 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.
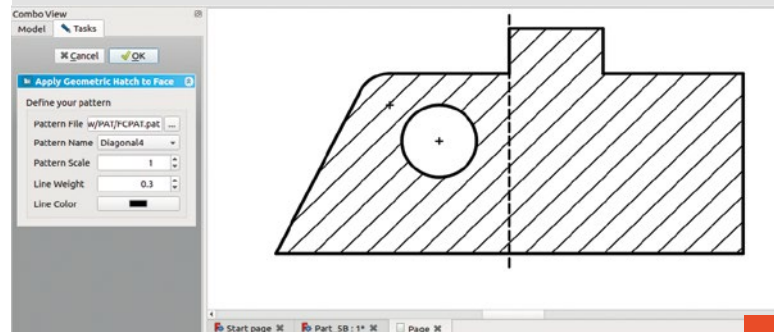
FreeCAD: add-on workbenches and interlocking parts

Use add-on workbenches to make interlocking parts

———

A**fter looking at how we install additional workbenches, we will focus on using an excellent workbench called 'Laser Cut Interlocking' or 'LCInterlocking', which offers some 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 **Figure 2**.

Before we install the workbench we'll be using in this article, 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 we 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 we can export in various file formats (DXF, SVG, etc.). While it's predominantly, as the name suggests, aimed at laser cutting, it also can
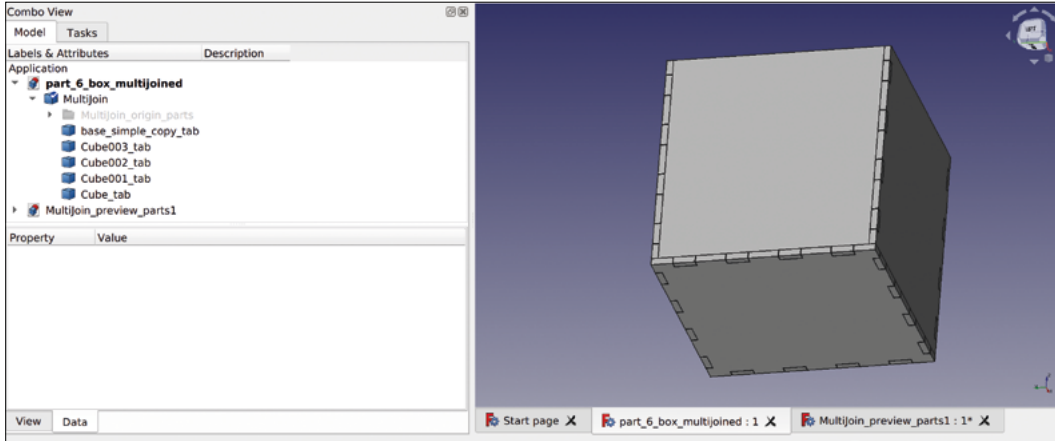
be useful for other processes, such as preparing files and objects to set up toolpaths for CNC routing. The 'interlocking' part of the LCInterlocking name hints at another handy feature: the workbench has tools that can automatically create tabbed joints in objects where one part has tabs and the other 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.

> "From aeroplane design tools to computational fluid dynamics **to 3D print slicing engines – there's a lot to explore**"

Having drawn the box sketch, use the 'extrude/pad' tool on the Part workbench to extrude the sketch to a thickness of 3 mm. Using 3 mm means we can imagine we are going to laser-cut our tabbed box out of 3 mm plastic or plywood. We could make this base part in a much easier way – namely, we could use the 'Create a cube solid' tool. However, as an example, we wanted one part to be made from an extrusion as we need to treat extruded parts a little differently when we get to the LCInterlocking workbench. We can use the cube generator to create the sides of our 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 we made earlier (**Figure 3**, overleaf).

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, as in **Figure 4**.

Before we move over to the LCInterlocking workbench, we 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. We 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' (**Figure 4**).

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 our open box in the file tree, and then click the grey 'Interlocking' tool icon. In the dialog box that

> Dog bone holes are where the machine is instructed to **cut a little further into the corners on both axes**

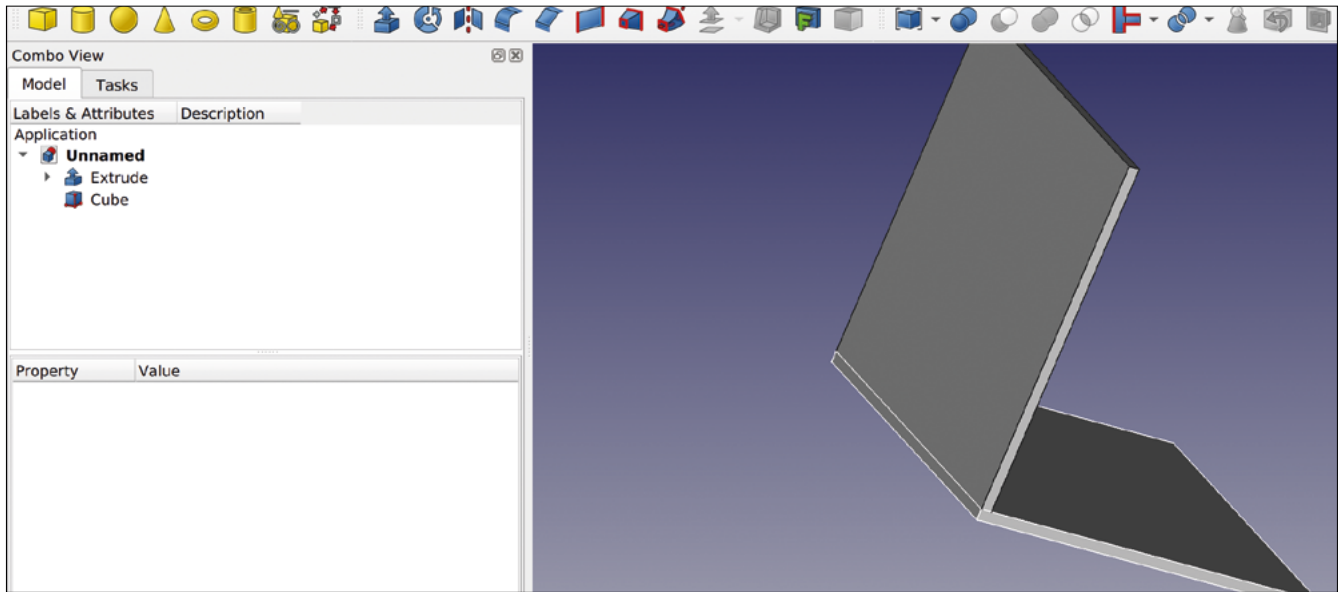appears, you will see a button that says 'Add parts'. Click this button and you will see the five selected parts appear in the window (**Figure 5**). This list of parts in the dialog box is similar to the file tree in the combo view in that you can highlight anything in the list – you can press the **SPACE** bar to toggle visibility. This is very handy for the next task. We need to tell the 'Interlocking' tool which faces of our 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 (**Figure 6**, overleaf). With that face highlighted, click the 'Add faces' button in the dialog. You should see

Figure 3
Adding sides to our extruded base using the 'Create a cube solid' tool

Figure 4
Our simple box base and sides, created and aligned on the Part workbench

ROCKET WORKBENCH

One add-on workbench we wanted to give a special mention to was the Rocket workbench. The Rocket workbench is in active development, and contains tools that help you design rocket parts parametrically. It has an exciting road map ahead that includes features like importing files from other rocket design environments such as OpenRocket, RockSim, and RASAero. This will open up great possibilities for rocket design, flight simulation, the realisation of parts via 3D printing, CNC milling, and more. We featured rocketry, including a tutorial on OpenRocket, back in issue 12 if you're interested in getting started with model rockets and rocket design.
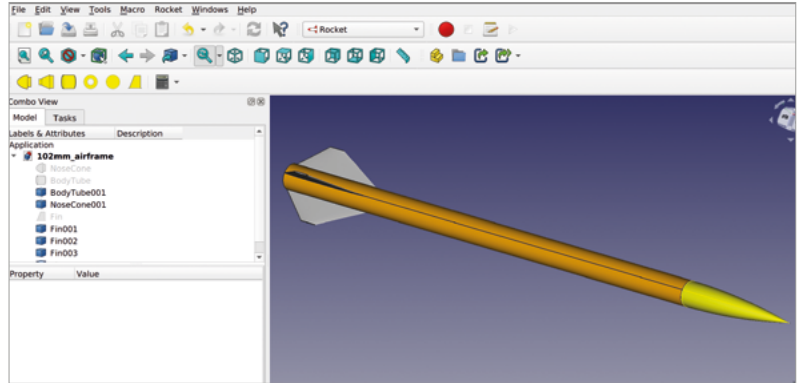
the added face highlighted in the list, filed under the part that it is attached to. If you scroll down in the dialog box, you will see a menu relating to that face. In this, you can input the details of the tabs that you wish to generate on this face.

Add the number of tabs – you can make your own choices, but we went with four. We 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' (**Figure 7**, overleaf). 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.

We deselected both of the checkboxes as we don't need them for this example.

Having input the tab parameters, scroll 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 our box appears (regardless of which parts are visible or invisible in our other tab). If we look at the joint →
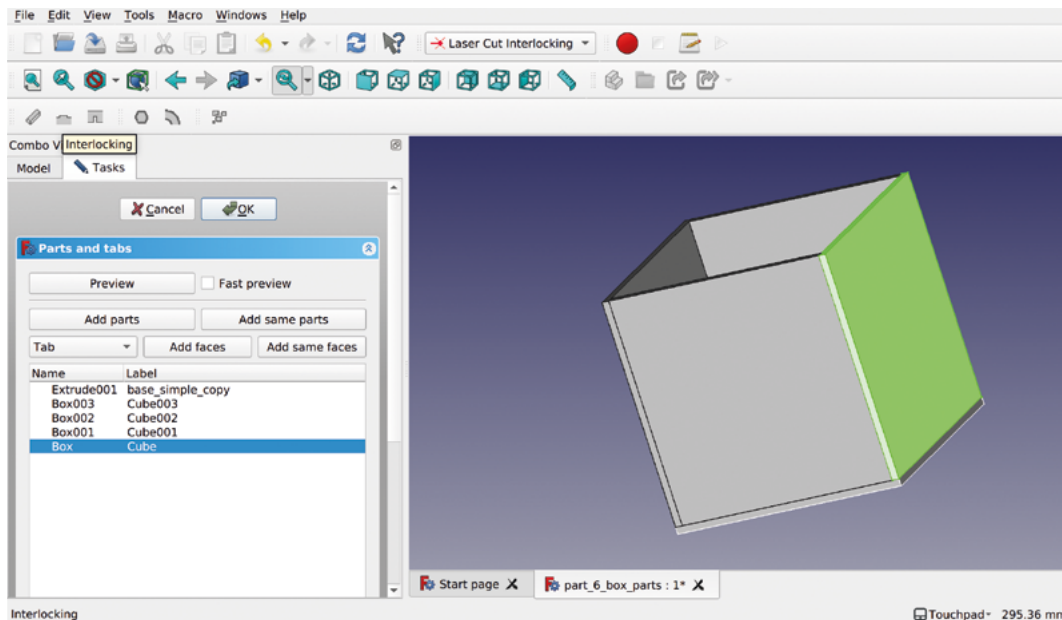

Figure 5
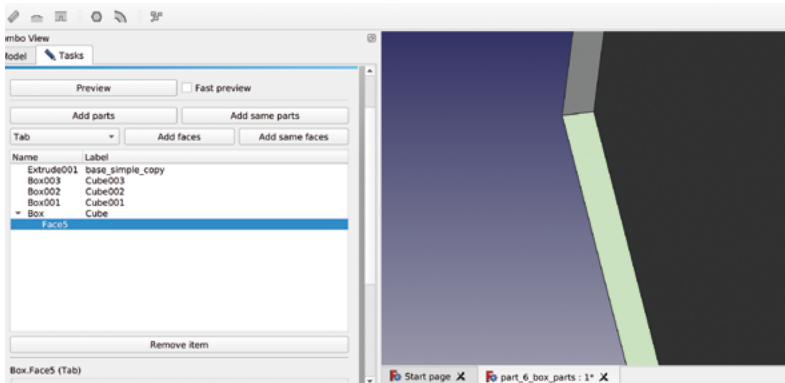Adding the parts of our box to create tabs and slots using the 'interlocking' tool

39

between the base and our tabbed face, there are now four tabs interlocking into the base (**Figure 8**, overleaf). We can now repeat the above process of selecting and adding faces to the interlocking dialog and configuring tabs. Working our way through the model, we 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 our box.

Once we are satisfied with our added tabs in the preview, click the 'OK' button to apply the interlocking tool to the parts. In the Combo View under our main document, there is now an object called 'MultiJoin' (**Figure 1**). Within this file tree, there are now parts that are the tabbed version of our 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, we can close the preview tab without saving it for this example.

The next task is to apply the 'Export' tool to the tabbed parts we have generated. Rather than actually export a file, this tool explodes our 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 we created under the MultiJoin object, and then click the 'Export' tool icon (**Figure 9**).
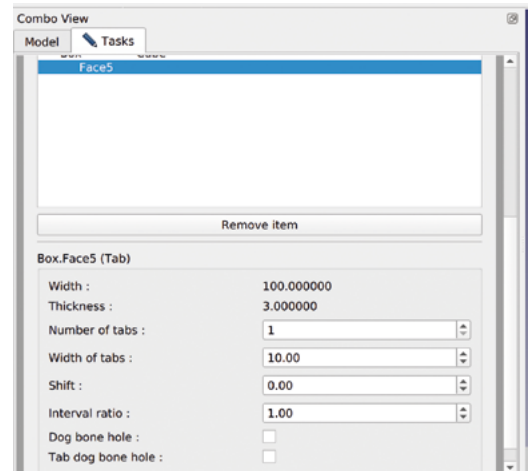
Once again, FreeCAD will open a new preview window tab – this will contain all the parts of your box separated and laid out on a single plane. You will also see some new objects in the file tree if you rotate the plane in the preview, as we have in **Figure 10**. These are the 'Shape2DView' objects – 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.

If we wanted to cut the parts for our tabbed box on a CNC router, we could easily use the Path workbench to create toolpaths using the 3D objects that the LCInterlocking workbench export tool has laid out. Using the Shape2DView parts, however, we 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 we just have the Shape2DView objects visible in the preview. 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 we created. The beauty of an SVG export is that we don't lose any dimensional accuracy, and we can open this file for editing in many graphics packages. We are huge fans of the free, open-source Inkscape package here at HackSpace magazine, so let's take a quick look at our SVG using that.

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 we will want to move parts around on an Inkscape canvas to arrange our components so that we are as economical as possible when cutting the parts on our laser cutter. That might be the only edit you
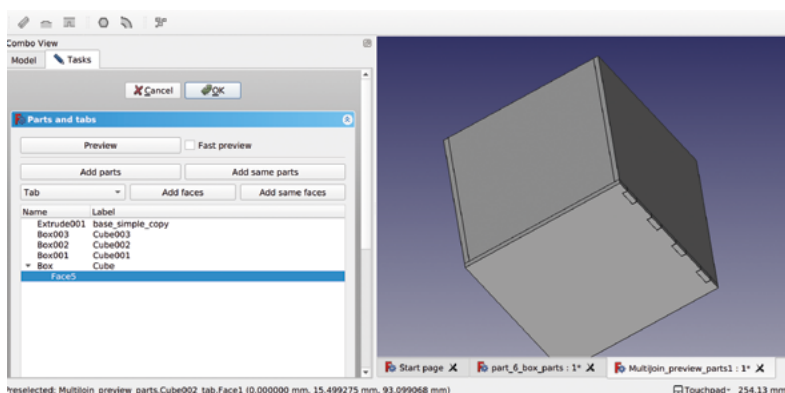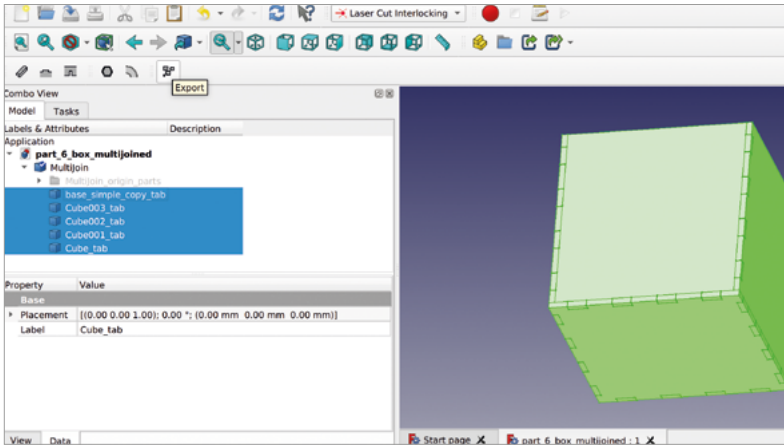
need to make, but often you may need to customise the SVG file to work with the settings of your laser cutter. We can think of a couple of different laser cutters that we have access to that have slightly different requirements. One needs any line that the laser is going to perform a cutting/vector action 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

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 we need to do our 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. We find that we barely need to post on the forum when we search first. We 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.

the laser cutter will engrave/raster; again, this is simple to realise in Inkscape, as we have done in **Figure 11**. Having made our changes in Inkscape, we 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! □

FreeCAD: get started with assemblies

Make complex designs out of a collection of parts

W **'ve already seen how to use add-on workbenches; in this chapter, we 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. We 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 we are focusing on the assembly, we aren't going to step through how we made every separate component, but we have posted the parts used in this project online, so you can just have a go at assembling the pulley block using those parts. 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

Figure 1
We are aiming to assemble this example of a pulley block made up of 14 parts
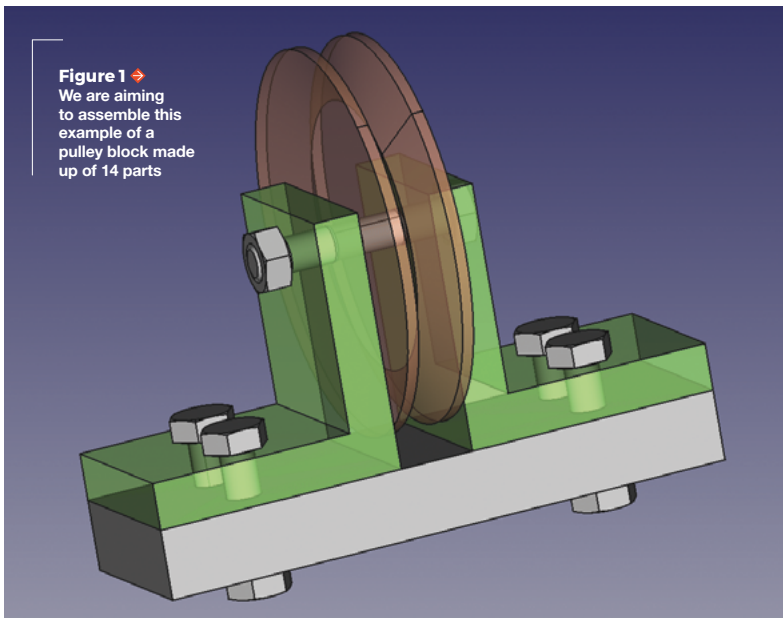
> This isn't supposed to be a particularly well-designed pulley block **... but as an example, it's a good project to play with**

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, we 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
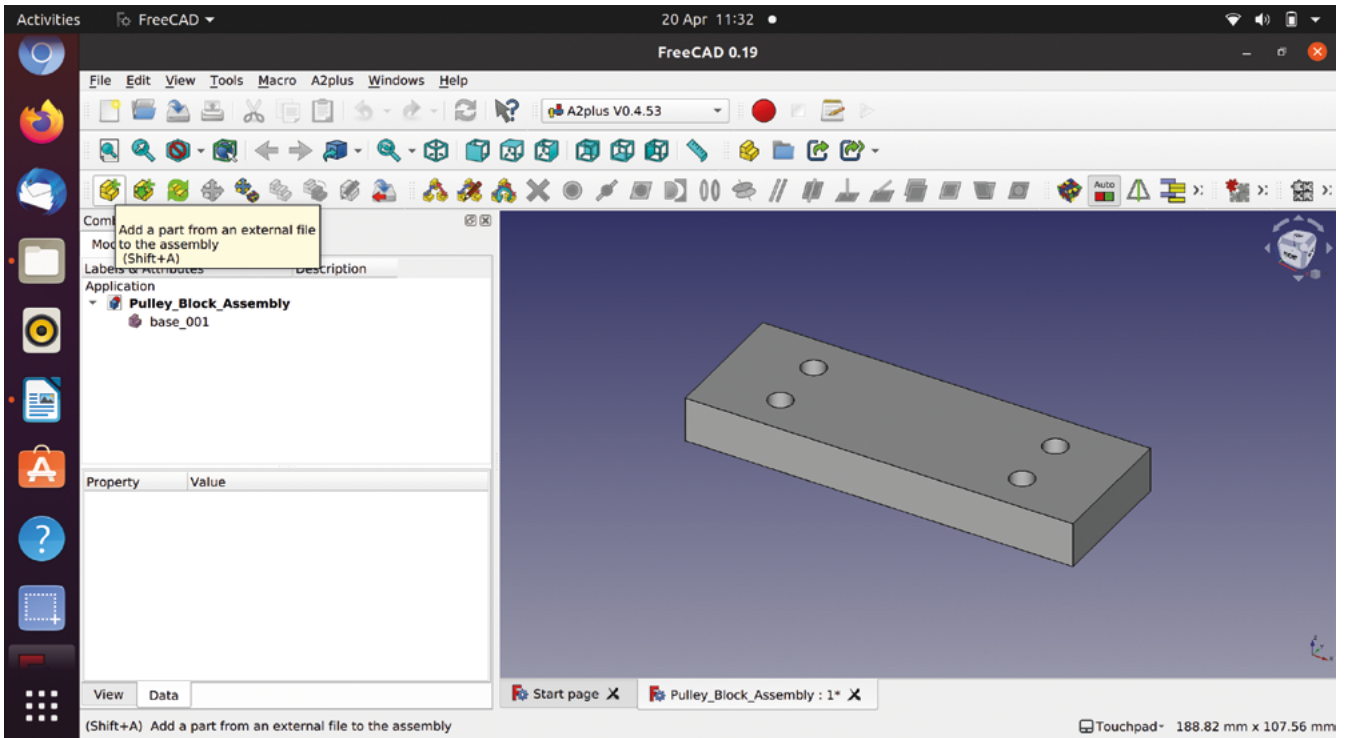
Importing our first part of the pulley block

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.

We began by making our components using Part Design to create bodies for each part. We created a base, two L-brackets, some nuts and bolts, a simple V-pulley, and a longer nut and bolt to act as a shaft. You'll see that we are going to import these items into the A2plus workbench, and when we do, we could import each body either from an individual project, a project with just that one body in it, or we can import multiple bodies from the same project. Most of our files are single items, but we 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. We can imagine borrowing items from unrelated projects in our library.

We 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 we 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.
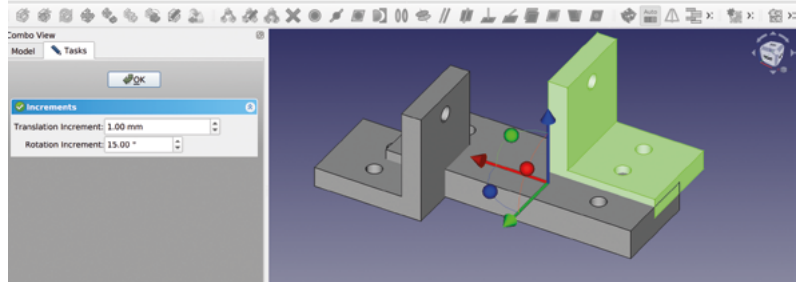
The L-brackets are again a simple part with holes that match the hole coordinates on the base and a hole on the upright to receive the spindle bolt. You can see the complete assembly in **Figure 1**.

Download the files (from **hsmag.cc/issue43**) and create a new project in FreeCAD. For ease, save the 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' (**Figure 2**). 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 (**Figure 2**). 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, →
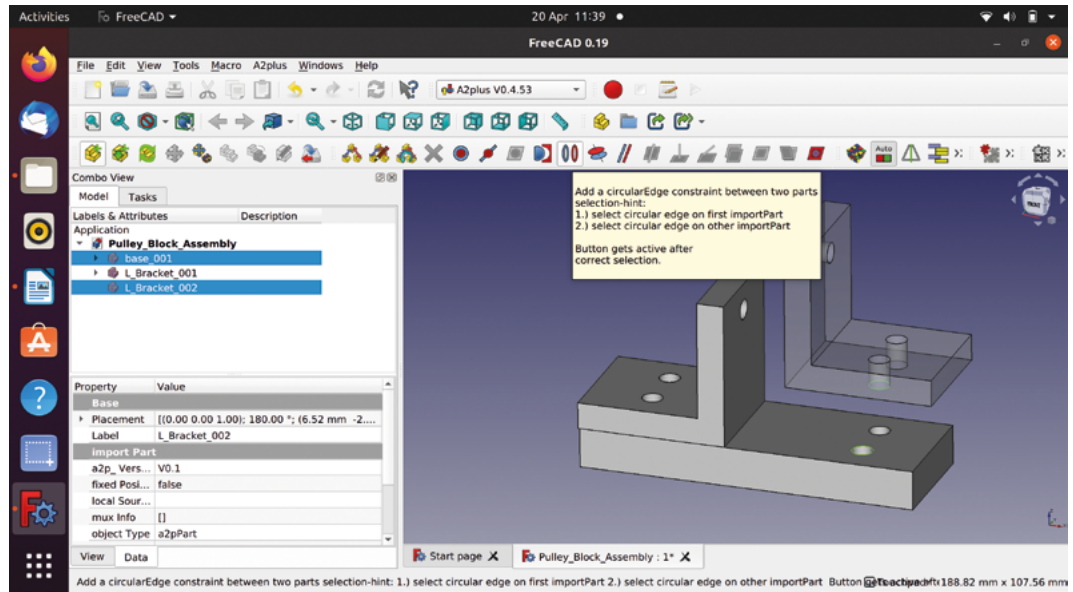
double-click on the item in the file tree menu and you can transform it incrementally using the dialog and the axis arrows. Move it to a position free of the base to make things easier to see. Repeat the process and import a second instance of the L-bracket file. The files in the file tree will automatically be given a number suffix, so we 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 (**Figure 3**).

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,

Figure 4 ⬚
Setting some parts as transparent enables us to show which hole circles we selected to constrain the L-bracket to the base

Figure 5 ◆
When applying constraints, you will see a dialog box with options
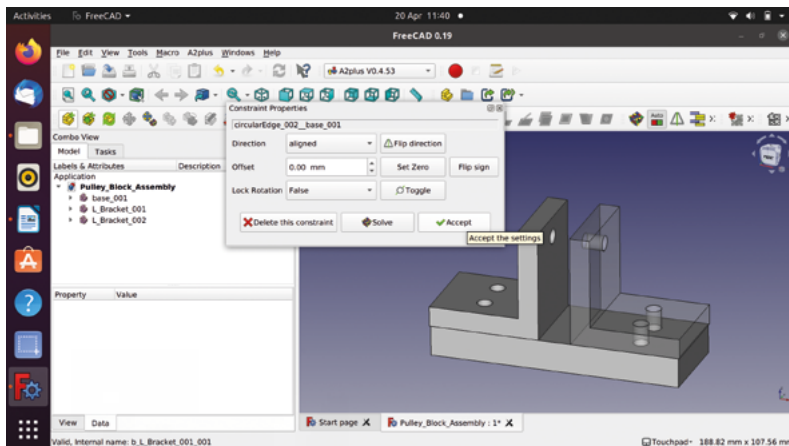
KEEPING BACKUPS

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 we can easily demonstrate in our project. Objects and parts – for example, the base of our 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 we apply constraints in the A2plus workbench, we are essentially telling FreeCAD to 'attach face X to face Y' or 'make edge 2 parallel to edge 5', etc. When we edited our base part to make the brackets fit, we 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 we 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.
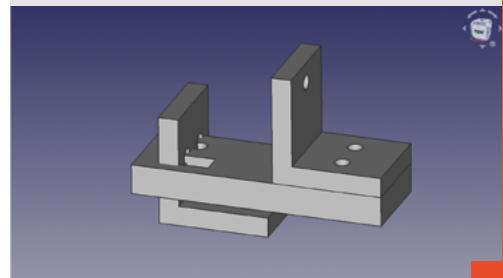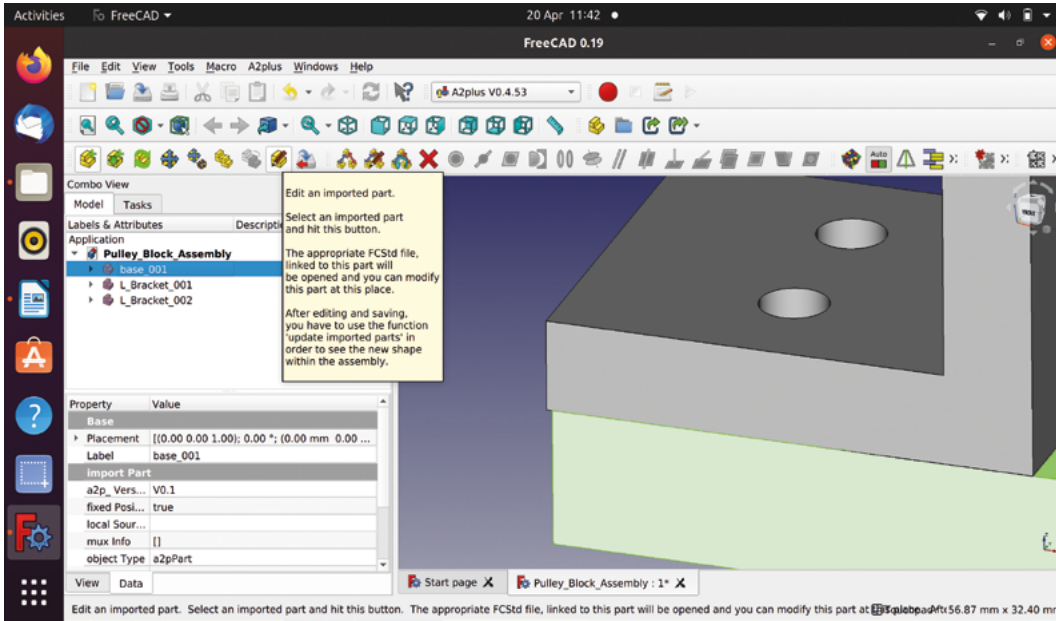
any constraint types that could be applied to the items automatically become available on the toolbar. You should now be able to click the 'Add circular edge constraint' tool icon (**Figure 4**). You will see that the L-bracket moves, hopefully into a correct position. You will also see a dialog box with some options showing the constraints you can adjust. We 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 (**Figure 5**).

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 our partly assembled project hasn't changed. To update the parts, click the 'Update parts, which have been imported
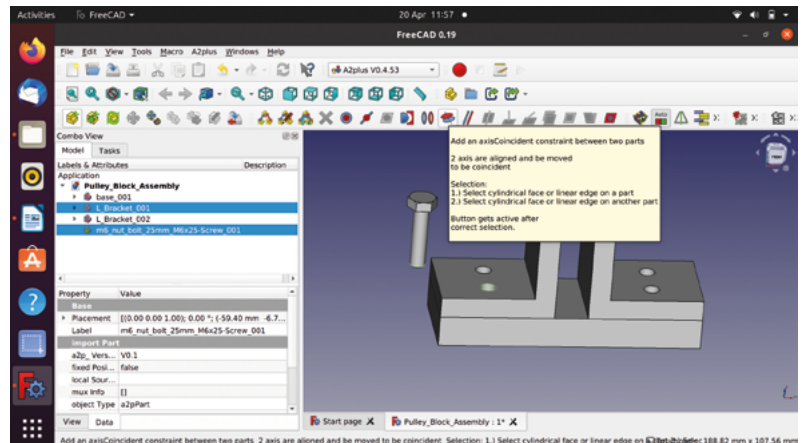
to the assembly' tool icon (a yellow part icon with two curved green arrows over it). Once clicked, you should see the base now fits, perfectly matching the L-bracket ends. A point to consider again is that the holes in the base file are constrained to the centre line of the base. This means as the base dimensions are expanded, the holes stay the same distance apart. If we had constrained the holes relative to the edges of the base, then our 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 either the project with the M6 nuts and bolts you just made or, if you didn't, the file we supplied in the download, 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 we did for the base and →

QUICK TIP

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

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 our 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. We want
these two circles to align on an axis, but we 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 axisCoincident constraint between
two parts' achieves this – the bolt should now align to
the hole (**Figure 7**, overleaf). 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.

> It's a great workbench to
> play with and experiment
> **with how the different**
> **constraints work**
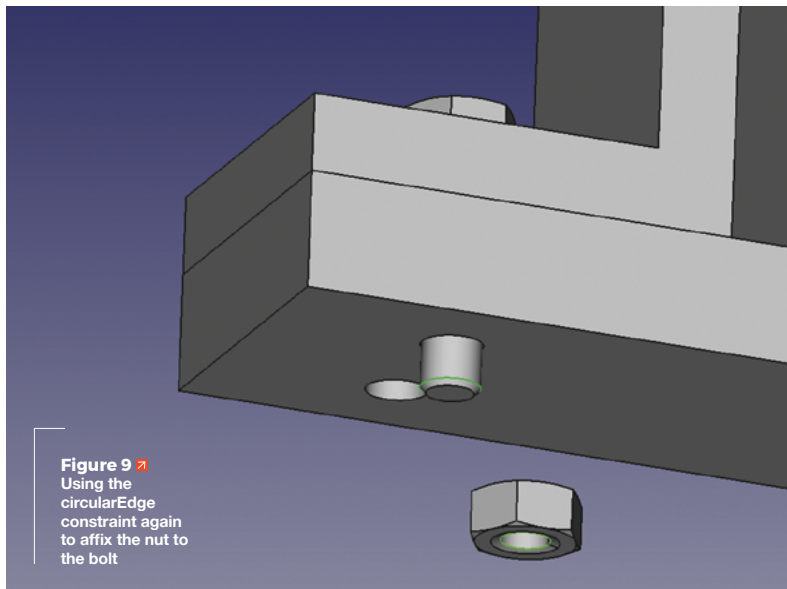
This time, click to select the upper face of the bracket
and then select the lower face of the bolt-head. These
two faces can then be brought together by clicking
the 'Add a planeCoincident constraint between two
objects' tool (**Figure 8**). Now the bolt should be
correctly in position. Click Accept in the dialog box to
finish the constraint. Moving to the nut, if we move the
nut closer to the bolt end under the base, we can then
select the edge at the base of the bolt before the slight
chamfer and the lower similar edge just inside the nut
(**Figure 9**). Again, apply a 'circularEdge' constraint to fit
the nut to the bolt.

Now that our nut and bolt are constrained, we
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,

Adding the pulley, and positioning it with an offset within the constraint, to create the clearance gap with the L-brackets

but now that we have added constraints to our first nut and bolt, if we copy and paste, that will copy all the constraints applied to that part. Of course, we 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 (**Figure 10**).

PULLING TOGETHER

Let's add the pulley wheel next. 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 (**Figure 11**). Currently, the pulley is positioned touching the bracket of which we 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 our 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 our 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. We are sure by this point that you can work out how to constrain this one into position without instruction. We hope that you've found this introduction to assemblies useful, and after you have put this together, it's a great workbench to play with and experiment with how the different constraints work. The simple assembly we've created is a good starting point and a foundation for making assemblies that move and can be animated, and much more. ▫
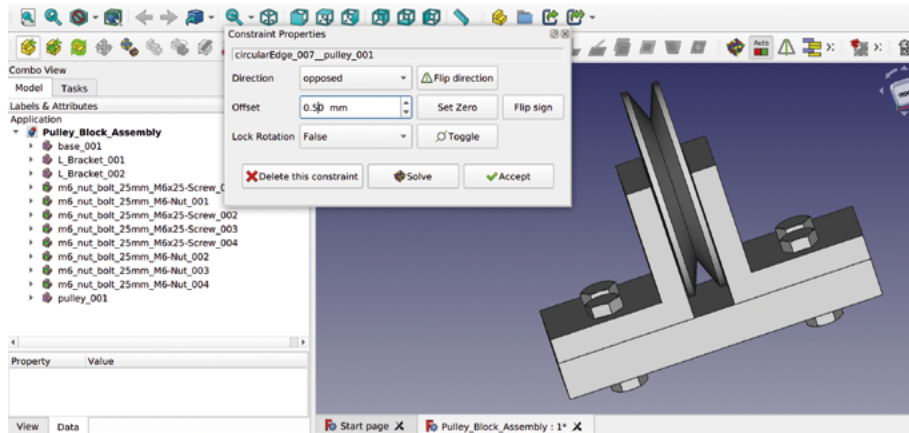
JOIN TOGETHER

We are going to make some nuts and bolts for our assembly. For this, we will use the Fasteners workbench we 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. We are going to use four M6 nuts and bolts that are 25 mm long to attach the L-brackets to the baseplate; however, we only need to model one and then import four copies to the A2plus assembly project. To create our 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, we 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 our 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 our 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 our assembly, or use the one we have provided.
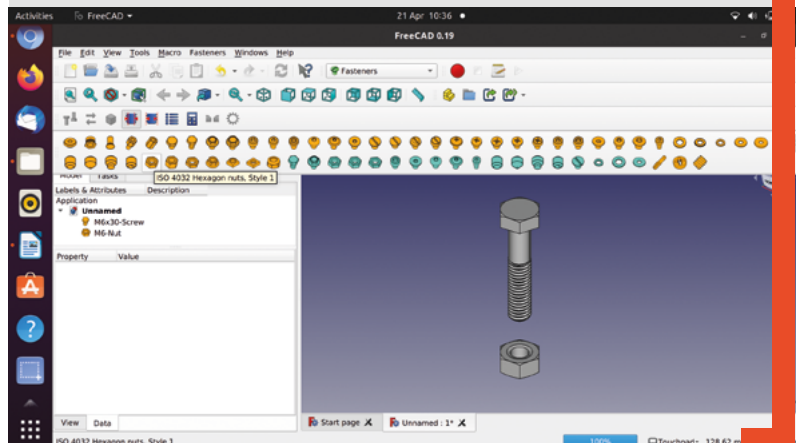
FreeCAD: making multiples of things

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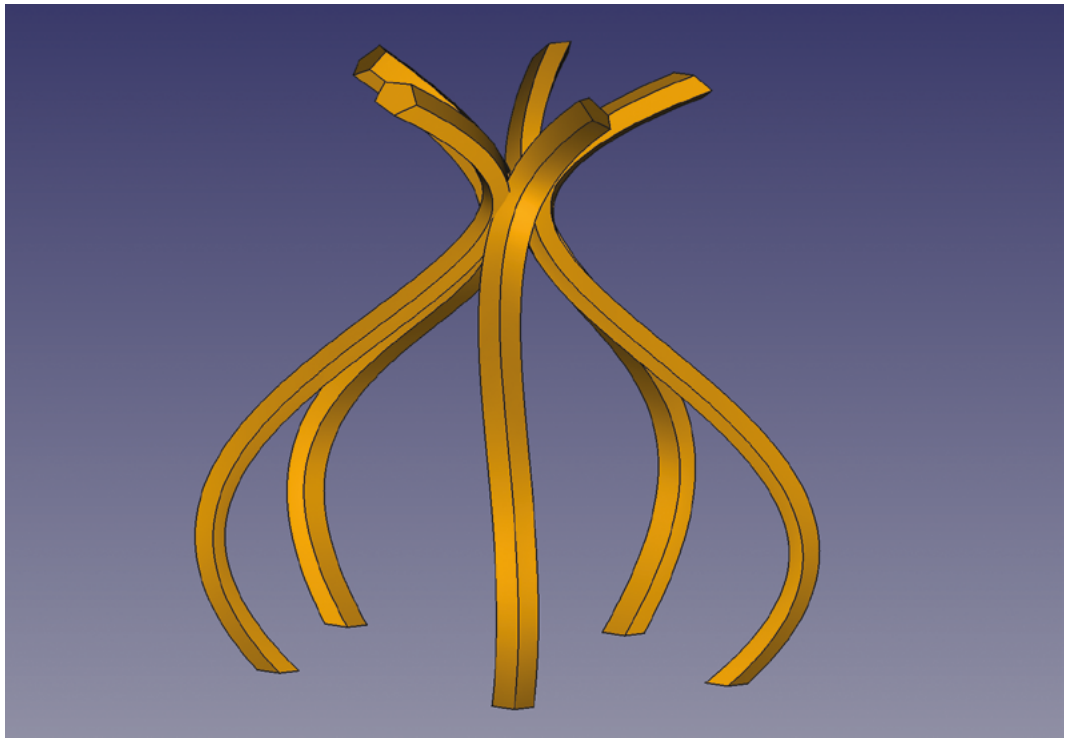
Often when we are designing something, we would like to make multiples of some aspect of the design and we want to do it as quickly and as efficiently as possible. A common example might be placing circles to become pockets, holes, or pads in a sketch. As a first example, let's go straight to the Sketcher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

In **Figure 2**, you can see the desired sketch item on the left, a small square with a hole in each corner. To create this sketch, we 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 we draw one circle for example in the upper left-hand corner of the square, we 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. We can now click the 'Create symmetric geometry with respect to the last selected line or point' tool icon. This should now create a second
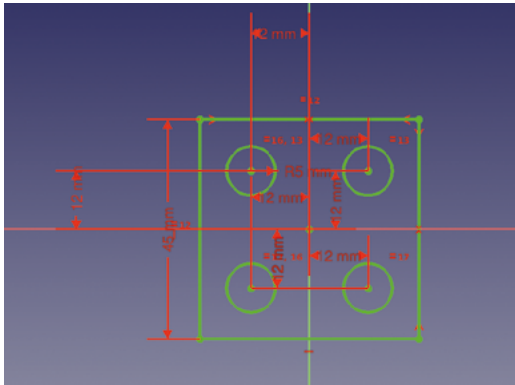
circle, with the same radius constrained and in a
mirrored position in the upper right-hand corner of
our square. It doesn't have the positional constraints,
but it is in the same mirrored position as the original,
so if you add a vertical and horizontal constraint
without changing the default position 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 our upper left constrained circle highlighted,
click this tool and you'll see a dialog box (**Figure 3**). 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 our original circle is 12 mm from the datum
point vertically and horizontally, we set the constraint
distance to 24 mm, which makes our square array of
circles fully constrained and accurately placed.

For our 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
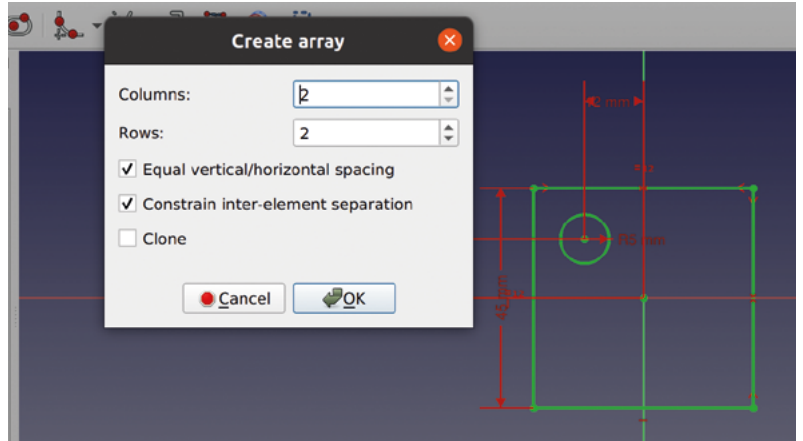
the rectangular surface. Close the sketch and then
perform a pocket to cut the slot into or through the
pad. Just to add to the example, select an edge of
the slot and apply a fillet, which should auto-complete
around the entire pocket edge.

DESIGN PARTS WITH PART DESIGN

The Part Design Workbench has some specific array
tools, and we 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 →
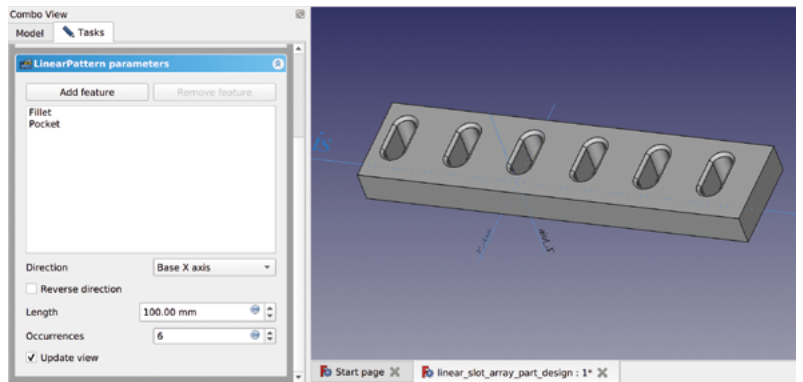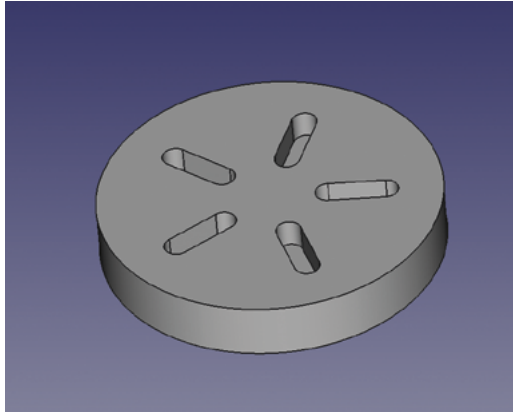
length of the array and isn't linked to the length of any attached object. At the default length of 100 mm with two occurrences, this might place the second occurrence of the pocket outside of the rectangular pad. Increase the number of occurrences to 5 and adjust the length of the array so that your 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 we close this example, you'll note that our 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 (**Figure 4**). 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.

Sometimes we 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 our earlier example, create a sketch on the top surface of our 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 our cylinder face and perform the pocket.

ROUND AND ROUND

Again, with nothing highlighted, click the 'Create a polar pattern feature' tool icon. The resulting dialog box is similar to our earlier linear pattern tool. Click 'Add feature' and select the pocket that we 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 our original slot was off axis, the array positioned the repeated slots relative to this original position; we can of course go all the way back to the root sketch of the pocket and adjustments will be recomputed to the pattern array. As shown in **Figure 5**, we closed the polar pattern dialog and opened the slot sketch in the file tree. In the slot sketch, we constrained the slot so that the centre of the slot was positioned on the Y axis datum line.
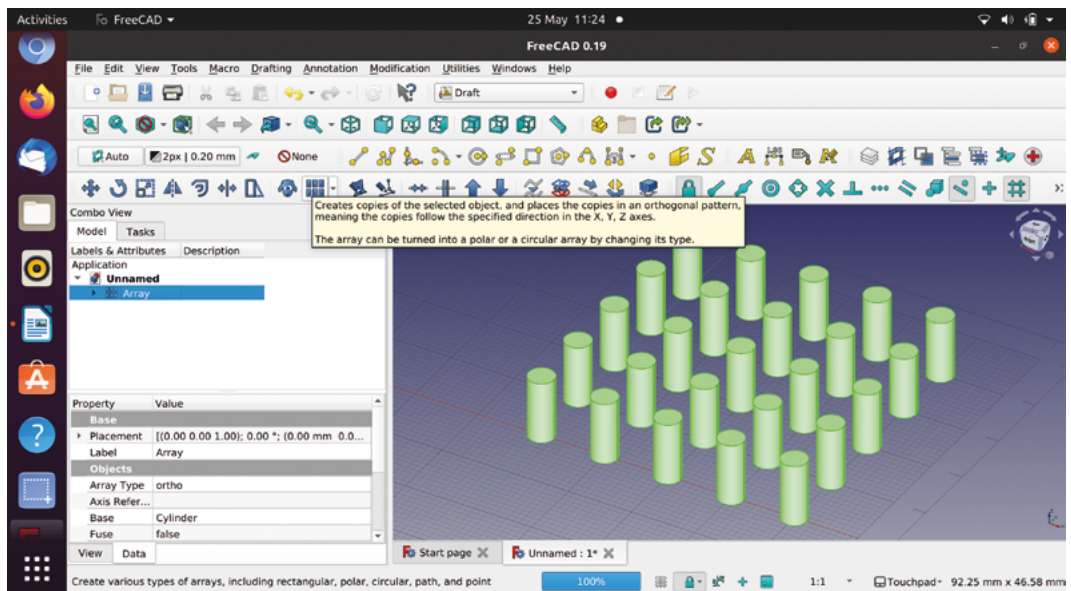
Many times we will want to create more complex arrays, sometimes with more complex geometries or sometimes not in a single continuous body, which means we would be working outside of the Part Design Workbench. The built-in Draft Workbench has some array tools that can be extremely useful for this, but it's a useful Workbench for much more as well.

ON REPEAT

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, we 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

> **Many times we will want to create more complex arrays, sometimes with more complex geometries**

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 we only have one layer in our array. Clicking OK, you should now see an array of our cylinders spaced at the intervals we set (**Figure 6**).

Highlight the array item in the file tree and delete it. We should now see our cylinder item and if we 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. We need to add a point which is the centre of rotation; if we left the centre of rotation set at the 0,0,0 point, we 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 our test cylinder and click the 'Circular array' tool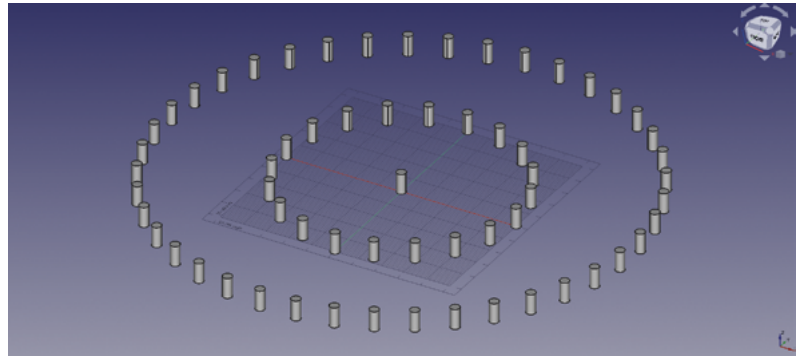 icon. This tool works differently in that the number of copied objects in the array is dependent on the size and spacing of the array rather than a directly input value.

In the dialog box, the 'Radial distance' is the radius of each circle in the created array; we set this to 50 mm for a test. If we don't change the centre of rotation and leave it at the datum or zero point, then the first circle in our array will be made with a →

FOLLOW THE PATH

Sometimes we 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 we wish to create an array; let's select our second, smaller cylinder. Next, we select the path we 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 we 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.
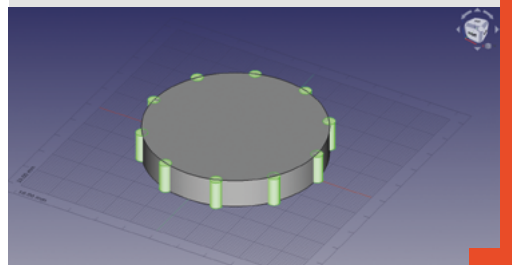
QUICK TIP

Near the Part Design pattern array tools, you might notice the 'Create a mirrored feature' tool, which is similar to the symmetric geometry tools on the Sketcher Workbench.
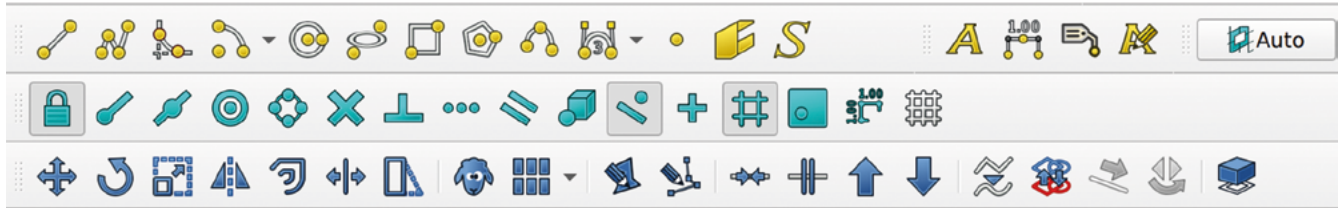
50 mm radius from this point. The second circular layer will be made at a further 50 mm from the first circle so therefore will have a 100 mm radius, etc. Next, the 'Tangential difference' is the spaced distance between instances of the array object in the same circular layer of the array. We initially 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 we said at the radial distance; we set this to 3 for our 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 **Figure 7**.

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.

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 co-ordinates. As such, the Draft Workbench has a lot of options for different snapping styles. For our example, we need to enable snapping by clicking the 'Snap Lock On/Off' padlock icon. This should now activate the other snap icons; we want 'Snap grid' and 'Snap near' to both be on, as seen in **Figure 8**.

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 and 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. In the properties dialog, you can adjust the number of sides, but as a default it is set at three. If you create the default triangle, you can
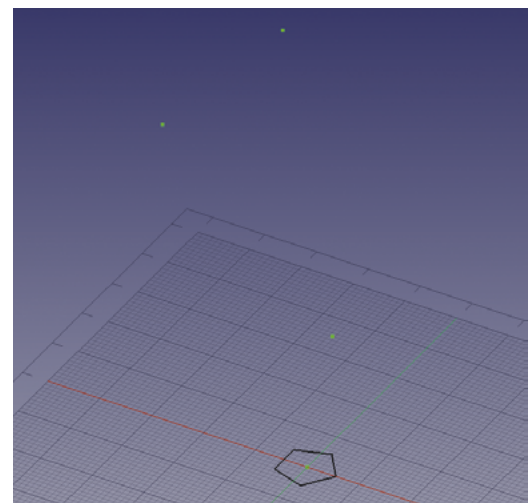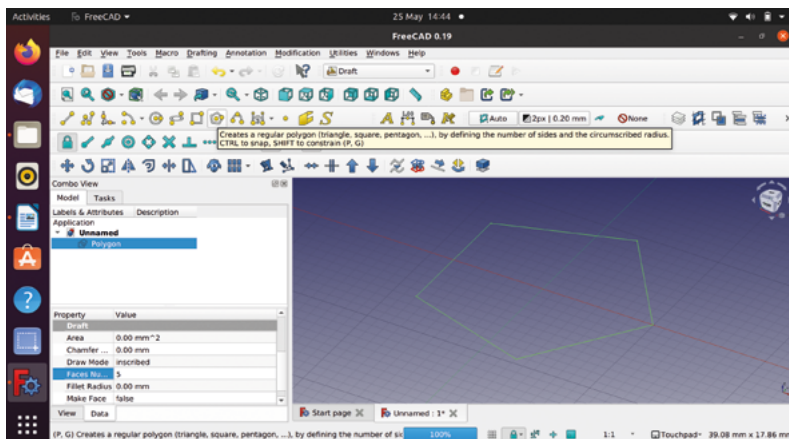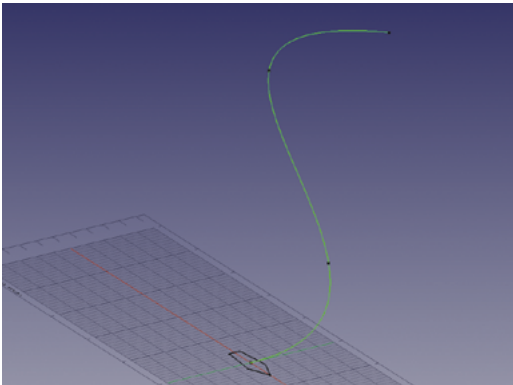
Figure 11 ◈
Using the B-spline tool to draw a curved line between the points we created

highlight the Polygon object in the file tree and adjust the number of sides in its properties table.

CURVED LINE

Next, let's create some points which we will use to create a curved line in three axes. The first point of the line is going to be the centre of our 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. We 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. We went with an X co-ordinate of 6 and a Y co-ordinate of -2. Having created this second point, we 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. We 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 (**Figure 10**).

Creating a curve between the points we just made is easy with the 'Create a multiple-point B-spline' tool. Select this tool and hover near the upmost point we 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 (**Figure 11**).

Just for a moment, we 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
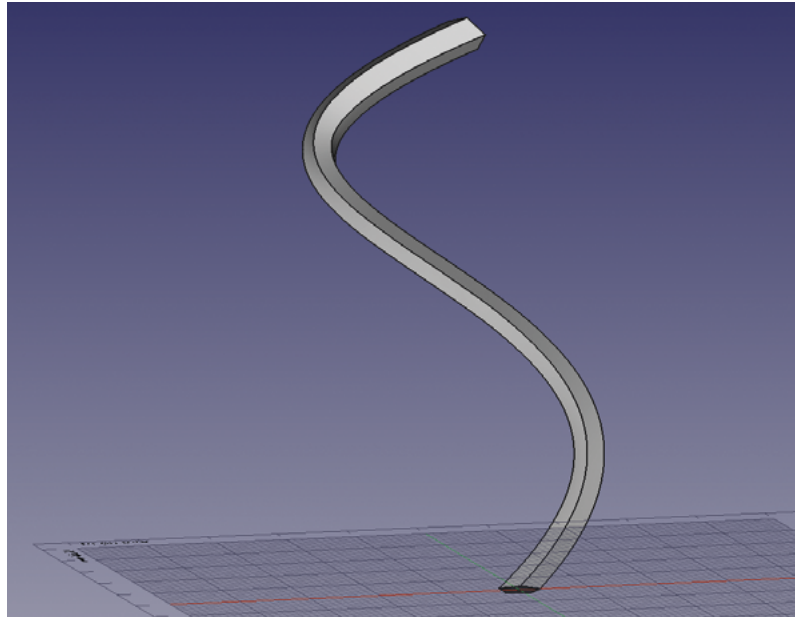
and in the dialog, select the Polygon item we drew from the left-hand list, and click the right-facing arrow to add it to the right-hand column. Next, click the 'Sweep Path' button, then left-click on the B-spline curve we drew in the preview window and click 'Done'. Check the 'Create solid' box and click OK. You should now have a curvy swept hexagonal object in the preview (**Figure 12**).

Moving back to the Draft Workbench, select the swept object in the file tree and then click the polar array tool icon. You can then left-click in the preview window to select a centre of rotation (**Figure 13**). We put our centre of rotation at around -30 mm on the X axis to create our array of five of our swept items; you can see the completed array in **Figure 1**. ▢

Figure 12 ◈
Using the sweep utility on the Part Workbench to create the final part for our array

Figure 13 ◈
Selecting a centre of rotation for the array of swept items; we chose -30 in the X axis
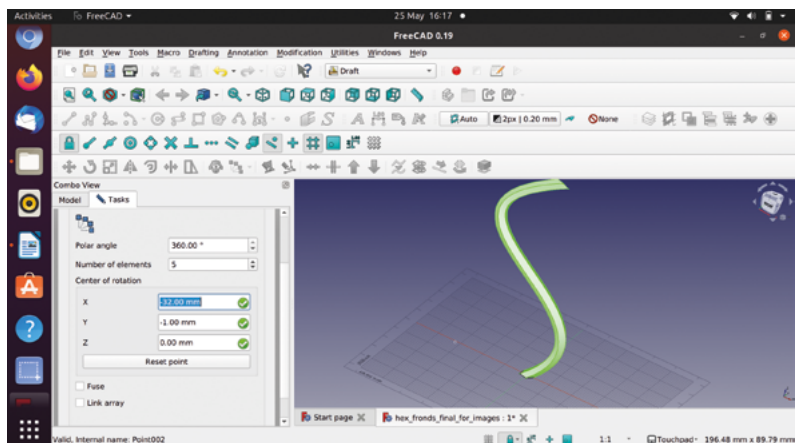
FreeCAD, surfaces, and projection

Turn lines and sketches into solid objects
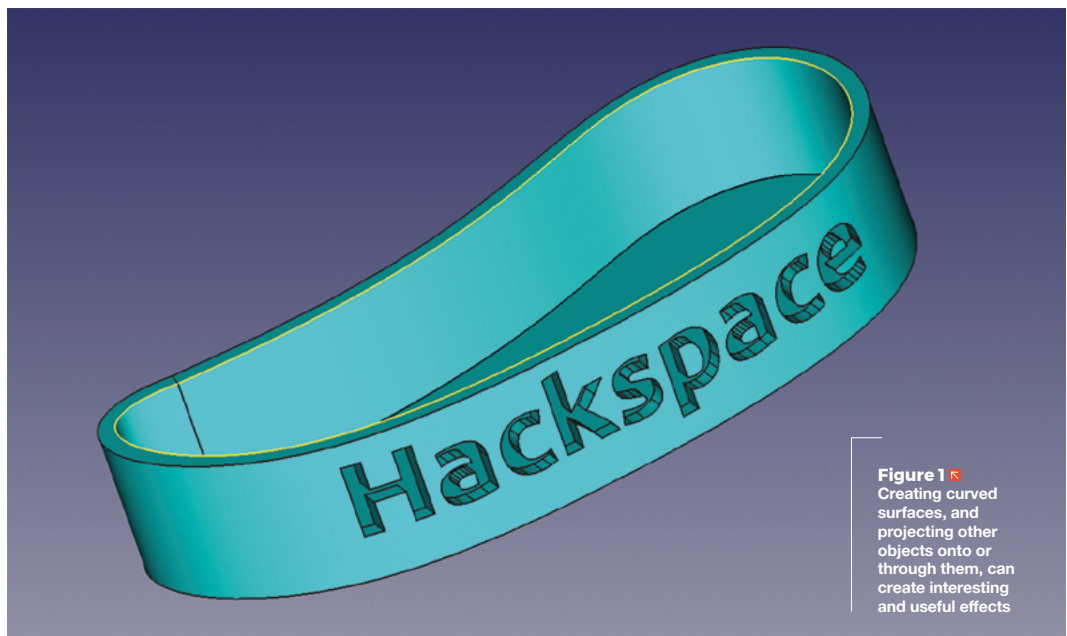
O ften, we've begun with a sketch and then extruded it to create the first building block of our project or design. Similarly, here, we are going to start with a sketch, but we are going to explore creating surfaces from sketches or wires to build an object. To begin, let's go to the Sketcher Workbench 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, and draw a collection of B-splines around the origin point, making sure that the final B-spline ends at the first B-spline point (**Figure 2**). We 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 (**Figure 3**). We don't need to constrain the B-spline that we 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. As a rough estimate, our curvy sketch is around 120 mm at its longest point, and we moved our copied sketch upwards by around 25 mm (**Figure 4**, overleaf).

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 our first surface by clicking the 'Create a ruled surface' tool icon. Hopefully, you now see that a surface has
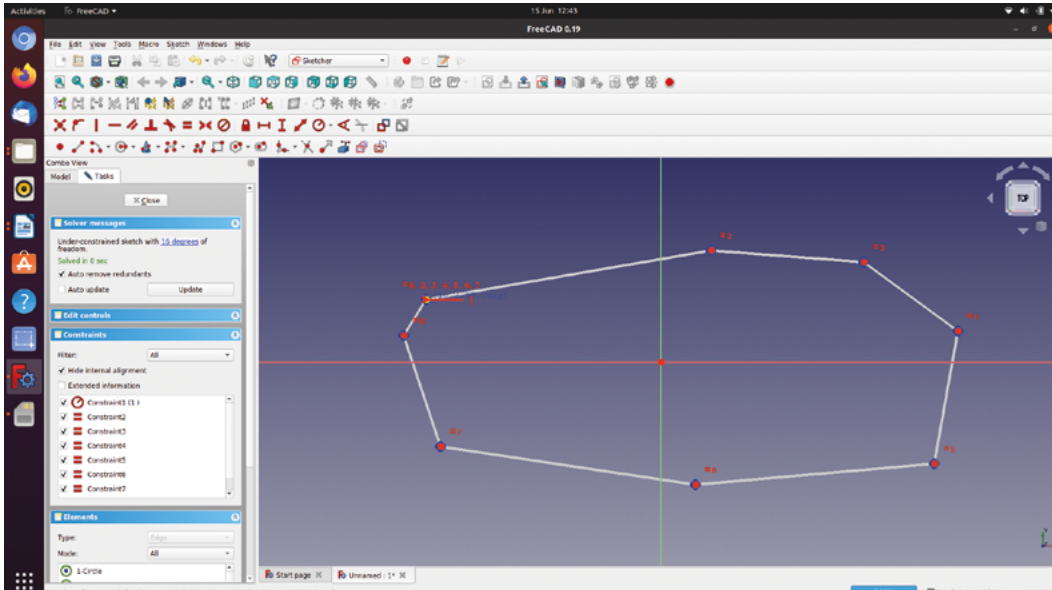
been created between our two sketches, similar to our example in **Figure 5** overleaf.

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

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 our curvy wall, to make it into our curvy tray, we can use the drop-downs in the

> **Doing this incorrect experiment makes us think** about the projection of axes through workpieces

file tree to go all the way back and select our original sketch. We can then click the Extrude tool and add -3mm to the Z axis custom direction and set the length option to 0. Clicking Apply, we should have added a base underneath the offset wall to complete our tray.

Previously in this series, when we have worked with solid objects, we 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 our tray, drawing directly onto the surface is not really possible. However, we 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. We can do this in FreeCAD, except the projector is referred to as the camera view and is simply the →
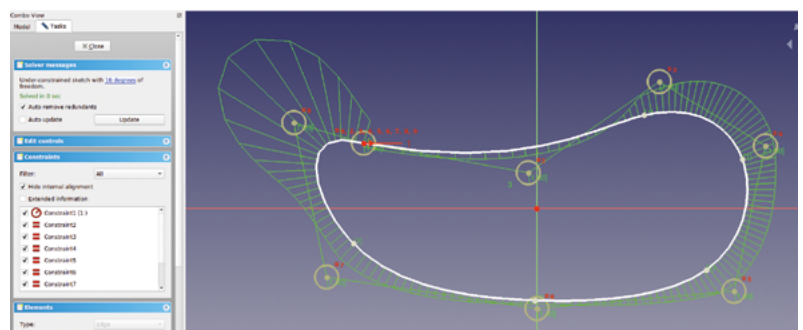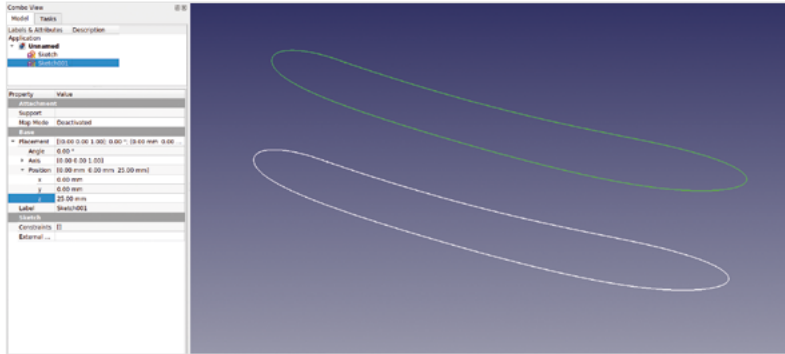
current view in the preview window. The equivalent of your hand is the object you want to add to a surface, and our ruled surface is the object onto which we want to project. Whilst this might sound complex, it's actually pretty straightforward to do.

Whilst we have used the imported, and then extruded, dog logo for this first example of projection, you can use any small extruded part, perhaps a small extruded circle or square. First, we need to choose which surface area of our object we 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 we chose a larger, smoother face. We 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, we then need to move our object we want to project onto the curved surface into a position in line with the projected surface and the preview window. We can do this via the usual right-clicking the object in the file

tree, selecting Transform, and moving and rotating the object until we 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, we have rotated the preview view to show how our 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 we 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 our example, the extrusion height was 3 mm – this gave enough height for the whole logo 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 (**Figure 8**). 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 our curved surface again. Then we 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
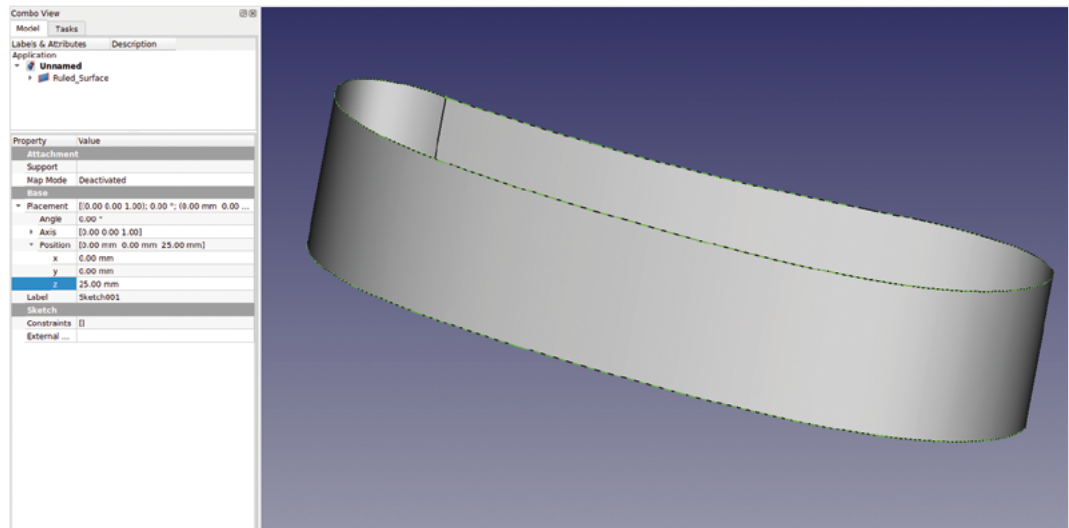
Figure 4 ◈
A copy of our B-spline sketch is created and raised above the original in the Z axis

Figure 5 ◈
Creating a ruled surface between two identical sketches
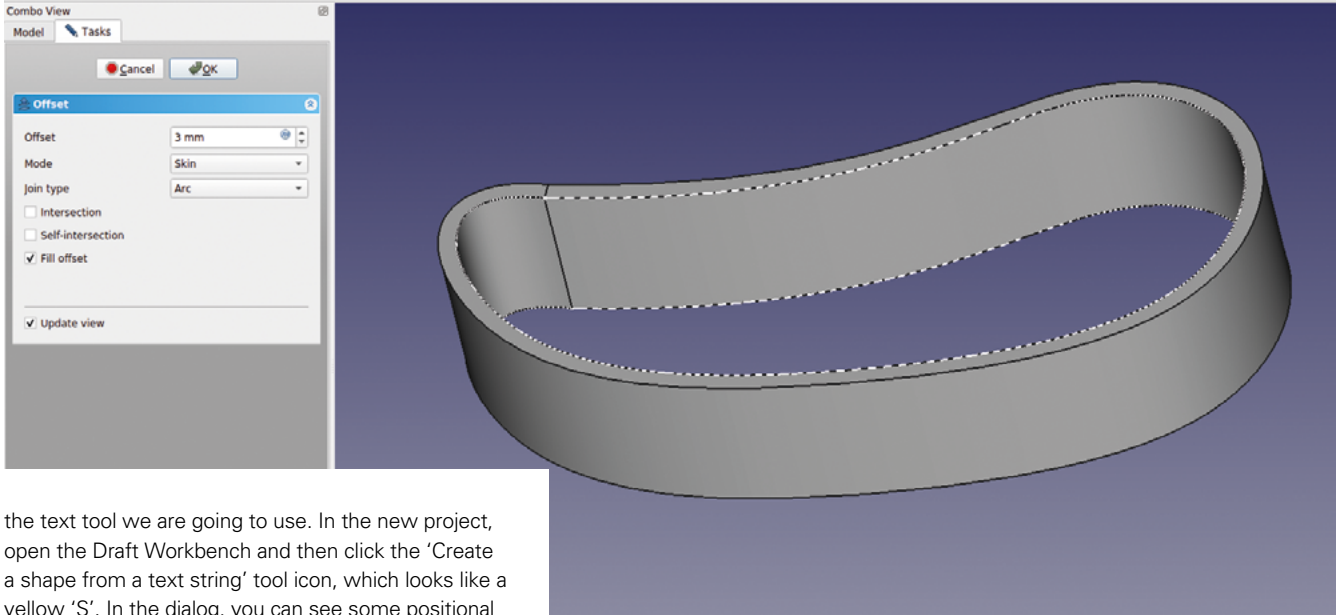
the text tool we are going to use. In the new project, open the Draft Workbench and then click the 'Create a shape from a text string' tool icon, which looks like a yellow 'S'. In the dialog, you can see some positional co-ordinate input boxes which select the point at which our text will be created 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

> **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

at your font installation folder. Clicking the ellipsis at the side of the 'Font file' input box should open a file browser. On Ubuntu flavours of Linux, you will need to right-click in the file browser and check the 'show hidden files' option, and font files can usually be found at **usr/share/fonts**. 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. →

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 we use in this article has some limitations – one main one being that, if you have a closed path like the base of our curvy tray, the ruled surface tool can't be used to create a surface filling the path. This is why we create a small extrusion from the original sketch. If you are interested, create a sketch similar to the first B-spline sketch we 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.
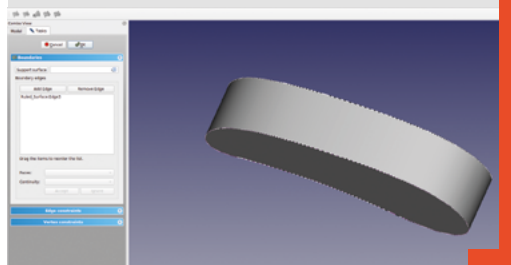
QUICK TIP

You could also project onto a surface using the usual height method, and then use the Transform function to move the projection to a specific depth.
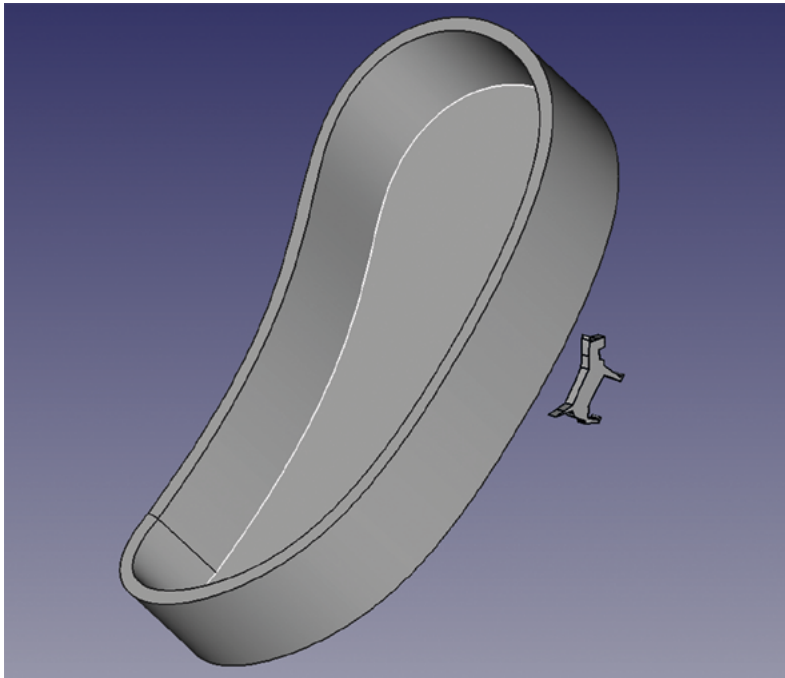
Figure 7
A rotated view
showing the setup
we created to project
the logo part onto
our curved tray wall

Figure 8
Our logo projected
onto our curved
part. It creates a
useful and pleasing
embossed effect

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, and then 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, and then right-

> **The surface tools and projection approaches can** create very complex effects and designs

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 we 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 we can see the projection is inside the cube to a depth of 5 mm. Finally, we 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. This is a useful effect if you are trying to make a text-based sign object for 3D printing (**Figure 11**).

You should also see that you have a 'Shapestring001' object in the file tree (**Figure 9**). 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 we extrude the ShapeString on the Part Workbench a little, say 3 mm, it means that we can use the simple 'Transform' operation we used earlier to move the object into our projection position.

We can now use the text in the same way as we used our small logo or part in the earlier projection example. In **Figure 10**, you can see that we have again projected the text onto the curved side of our curvy tray example.

You may have noticed when we 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

Figure 9
Making a ShapeString object is the basis of creating 3D text effects

PUTTING IT ALL TOGETHER!

As a final experiment, we can put together all the techniques we've covered in this part of the guide. We've returned to our slightly odd-shaped curved tray, and we have once again set it up with our example text ShapeString projected onto it. This time, we have kept our height at 10 mm and set our depth to 10 mm, which provided more than enough depth to fully project the text through the curved wall of the tray. You can see above that it looks a little messy, but in **Figure 1** you can see the result when we have performed a cut operation to remove the projected object from the tray. It's an interesting fretwork-type effect that can make a very pleasing object, although for 3D printing we would have to model some struts to keep the floating parts of the lettering in position!

The surface tools and projection approaches can create very complex effects and designs, but we mentioned earlier that they have their limits. How, for example, could we create a pattern or text that flows completely around a cylinder or other curved shapes? Next up, we'll look at other tools and techniques to get us solving these issues, and more. ◻

VECTOR GRAPHICS

For our example of projecting a part onto a curved surface, we have used a small dog logo, which was imported as an SVG and then extruded into a part. The SVG file for the dog logo is a single path outline – experiments with more complex SVGs had varied results.

The ability to import an SVG is a really useful feature occasionally, and works well. We simply went to File > Import and navigated to our SVG. When selected, we get a dialog asking if we want to import as a 'Drawing' or 'SVG as Geometry'. We need to select the latter, and then the dog logo is imported and appears in the file tree as an object labelled 'Path'. On the Part Workbench, if we select the imported SVG and then click the Extrude tool, we can extrude in the usual way. We extruded the little dog logo to 2 mm.

There are also ways to import an SVG and convert the SVG into a sketch so that we can constrain its contents, making it usable in other Workbenches. A useful video tutorial on this, by the author, is available here: **hsmag.cc/ImportSVG.**

Design your own ring

Learn to work with curves as
we make a piece of jewellery

Figure 1 ◆
An example of mapping sketches around cylinders, this 'maker chic' 3D-printed ring acts as a spanner for M2 to M5 nuts!

In the previous section, we looked at
projection as a tool to add geometries
or objects onto curved surfaces. In this
month's tutorial, we will discuss the limitations
of projection and look at how we can map
sketches onto the surfaces of curved objects.
Along the way, we will make a geeky maker ring
to 3D-print, and we will also explore some more
complex uses of the Curves Workbench.

Last month we 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 we can
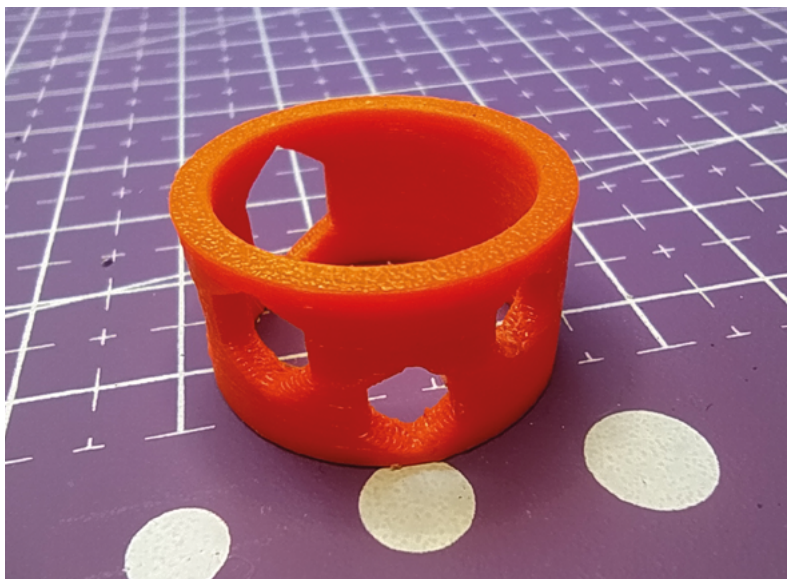wrap a sketch around a cylindrical object.

Open FreeCAD and use the Addon Manager to add
the Curves Workbench. Click Tools > Addon Manager.
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 our 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 our 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 our mapped sketch. For a first simple example,
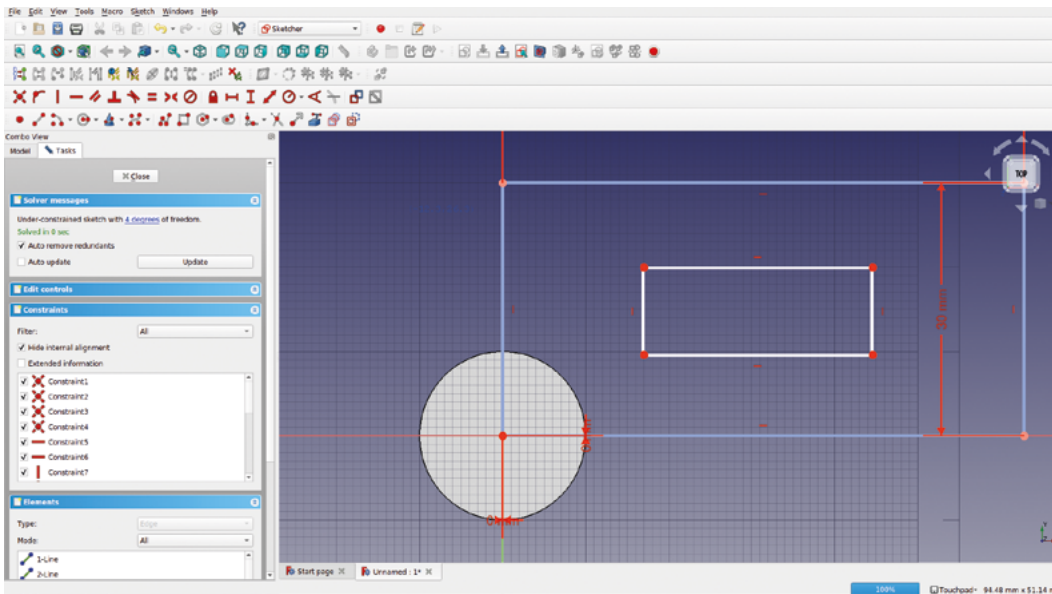
let's draw a rectangle somewhere inside the blue construction rectangle (**Figure 2**).

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 we 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, we 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 our cylinder (**Figure 3**).
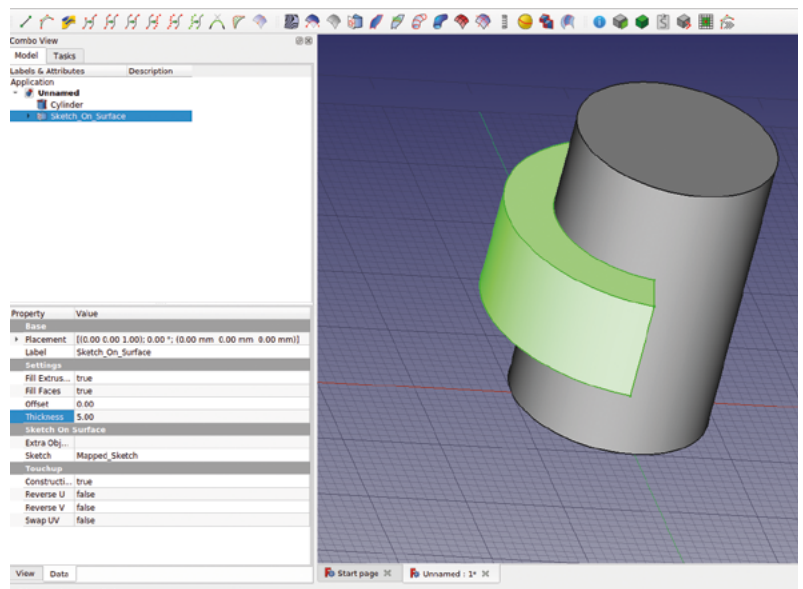
It's perhaps worth pointing out that the resulting objects made using mapped sketches work in the same way as other objects with regard to the Part Workbench and other tools. In **Figure 4** (overleaf), you can see that we have added some fillets to the edges of the part and used the appearance menu to set the object's colour. We've also gone one step further in that we have clicked the surface of our 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, we can use Part Workbench tools such as the Boolean operation type tools to make cut-out parts using the mapped sketch. As an example, we thought of a fun mini project to make and 3D-print. We are going to make

a useful (ish!) 3D-printed ring. Let's 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. →

Figure 4 ◈
Figure 4 ◈
Parts created using mapped sketches behave similarly to other objects, and you can add fillets and chamfers and change their appearance settings as usual

Figure 5 ◈
Our mapped sketch laid out and partially constrained with the M-series nut dimensions, plus a small clearance allowance

Figure 6 ◈
Our hexagonal spanners extruded through the ring using a negative thickness value

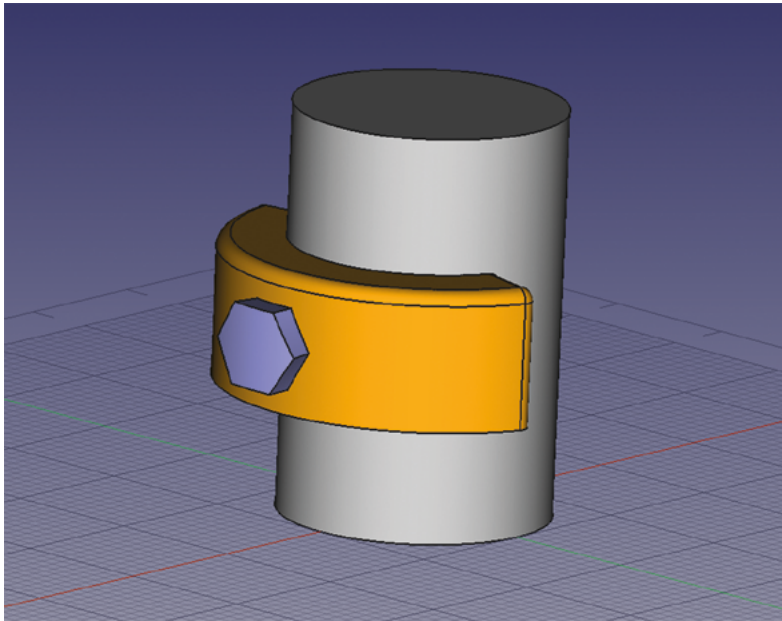Moving to the Curves Workbench, we will map a sketch to the outside surface of the ring as we did earlier with our cylinder. Click the outer surface and then click the 'Map a sketch on a surface' tool icon once again. Click to drop-down and double-click the mapped sketch, which will open in the Sketcher Workbench.

Our idea for this maker ring is that we will cut out some hexagonal shapes that not only look decorative but will serve as an emergency small spanner tool. As such, inside our construction geometry, we added some hexagons sized to common small metric nut sizes. We went for M2, M2.5, M3, M4, and M5. You don't need to fully constrain the sketch, but we opted to align one line of the hexagon with a vertical constraint so that we could then constrain the nut size by selecting two opposite points and applying a horizontal dimension. We added roughly 0.1 mm to the M-series given nut sizes to make them an easy fit – you can see the sizes we used in the constraint values (**Figure 5**). We preferred our hexagon spanners positioned randomly, but 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 our 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. Our ring is 2 mm thick, so we went with -2 mm. This should now have extruded the mapped sketch through the ring (**Figure 6**).

Now that we 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 our ring. We can then use File > Export to save our ring as an STL file suitable for slicing and printing. You can see the ring we made in **Figure 1**.

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.

We are aiming to make an object that looks like an aeroplane cockpit canopy cover as it's a challenging shape to model. Again, we are not particular about the dimensions of the object and, as such, we are not worried about constraining aspects of the design.
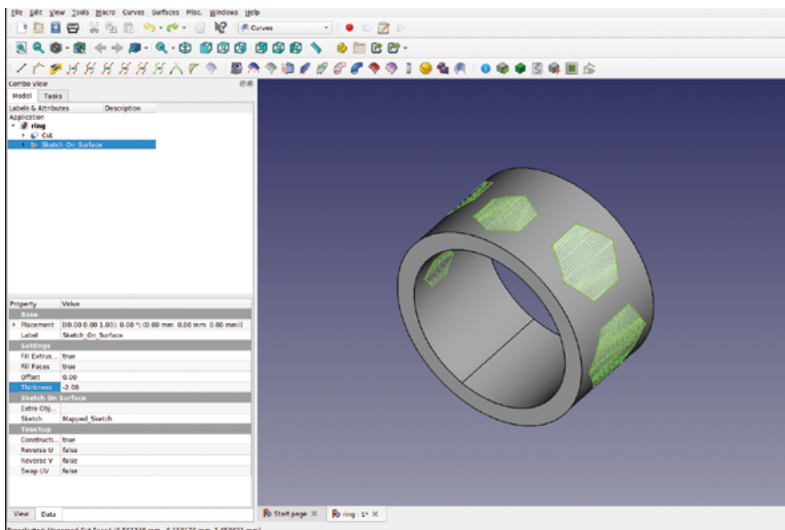
To begin, create a new project, move to the Part Design Workbench and create a body. Create a sketch in the XY plane; in this sketch, let's create a B-spline. This B-spline is going to be the lower side edge profile of our canopy cover. Click the B-spline tool and start the B-spline attached to the X axis to the left of the origin point. Add some points and then finish the B-spline to make it look similar to the left-hand side of **Figure 7**. Before we close the sketch, select the B-spline that we 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 (**Figure 7**).

Create another sketch in the active body, but this time in the XZ plane. We 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 we have those points available, draw another B-spline 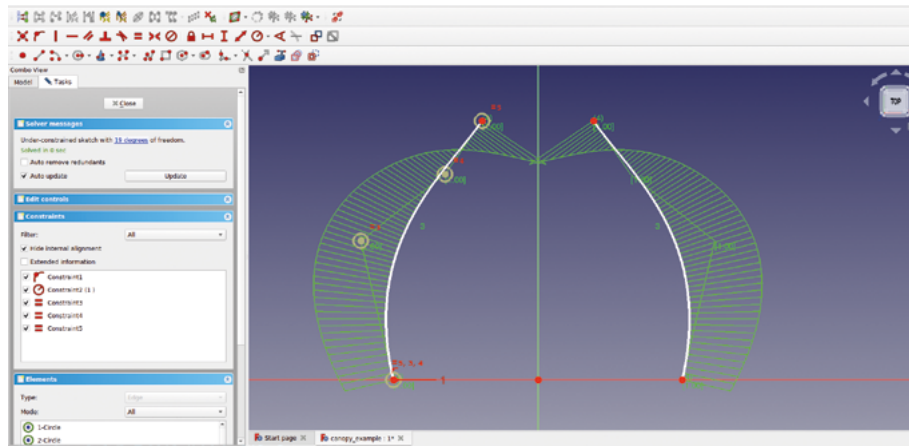that begins with the point constrained to the left-hand B-spline from the first sketch. Place the second B-spline point on the Z axis line above the origin and attach the final B-spline point to the imported edge of the mirrored B-spline. →

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 our 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 – we 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 we 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. We went for a 'slot' object. Close the sketch to return to the Curves Workbench. Finally, select both the 'additive loft' object that is our 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 our slot onto the cone's surface. Similar to earlier examples, we can add thicknesses and perform cut operations to create our design.

> Creating datum planes is an incredibly useful **tool in many projects**

Figure 9
Setting up a new datum plane to attach a sketch to

Figure 10
Creating a ruled surface is the first step of creating our final curvy canopy

When you finish this B-spline, you should have a symmetrical curve that will form the rear edge of our canopy design (**Figure 8**).

We 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 our canopy design. Of course, these points don't sit on a datum plane, so we 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 our new datum plane, let's make the three standard plane origins visible. To do this in the file tree, highlight 'origins?'

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 we imagine our desired new datum plane by visualising where we want to sketch, we could describe the plane as a sort of XZ plane moved to the far end of our original mirrored B-splines. This kind of hints as to how we will create the new datum plane. First, let's click on the XZ plane that we 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 'XY_plane' in the input, and the button now says 'Plane'. Click the button until it returns to 'Selecting'. Finally, we need to move and zoom to select the endpoint of one of the mirrored B-splines we made in the original sketch. This should make that vertex point the reference of our 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 (**Figure 9**).

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 we did the sketch for the canopy rear on the XY plane.
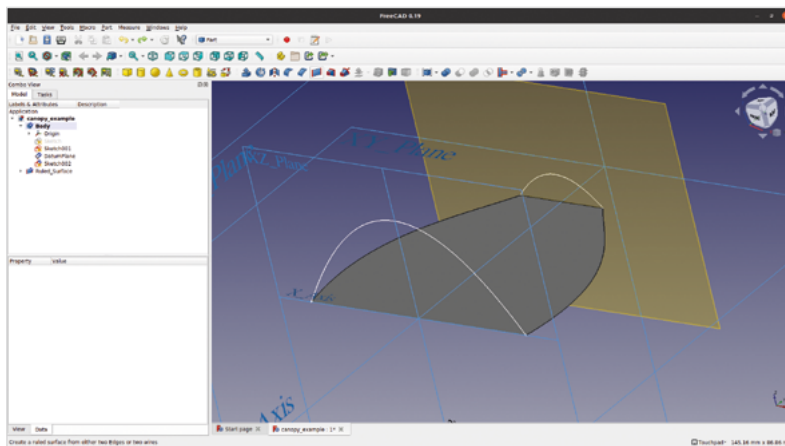
To create a surface for our canopy, first we are going to make a flat ruled surface between the original sketch lines and then we are going to use the 'Sweep profiles on 2 rails' tool and the 'Approximate points to NURBS curve or surface' tool. To create the ruled surface, move to the Part Workbench, select the two B-splines of the original sketch, and click the 'Create a ruled surface' tool (**Figure 10**).
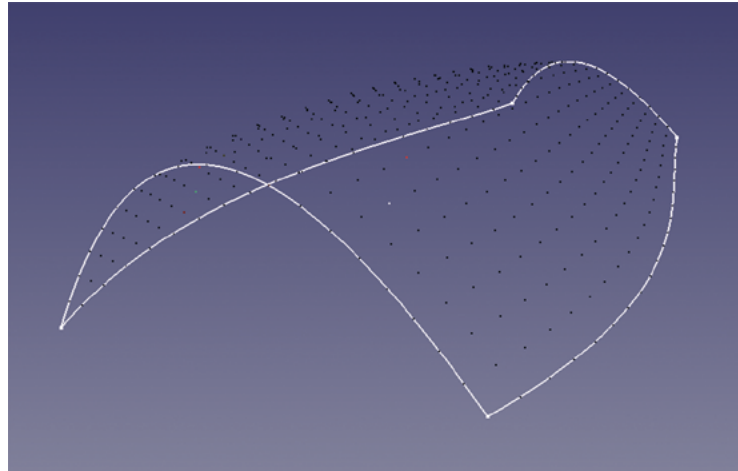
Moving to the Curves Workbench, in the file tree view, select the new ruled surface object and select the two sketches that create the rear and front edge shape of our canopy design. With these three objects highlighted, click the 'Sweep profiles on 2 rails' tool. Your computer may take a little time 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 our aeroplane canopy (**Figure 11**). 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 our earlier ruled surface in that it has no thickness dimension – we 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 (**Figure 12**).

We've seen a small range of tools on the Curves Workbench in this chapter, 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. □

Figure 11
Creating a point cloud which we can convert into a curved surface using the Curves Workbench

Figure 12
Our completed canopy design finished by creating a filled offset

FreeCAD: Working with circuits

An open-source workflow for electromechanical creations
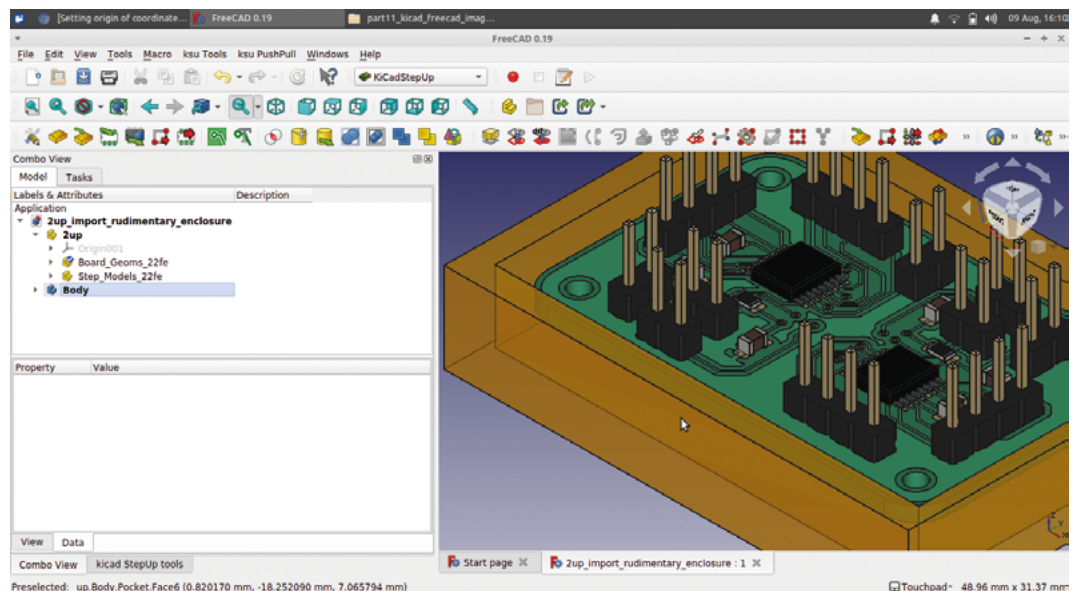
KiCad is an excellent open-source PCB design package that can render populated 3D images of circuit board designs.** In combination with FreeCAD, we can create custom component models for KiCad or import KiCad PCBs whole, allowing us to design enclosures or other surrounding systems. Let's check out some FreeCAD tools that simplify these processes.

One of the concepts in KiCad is that a schematic component is assigned a physical footprint from a list; that footprint consists of numerous items: copper pads, silkscreen drawings, and more, which all appear in your PCB design. Commonly, a 3D model is assigned to the footprint so that you can view your components and your PCB in 3D. Occasionally a 3D model might be missing or might not fully represent your component, or indeed, you might be working with a custom footprint and

component and need to create a 3D model for your part. FreeCAD has us covered for this – there's an add-on workbench that contains specific tools to make this process much easier.

Start FreeCAD and then install the KiCadStepUp Workbench. To do this, go to Tools > Addon manager and find the KiCadStepUp Workbench in the list, then click Install. Once downloaded and installed, you will be prompted to restart FreeCAD. Once restarted, you should see the workbench in the workbench list. Start a new project in FreeCAD and move to the KiCadStepUp Workbench, where we'll first take care of a little housekeeping.

Once in the workbench, click Edit > Preferences and scroll down the icons on the left-hand side to find the preferences for the KiCadStepUp Workbench. Click that icon, and at the top of the page, there is an area to set up the paths to where KiCad stores its 3D models. To find these, open
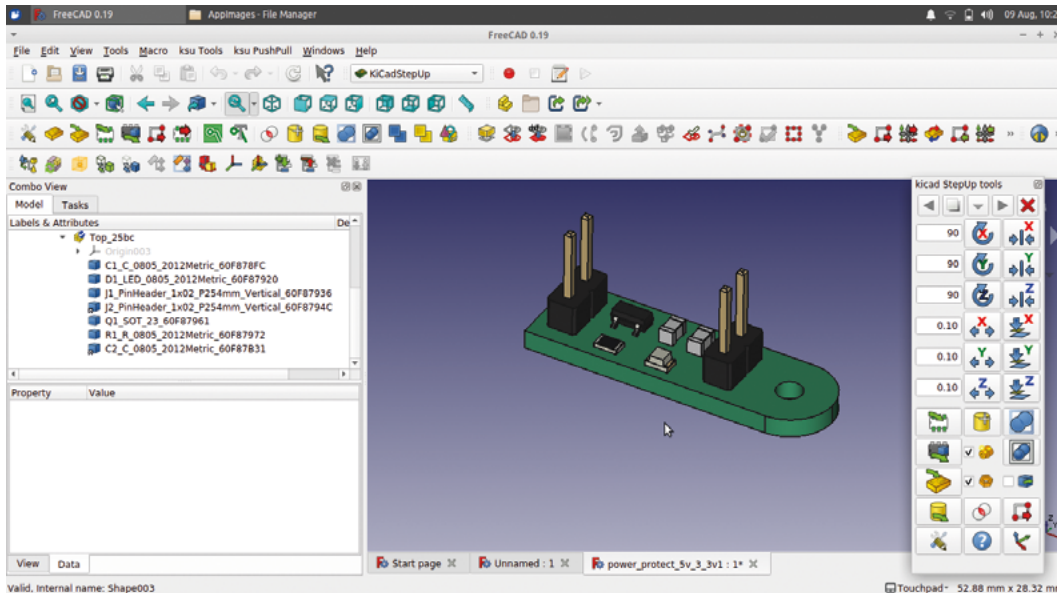
KiCad and click Preferences > Configure Paths, this should open a dialog box that shows the locations and paths set within KiCad. Copy the paths to the relevant 3D model file locations and put them in the boxes in the FreeCAD KiCadStepUp preferences dialog.

The first thing you might want to do in FreeCAD is to move over to the KiCadStepUp Workbench and do a quick test to import a 3D model of a KiCad PCB. In FreeCAD, open a new project and then move over to the KiCadStepUp Workbench and click the yellow icon with a green arrow which reads 'Load KiCad PCB Board and Parts' as you hover over it. Navigate to one of your KiCad PCB board files (the file in the KiCad project that ends .pcb) and open it.

You should notice a couple of things (**Figure 2**). The first thing is that when you use a tool icon on KiCadStepUp, it automatically launches a floating toolbar with a collection of the most commonly used KiCadStepUp tools on it. You can move this out of the way, or indeed close it, but the toolbar will persist when you move to other workbenches, which, later in this tutorial, we will find useful!

Secondly, you should see your PCB import as a 3D model. You will receive a couple of messages about the import – just close these, and you should be left with your PCB in the preview window. The imported PCB doesn't have the traces visible or the silkscreen items. You can import the silkscreen using a KiCad DXF export and importing into FreeCAD, but most of the time, we probably won't need that level of detail in FreeCAD. You can create

the copper traces and pours – we will look at this later in the tutorial – but doing so adds complexity to the model that you may not need and might make it difficult or slow to move the PCB model. You will also see that the different components on the PCB are listed in the file tree view as separate parts. This means that the components can be treated in a similar way to any FreeCAD part. You can toggle →

GETTING PERMISSION

It's highly likely that if you are creating 3D models for KiCad components, you will want to save them into the 3D packages folder so that you can quickly find them and use them in KiCad. We run KiCad on Linux, and by default, when installed, the **library** and **modules** folders are only read and writeable as the root user – the general user won't be able to make changes. On Linux, we can use a terminal to switch to be the root user and use the classic `chmod` command to make the relevant KiCad folders read and writeable for our regular user login. Press **CTRL**, **ALT**, and **T** to open a terminal, then switch to root user using the command `sudo su` and put in your password when prompted. Once logged in as root, input the following command using the file path to the folders you want to make read and writeable. We've opted to make all the folders inside our KiCad **modules** folder read and writeable, and as such, our command and file path read: `chmod -R a+rw usr/share/kicad/modules`.

individual parts' visibility, for example, by highlighting the part in the file tree and pressing the **SPACE** bar.

Sometimes when working in KiCad, you'll design a board using a component or components that don't have a 3D model for you to use in the 3D viewer. Or indeed, you might be creating a custom footprint and custom components for use in KiCad and they won't have a standard 3D model or models. We say 'models' because often in KiCad, an object or part in the 3D viewer will have multiple models lined up to create the 3D view. In **Figure 3**, you can see a PCB we are working on that uses a 4.4 mm wide TSSOP16 package IC which is missing in KiCad's 3D viewer. The footprint is there, and we can see the copper pads, vias, and silkscreen, but the actual placed component isn't there. The footprint that
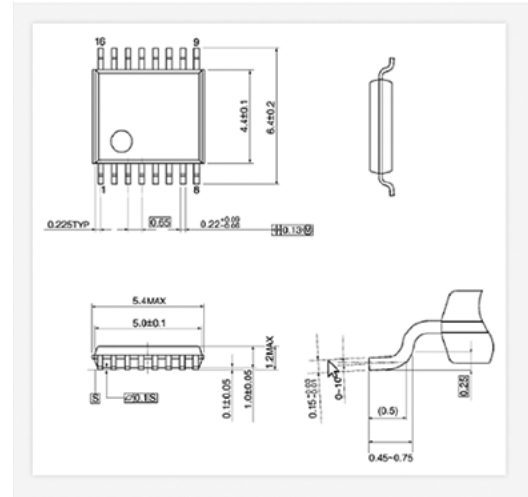
is there is actually a 3D model of the small part of PCB board it populates and all the geometry of the footprint. If we create a model of the component that would be placed on top, we can add this to the list of items included in the KiCad 3D viewer to complete our 3D board view.

The first step of this is to identify some accurate dimensions for the part. Most electronics components will have these details in their datasheet, which are available online. We found a diagram for the correct TSSOP16 package that matched our component and used this to create our simple 3D model.

Armed with our dimensions, we fired up FreeCAD and began to model our part. We went for a simple approach that dimensionally represents the component but is only representative in design and lacks detail and refinement. We used the Part Workbench and began with a rectangle sketch that we cloned 16 times and aligned to create the pins of the IC component. Using the simple clone tools on the Sketcher Workbench is definitely useful for creating multiple similar parts such as these pins. Our IC is a little over 4 mm wide, and as such, each pin rectangle was 0.65 mm wide. It's worth noting that at this scale, FreeCAD will automatically slip between mm, µm, and nm when you are applying constraints, but if you replace the unit letters, you can, of course, input them as fractions of a mm.

Having laid out our 16 rectangles to represent the pins, we extruded the sketch on the Part Workbench up to a height of 0.5 mm. Right-clicking on the extrusion in the file tree, we clicked Appearance and set the colour to a silver/grey (**Figure 5**).

We then added a second sketch via the Part Workbench and drew a simple rectangle that

CENTRE LINES

When we found the dimensions for our IC package, we noticed that the dimensions for the pin spacings were drawn and labelled relative to the centre line of the pin. As we had this information, it made sense to use it, and as such, we added an extra point constrained to the centre of the short vertical line of the pin. This meant that we could then use this point on each pin rectangle to positionally constrain the pin without converting any dimensions off the original drawing. To add a point in the Sketcher Workbench, you simply use the 'Create a point in the sketch' tool.
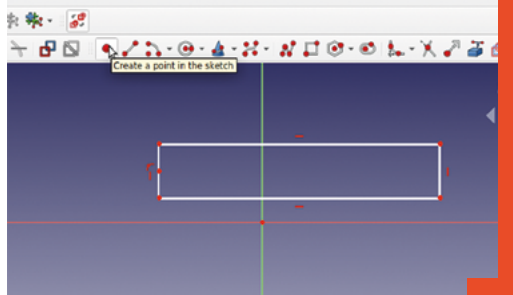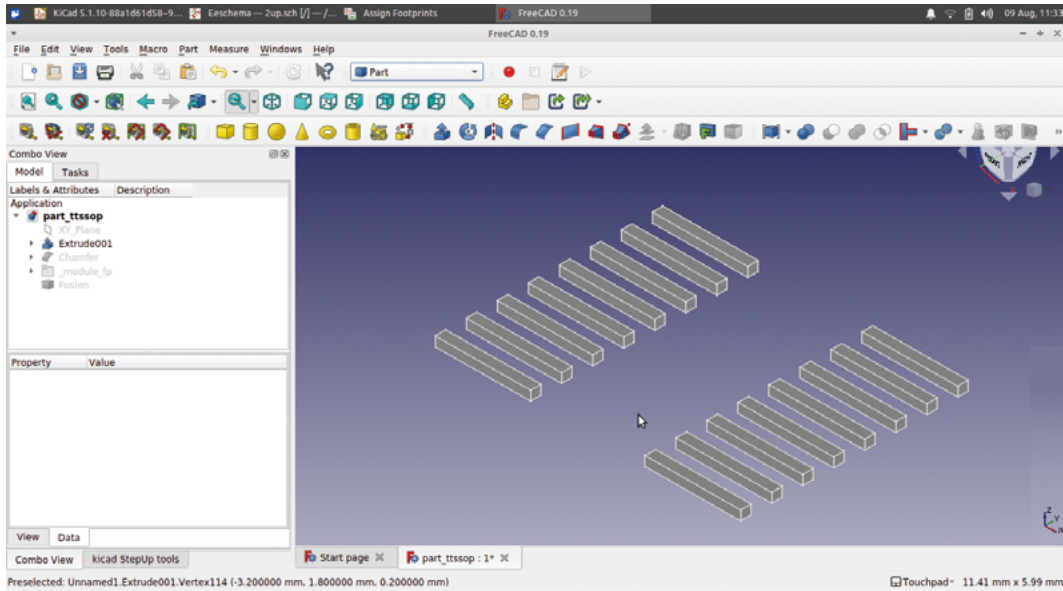
would represent the body of the IC component. As we had positionally constrained our pin sketches around the origin point of the XY datum plane when we constrained the rectangle, similarly it meant it was in the correct position. We then extruded the rectangle to the height of the IC package taken from the datasheet. We added a tiny chamfer to the upper outer edge of the extrusion as a token nod to modelling prettily! Finally, we changed the appearance of the extruded rectangle to black – you can see this in **Figure 7**. Once we were happy with the look of our package, we selected the pins and the body of the IC model and used the Union operation on the Part Workbench to create a single fusion object. This is important as KiCadStepUp can only export a single object as the 3D model to be used in KiCad.

Having created our simple model, the KiCadStepUp Workbench has some great tools allowing us to check the fit and position of the model relative to the KiCad footprint model. On the KiCadStepUp Workbench, click the 'Load KiCad PCB Footprint' tool and navigate to the footprint file for the component you are modelling.

Select the footprint and it will be imported into the preview window of FreeCAD and appear as a folder of items in the file tree view (**Figure 6**). When

> ## We used the Part Workbench and began with **a rectangle sketch that we cloned 16 times**

we imported the footprint, it aligned pretty well with our model of the IC, but we wanted to move it very slightly. If you have created your model away from the origin point, you can right-click and select Transform in the usual way to move the part around in the preview window to get it roughly positioned. Then in the KiCadStepUp toolbar, there are some clickable buttons for the X, Y, and Z axes, and you can set tiny increments in their dialogs. We tweaked our model in increments of +- 0.01mm to get the pins of our package well →

Figure 5
Our very simplified approach to creating the IC pins – the model will be representative and will be accurate in the area that the component sits in, but isn't finely modelled

Figure 6
Loading the KiCad footprint file into FreeCAD allows us to position and check our 3D-modelled component
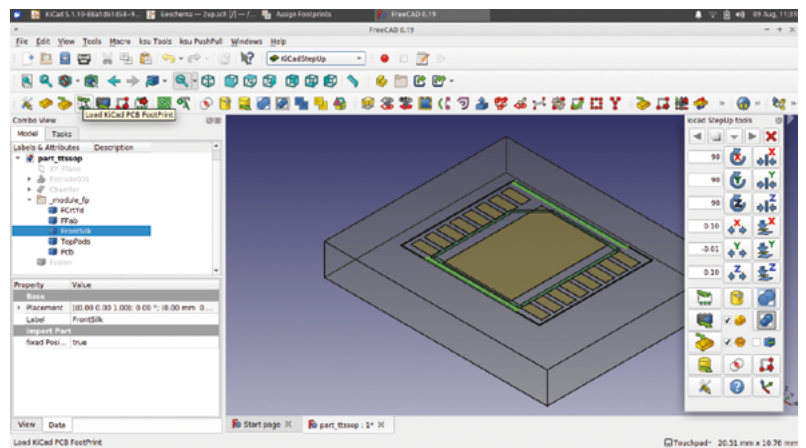
aligned on the pads of the imported footprint (**Figure 7** overleaf). You can also see that the KiCadStepUp toolbar is available for use when open and undocked, even though we are on the Part Workbench. This is a really useful feature.

Once you are happy with the position of your component, it's time to export the model to the correct file path to be then used in KiCad. Again, with the model selected as our single fusion object in the file tree view, we clicked the 'Export 3D model to KiCad' tool icon. When you click this icon, some dialog boxes will appear: the first two are 'mesh deviation' and 'crease angles', which we left with their default values. The next two dialogs ask if you want to change the colours of the parts of your model from their current values to suggestions that match common options used in KiCad. We left both our colours on the 'as is' option to use the colours set in FreeCAD, but feel free to experiment. Once you have OK'd the dialogs, two files will be created: a WRL file and a STEP file. On our machine using the stable Linux app image of FreeCAD 0.19, these two files were written to the folder that our app image is stored in. Find the two created files on your machine and note that they are titled with whatever the highlighted part was called in the file tree, in our case, 'fusion'. We renamed both the files to a more useful and descriptive name, **TSSOP16_4.4mm** – we then copied and pasted them into the correct KiCad folder. In our instance of KiCad, the 3D packages are located as **usr/share/KiCad/modules/packages3d**. Within that folder,
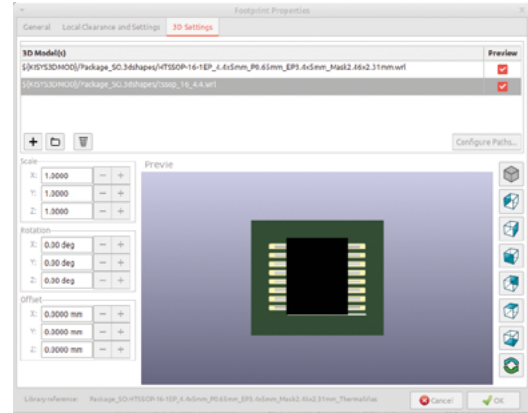
we then navigated to the folder that would make sense to store the model we had created; for our model, this was the **Package_SO.3Dshapes** – we pasted our two files here. Having saved everything, we can now close FreeCAD and launch KiCad once again. Opening up our PCB project, we now need to assign the additional component model to our footprint. There are a few different places in KiCad that we can create this association, but as this is a complete PCB project, we did it from within PCBNew.

Opening the project in PCBNew, we selected the component footprint and pressed the **E** key to edit it. In the resulting 'Footprint Properties' window, there is a tab called '3D Settings'. At the top of this tab there is a window with a list of 3D models currently associated with this footprint. In this case, there is just one model, and the model is the 3D render of the footprint without the placed component on top. Brilliantly, KiCad allows you to connect multiple models to each instance of a footprint, so we can add our component here. First, click the '+' icon to create a new empty item in the list. Next, click the folder icon, which is next to the + icon, to set a path to the component model file. Navigate your way to find the WRL version of the files we exported from FreeCAD and moved to the correct KiCad module subfolder. Once selected, you should see a preview of the model. Click OK and check that the new listing is ticked in the checkbox (**Figure 8**). Close the Footprint Properties window and then select View > 3D Viewer. If everything is well, you should now see your PCB rendered with your nice new 3D component model. In our project,

> **KiCad allows you to connect multiple models to each instance of a footprint, so we can add our component here**

Figure 8
Adding our new 3D model to be rendered in position on our PCB project in KiCad

Figure 7
With the fusion part selected in the file tree, you can use the translation tools from KiCadStepUp Workbench to nudge the part into an exact position
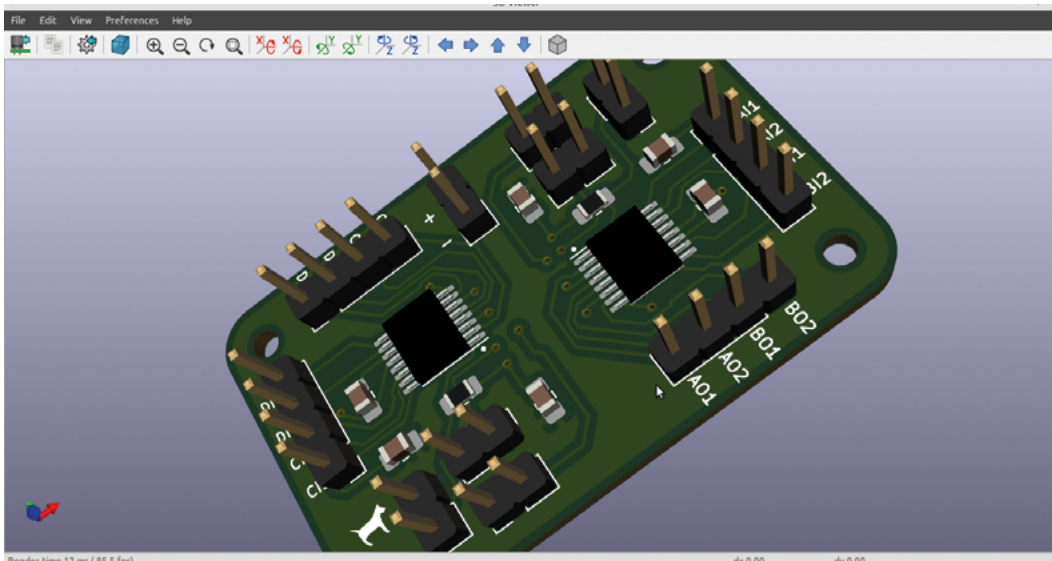
we repeated the above steps twice as we had two
of the same IC component on our board. You can
see the 3D view in KiCad is now complete with all
its components (**Figure 9**).

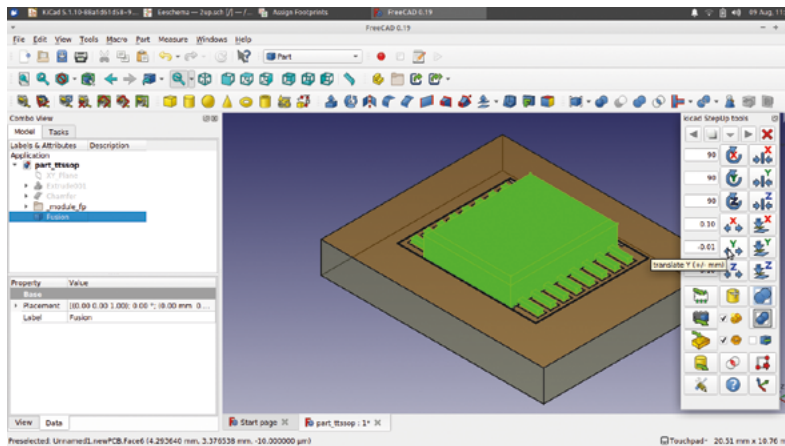While creating models for missing 3D
components for KiCad is a great achievement,
we have only really scratched the surface of what
the KiCadStepUp Workbench can be used for.
Of course, the most obvious use is to be able to
create CAD models of projects around the PCB
model. All the usual FreeCAD tools are still at our
disposal, and as a super-quick example, we made
a simple tray enclosure using a body in the Part
Design Workbench (**Figure 1**). You can use all the
usual tools and tricks; for example, you can toggle
between section and full view in the Sketcher
Workbench with the PCB model behaving like any
other object. We created a pocket in our padded
enclosure idea and moved the PCB model into a
rough position inside the pocket. As an example of
how useful this tool can be, we highlighted both
the enclosure body and the PCB model in the file
tree view and then clicked the 'Check Interferences
and Collisions' tool icon on the KiCadStepUp
Workbench. It took a short while to process, but you
should see that the Report View window opens and
a list is slowly generated of any components that
collide with any other components in the complete
design. If you have parts of your PCB touching or
overlapping your enclosure or design, they will
be picked up in this operation; however, it will
also list the touching components on the PCB, for
example, where a pin passes through a hole. This
is useful in itself, but you might also notice that it
creates collision parts in the file tree – these are

small 3D parts created only where collisions and
interferences occur. It's not immediately apparent
why this might be useful, but if you consider that
you could use these collision parts in combination
with boolean operations to cut away parts of an
enclosure where objects are overlapping, you can
see that these models have merit, and, of course,
if they aren't of use, you can simply delete them! ◻

FINER DETAILS

Often a simple model of a PCB is going to be most
useful, enabling us to design around it and see how it
fits within our design. As such, KiCadStepUp imports
the simplest version of the PCB file that it can, simply
the board edges and the component models, so we end
up with a model that has the dimensional geometry of
our PCB. However, we can also use KiCadStepUp to
add more detail to our PCB model. One of the simplest
ways is to add the traces and copper pour layers into
our 3D model. To do this, select the imported PCB part
in the file tree and then click the 'ksu tools Add Tracks'
tool icon. When you click the item, reselect the KiCad
PCB file for the project and then click Open. This may
take some time and also will increase the complexity of
the model, but you should see your traces and copper
pour appear on your PCB model.
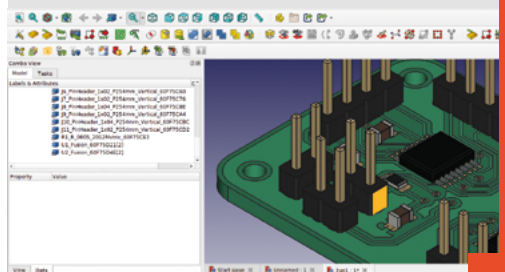
Getting Meshy in FreeCAD

Get your designs ready for 3D printing, CNC milling, and more

S**TL 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. In **Figure 2**, you can

see we have created a cube object and a sphere object in the Part workbench and cut part of the sphere out of the cube.

With the cube 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 our exported part. Put simply, FreeCAD makes a choice about the number of triangle facets our exported STL mesh object contains.

Sometimes we might want to have more control over the mesh that's created for our part, and we might want to change the resolution to get the best results for our project. This is one area where the
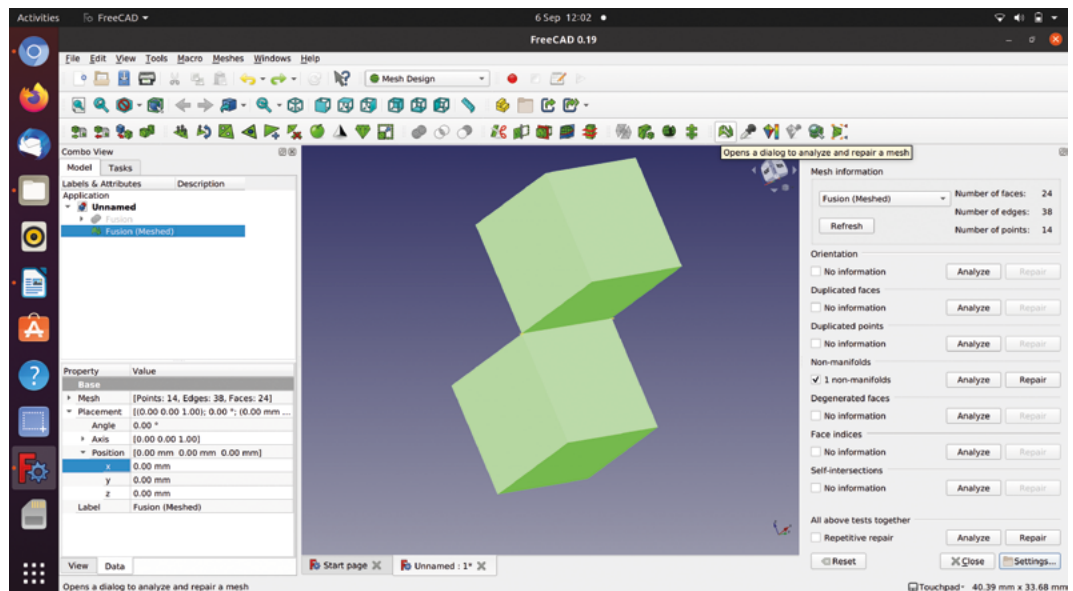
Figure 1 ◈
A simple example of a mesh with an issue we have discovered using an automated analysis tool
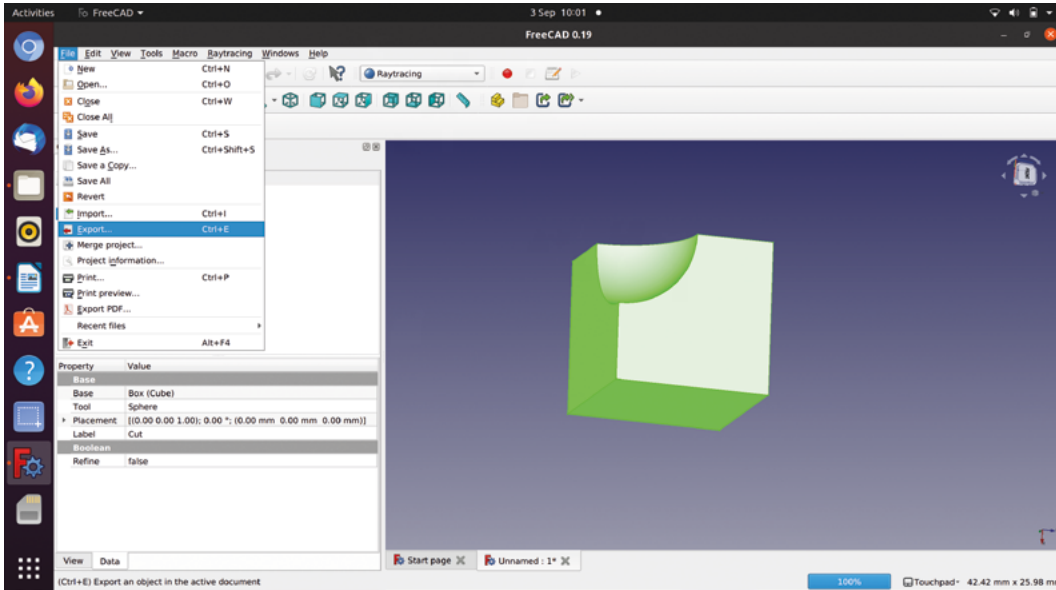
more specialised tools on the Mesh workbench can help. As an example, we have a nose cone part for a rocket designed – let's move to the Mesh workbench using the drop-down menu to create some different resolution meshes to compare.

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 0.10mm, and the second value, 'Angular deviation', should be 30 degrees. If they aren't, set them to these values and then click OK (**Figure 3**).

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 we just see the mesh object in the preview window.

You can see in **Figure 4** that the created mesh consists of lots of triangles closely approximating the part, but we can adjust the values to increase the
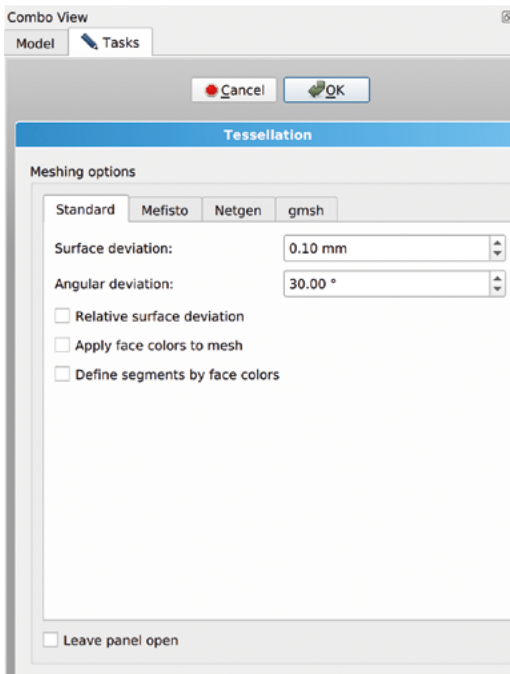
> **FreeCAD has made a lot of sensible choices about the resolution of the mesh of our exported part**

resolution of the mesh if we delete our first mesh, make the nose cone part visible, and highlight it so we can create a new mesh. For this mesh, let's set the surface deviation to 0.03mm and the angular deviation to 10 degrees. You should see, similar to **Figure 5**, 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 we tend to export as 'Binary STL' and, →

QUICK TIP

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.
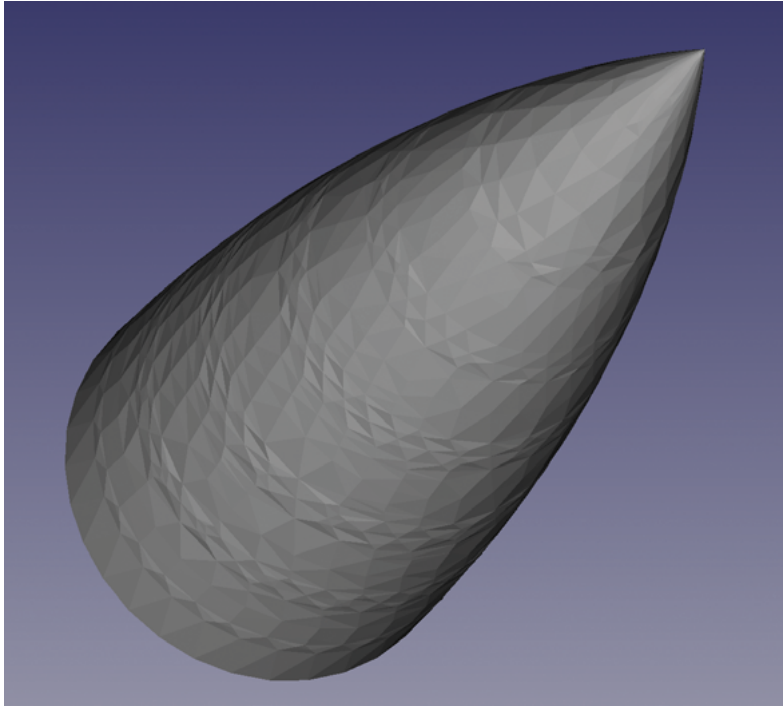
Figure 4 ◈
Our first attempt at creating a mesh of the nose cone part

Figure 5 ◈
Our second mesh resulting in a higher resolution mesh, which more closely matches our nose cone part

these seem to work well with our 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 we 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 we 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, we have an STL file of a hands-free door puller. Let's pretend that, in use, we decided it would be better if the hook faced the other way round. We could, of course, with the knowledge within this FreeCAD, 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 (**Figure 6**).

The Mesh workbench has some tools that are similar in approach to the Part workbench, where we create simple solid objects and combine them

> The Mesh workbench has some tools that are **similar in approach to the Part workbench**

using Boolean functions to stick objects together, or subtract one shape out of another. If we want to turn around the hook part of our mesh file, we 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. We 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, we can then move the cube into place over the hooked end of the door puller (**Figure 7**).

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 we toggle the cube item to be invisible and then select the intersection item, we 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 (**Figure 8**).

All that remains is to move and rotate everything into the new position, and then we can use the 'Union' Boolean operation to create a new single mesh object. We can then highlight the new door puller union object and click the 'Export a mesh to file' tool to create our new STL file ready for 3D printing.

Continuing to use our door puller tool project as an example, let's explore how we create a Part object from a mesh. Moving to the Part workbench, we can highlight the mesh – in our 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 our door puller object (**Figure 9)**. Let's make our mesh object invisible in the file tree to explore our 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 our 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 we 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 →
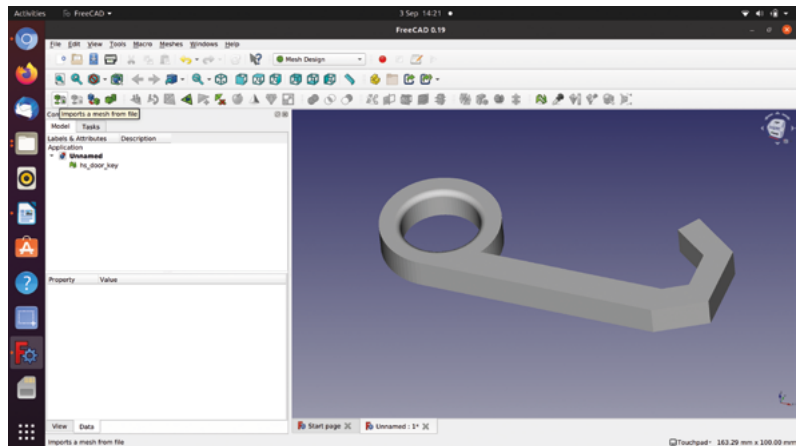
OPENSCAD

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 we 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.
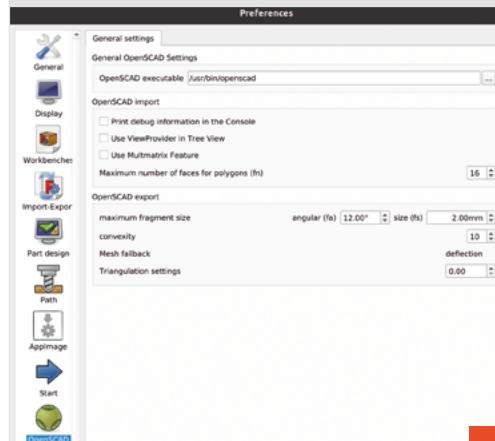
Figure 7 ◈
Creating a mesh solid cube and positioning it over the part we intend to rotate
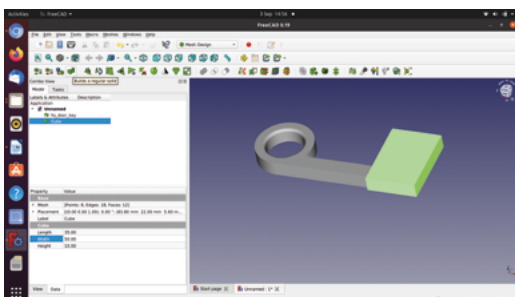
QUICK TIP

You might want to install the brilliant Rocket workbench using the Add-on Manager if you want to play with the nose cone design.
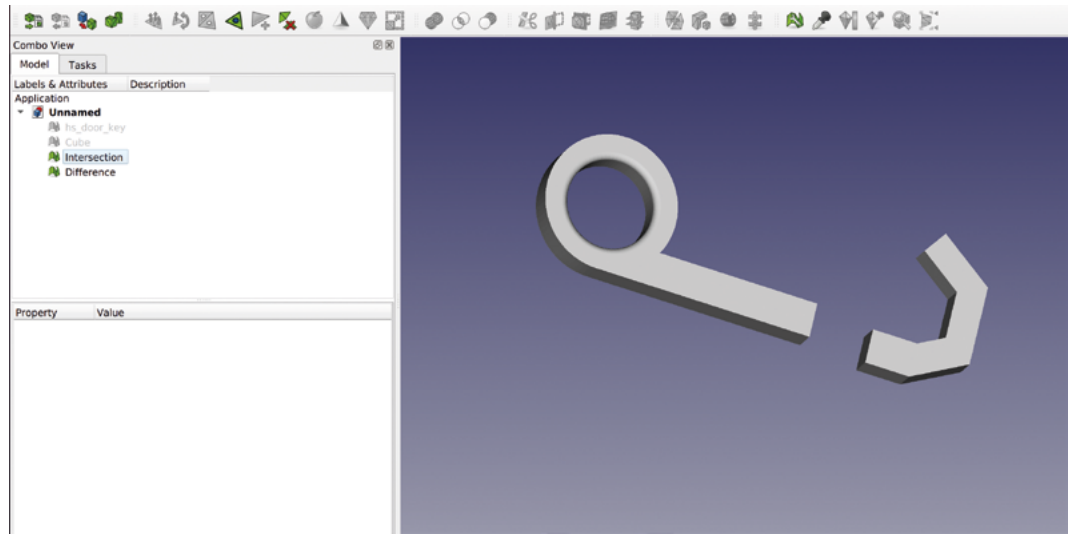
> **"** Mesh decimation must be used in moderation, as taken **too far, it's easy to completely destroy the object "**

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 we have a close view of a part of the internal chamfer on the large hole in our 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 our case called 'Union' in the file tree. Notice that in the dialog box, once selected, we get some details of the number of points, edges, and faces our 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 our puller part change, it should look cruder and the triangles should be more obvious (**Figure 10**).

If we check the details of our mesh object in the dialog box, we 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 our 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 we 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 our 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!

Figure 8 🡵
New parts created and ready to be put back together in our improved configuration

Figure 9 ◆
With the two parts of the puller rearranged, we combine them back into a single mesh using the Union Boolean operation ready for export
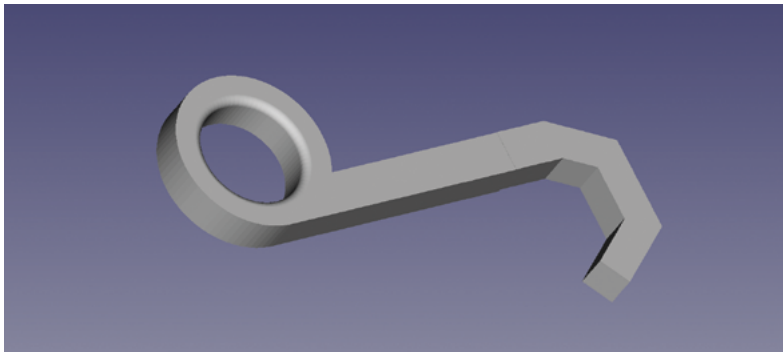
In a new project, move to the Part workbench and create two identical default cube solid objects. These default to 10 mm cubes, using Transform, move the second cube object up 10 mm in the Z axis and across 10 mm in the X axis. Finally, highlight both cubes in the file tree view and perform a Union operation to make the two cubes – one part called 'Fusion'. Moving to the Mesh workbench, with our fusion object highlighted, click the 'Tessalate shape' tool icon to create a mesh object. You can see the resulting mesh imported in **Figure 1**. 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, our 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 our cubes example and found the non-manifold edge issue, we 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 we move back to our part object and delete the fusion object so that we have our 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, we 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. □
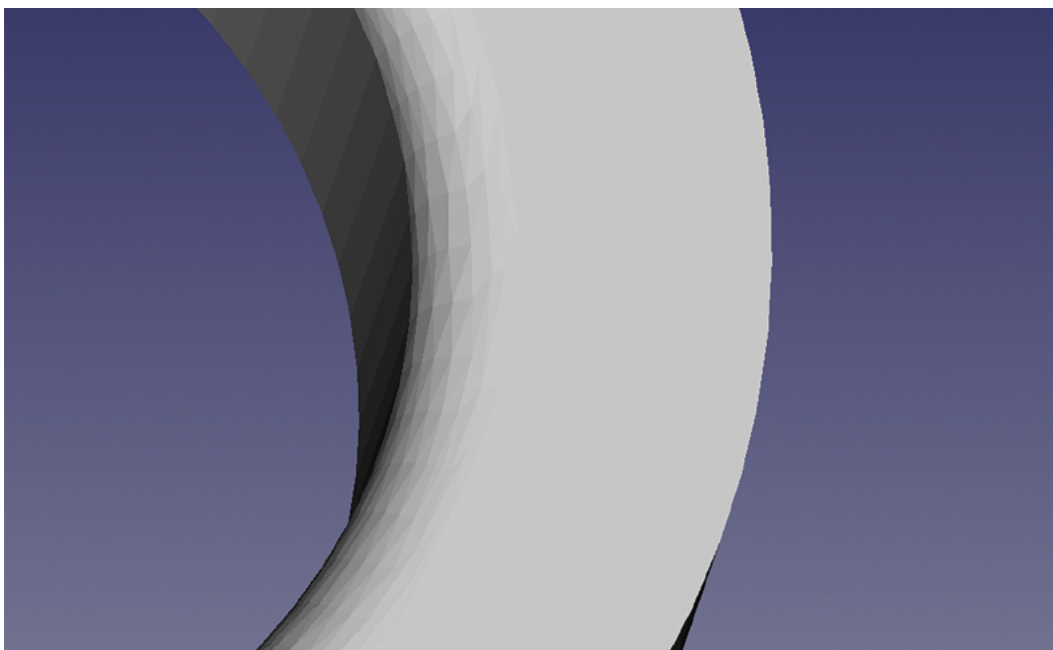
Figure 10 ◆
Using the 'Decimate a mesh' tool to reduce the resolution of a model can make a complex model easier to work with

FreeCAD: intermediate sketching

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

We have already 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 (on page 48). 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 (on the 'Task' tab in the combo view window) to pad the

Figure 1 ➜
Using the Polyline tool, in various modes, to trace accurately over the imported image object

rectangle sketch into a larger cuboid. Next, select any face on the cuboid in the preview window and click the 'Add 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 (**Figure 2**).

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
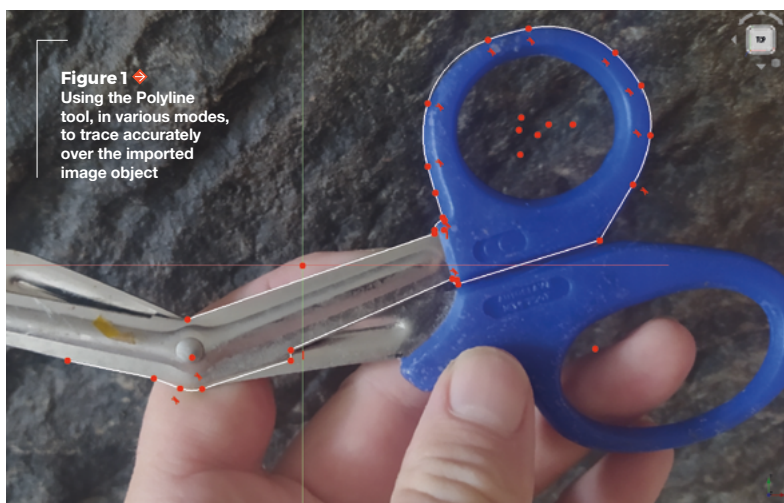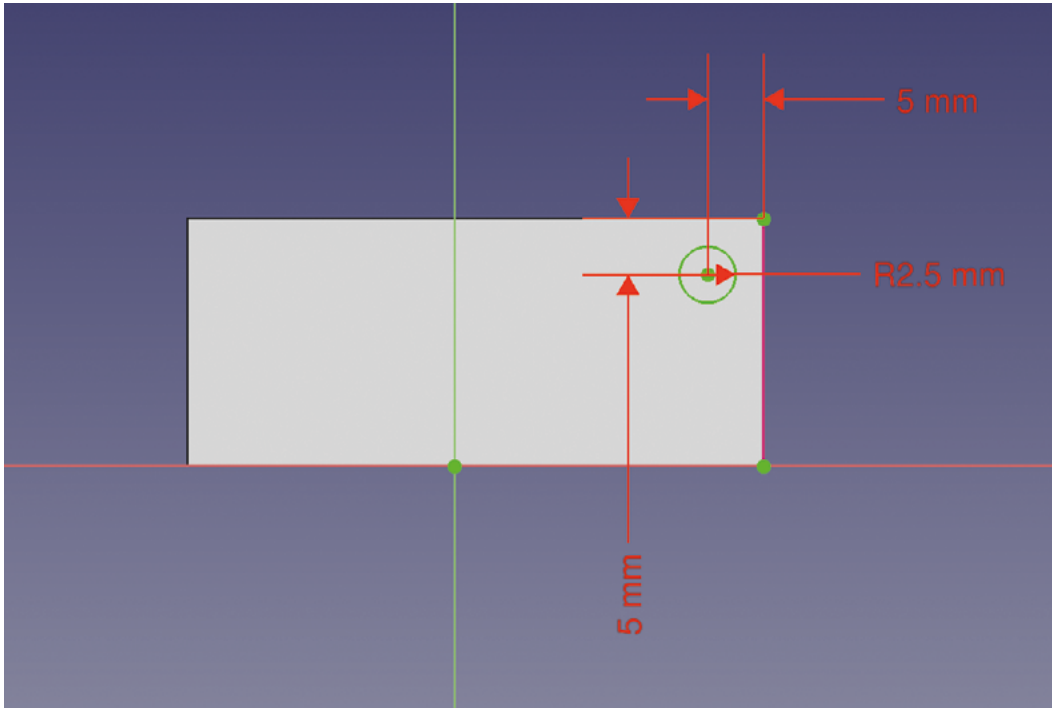
5 mm

R2.5 mm

5 mm

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.

In **Figure 3**, 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 (**Figure 4**).

> Often we may need a guide point in a position where there **is no underlying geometry that we can import**

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 →
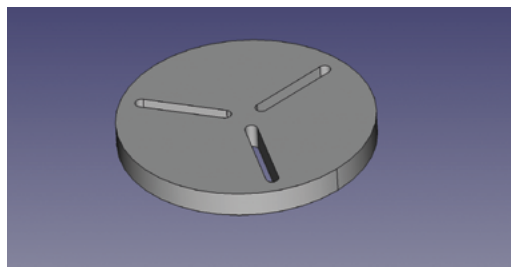

Figure 2
Using imported external geometry to create a guide point to constrain items to

Figure 3
This slotted plate uses construction mode geometry to aid the placement and constraining of the slots
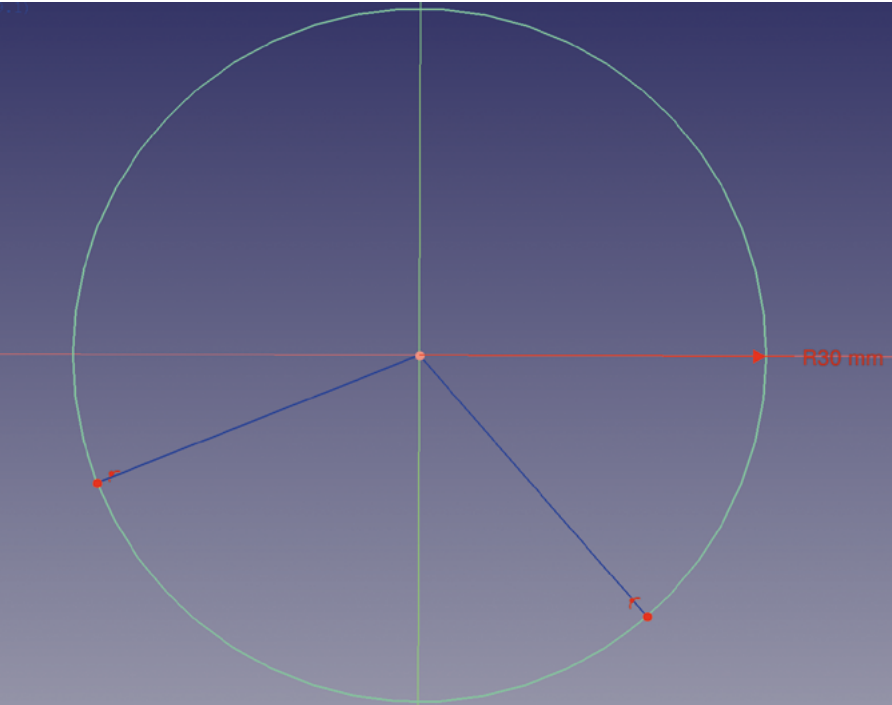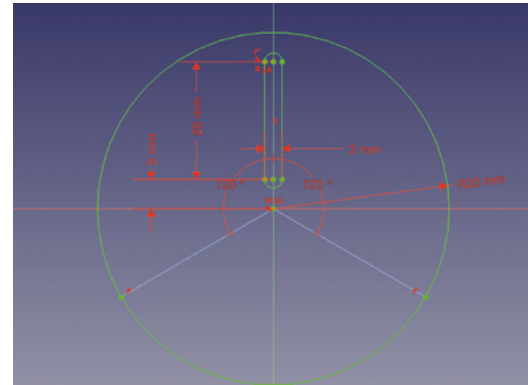
Figure 4
Setting up a pair of construction lines to be constrained to the correct angle to align our slots
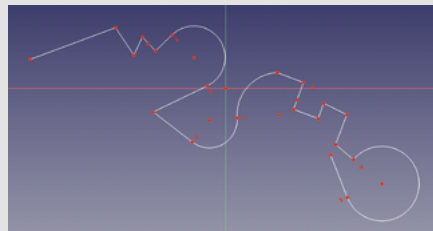
Figure 5
Our construction lines created and the first slot added and constrained on the Y axis line

'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!

we have both our construction lines correctly positioned (**Figure 5**).

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 (**Figure 5**).

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
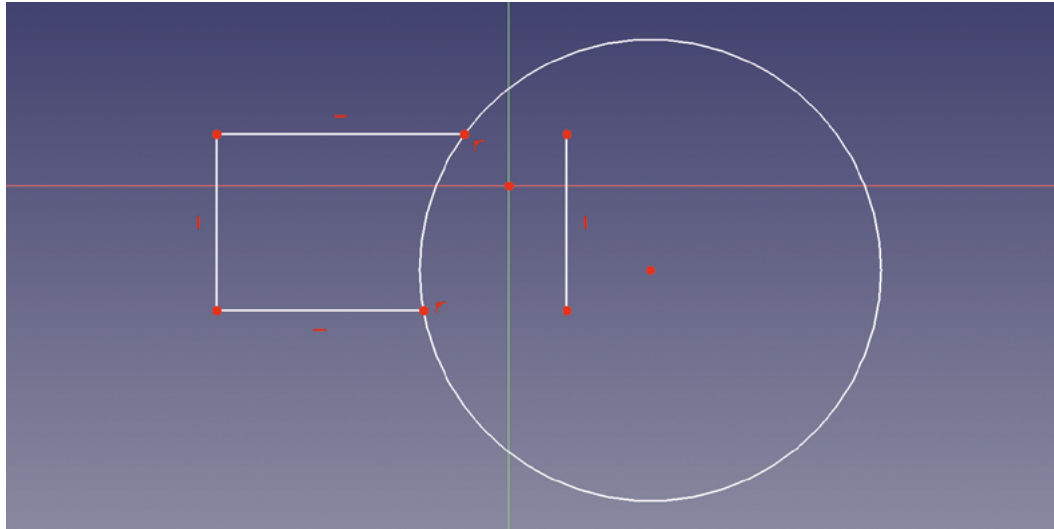
Figure 6 ◈
Using the 'Trim an edge' tool to remove intersecting lines

to be aligned with the construction line perfectly. We 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.

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,

> **The Trim tool is only of use when a line intersects** or is connected to another line

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. In **Figure 6**, 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 (**Figure 7**). 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. →

Figure 7
Trying to use the Trim tool to delete the final unconnected edge will result in an error. However, we can simply select the line and delete it

Figure 8
Setting up some lines that don't meet, for our example of using the 'Extend an edge' tool
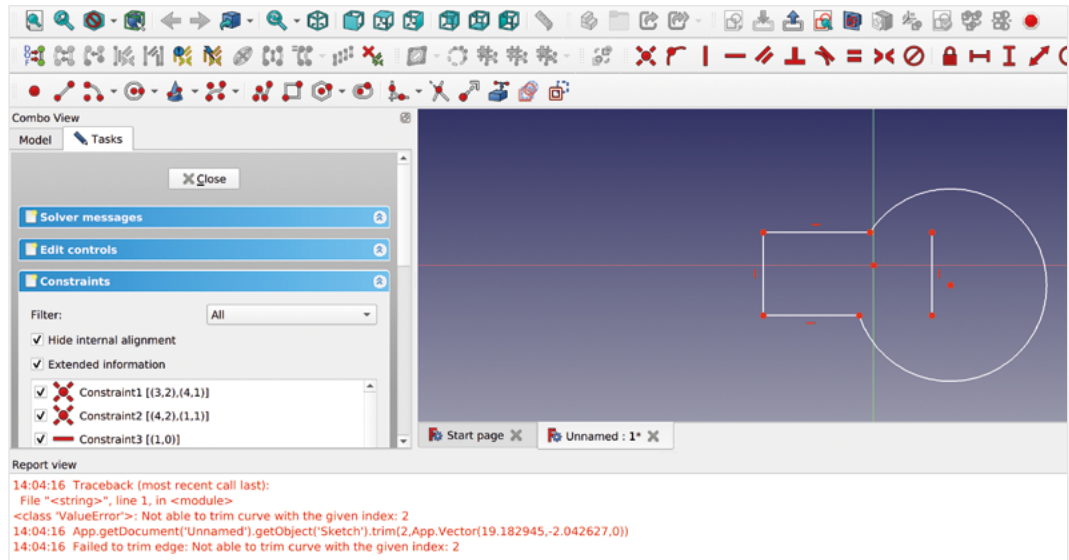
A similar useful tool is the 'Extend an edge with respect to the picked position' tool. In a new project and a new sketch, draw a rectangle and then draw some polylines near the rectangle that don't quite meet the rectangle lines, similar to **Figure 8**. Next, select the 'Extend an edge with respect 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 (**Figure 9**). This tool works not just on straight line geometry, but will also work to extend arcs and more.

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 (**Figure 10**).

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 (**Figure 1**).
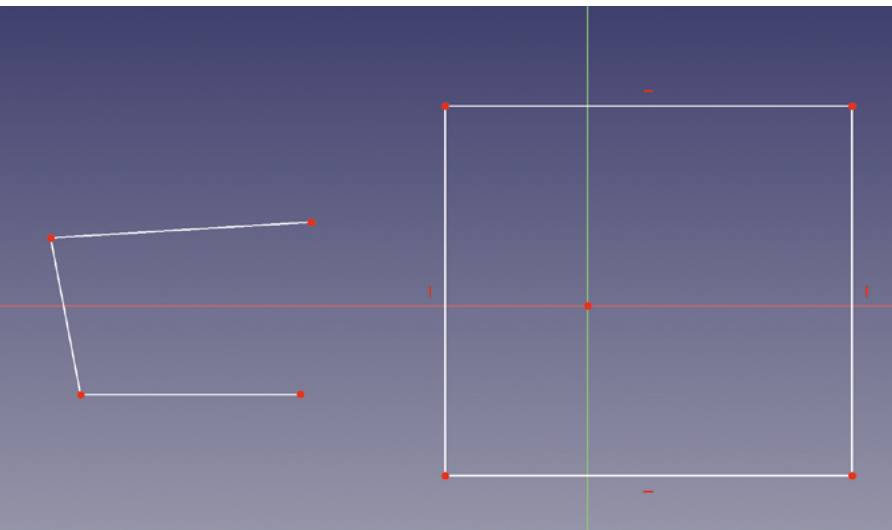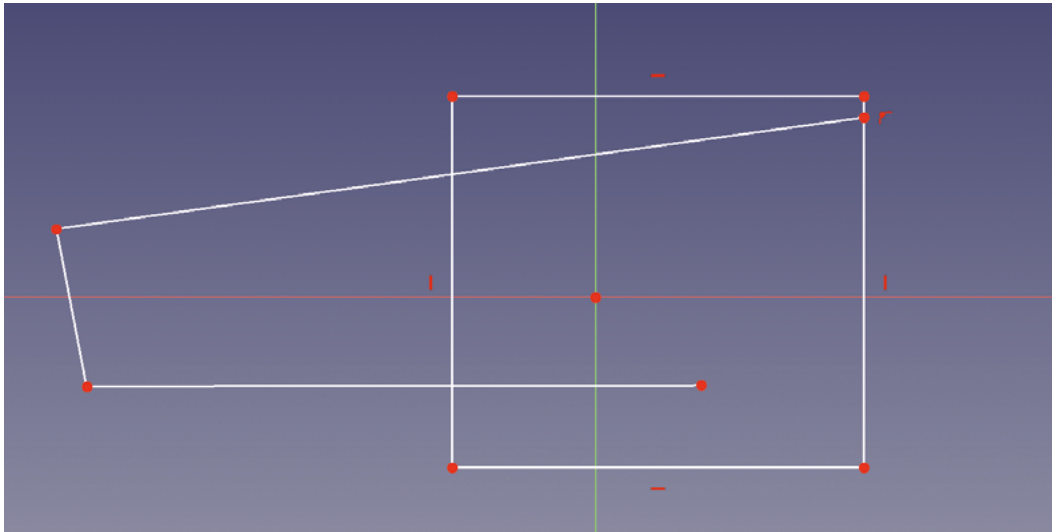
Tracing imageplane objects is an excellent approach, but one problem is that images, especially photographs, rarely scale correctly when imported. If you have followed along and imported an image and traced (or partially traced) parts of it, as a test, select a line and click the 'Fix a length of a line 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 in **Figure 1**, 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 (**Figure 11**).

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
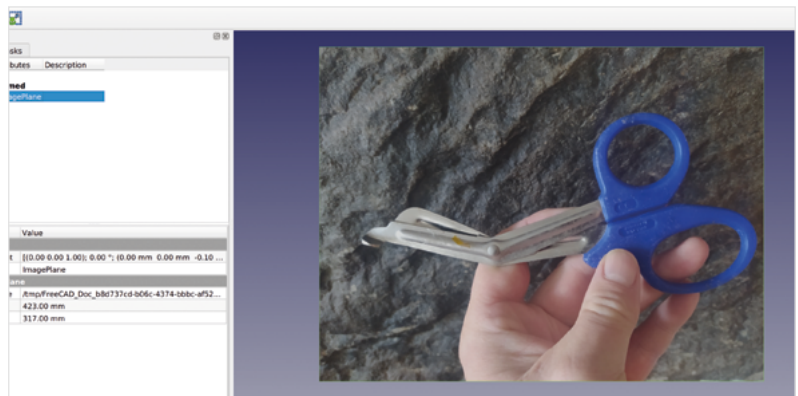
is parallel to the object means that when we trace the item, we aren't tracing a skewed version of it. Making sure a photograph is taken directly from above can help; or, if sizes allow, even better is to create a scan image of the object on a flatbed scanner. Finally, if you add a ruler or a cutting 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. ◻
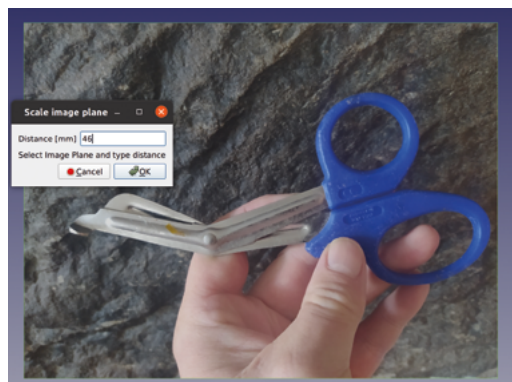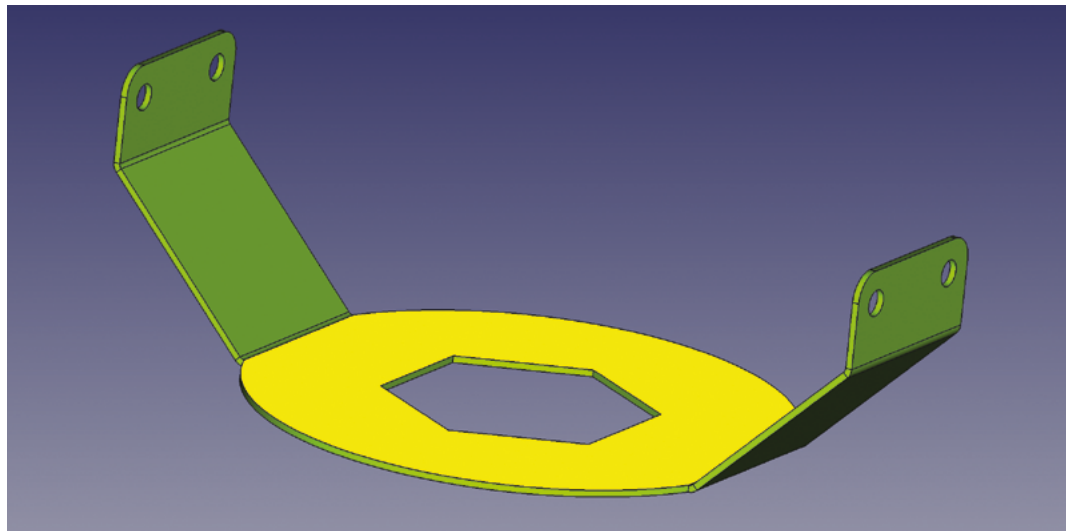
Sheet folding in FreeCAD

Let's look at an amazing workbench that makes working with sheet metals,
or other foldable sheet materials, simple to CAD

In this section, 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 'Sheet Metal'. 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 12 cm by 6 cm. 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, which should be the furthest to the left on the Sheet Metal tools toolbar.

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. 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
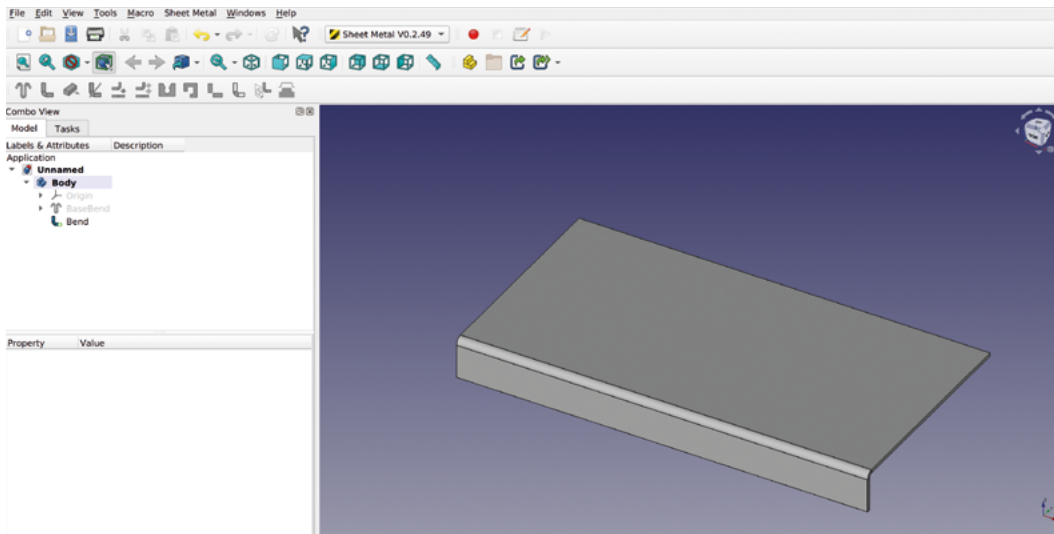
from the original BaseBend object in the Z axis. If we wanted this section to fold upwards, we can change the 'angle' parameter in the dialog. Setting this to -90 rotates the folded section to point upwards. Similarly, we can set the angle of this fold to any amount we choose. Moving down the parameter 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, in **Figure 3**, we have added a gap value of 10 mm 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.

> **Corner relief is a small cutaway that, similar to the earlier notches,** stops the metal distorting when folded

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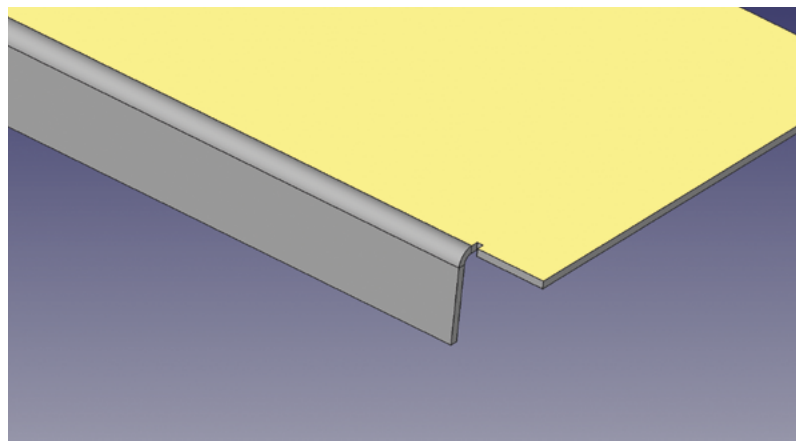 in the piece as it is folded, so it can be a good idea to add some corner relief. Corner relief is a small cutaway that, similar to the earlier notches, stops the metal distorting when folded. To add corner relief, we need to select the two edges which run into the corner on the BaseBend object. You can see these edges selected in **Figure 4**. →
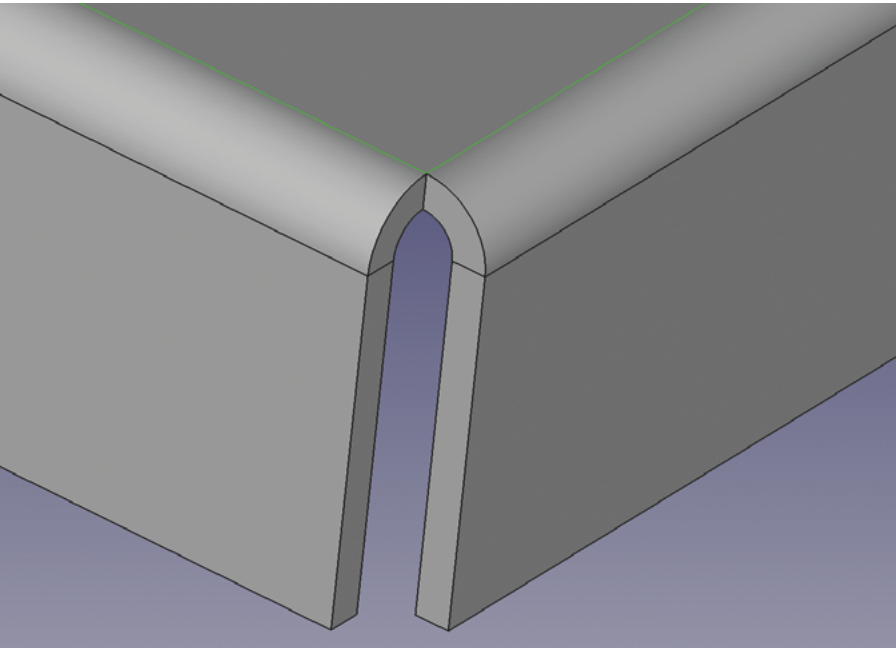
Figure 4 ⤢
To set up a corner
relief, you must select
the two edges that
run into a corner point
between folds

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 1 mm 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.
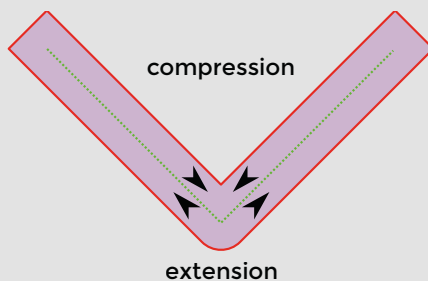
With the two lines selected, the 'Corner Relief to sheet metal corner' tool icon should become active (seventh from the left on the Sheet Metal workbench toolbar). Clicking the tool should create a corner relief object in the file tree, and also create a circular hole in the corner of the two folds (**Figure 5**). 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 (**Figure 6**). 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 new 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 1 mm thick, so if we change the offset amount to 1 mm, the new section is on the outside of the two tray walls. Or, if we input -1 mm, 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 1.1 mm 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
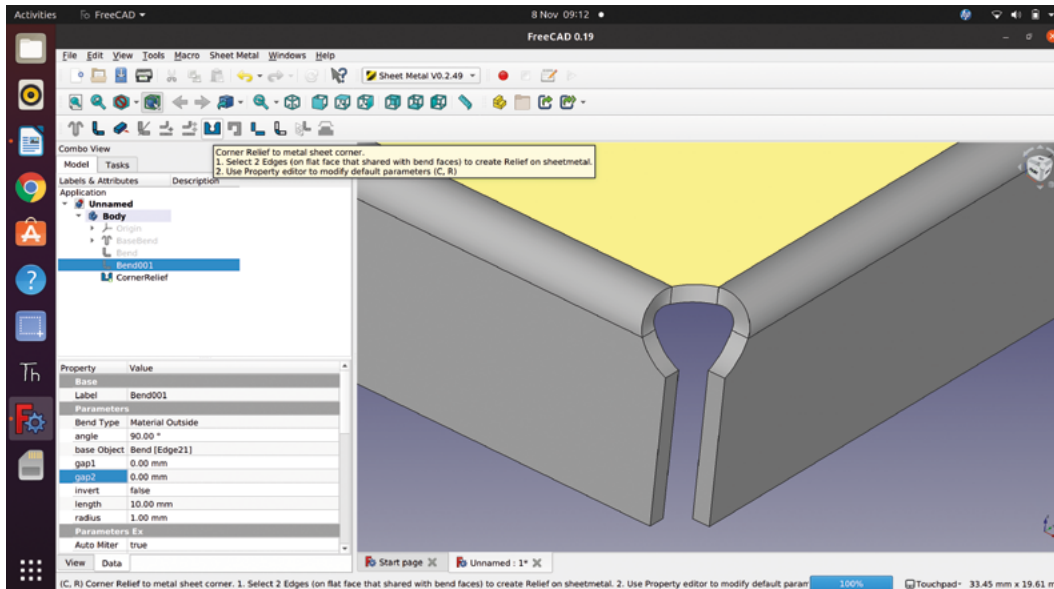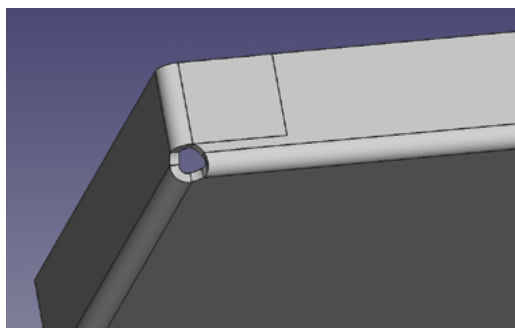
move to the Part Design workbench, select a suitable edge, and apply a chamfer or a fillet. A more useful example is to perhaps add sketches to our metal tray design. If we were to fabricate this tray, we might like to add holes in the design for bolts or rivets to fix the folded corners firmly. To add 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 'Add a sketch' button. We then drew a 2 mm radius circle, imagining we might use a 4 mm 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, (**Figure 7**).

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 1 mm thick sheet metal design when working in aluminium. →
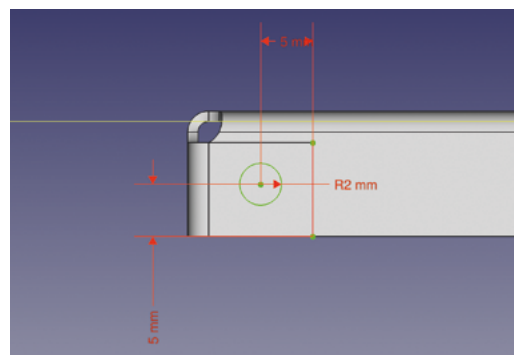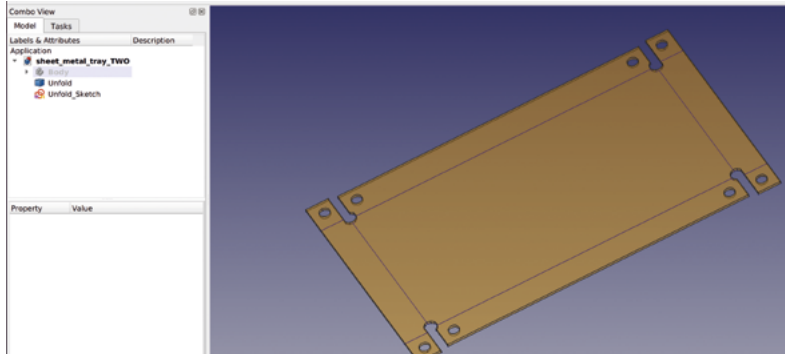
Having input 0.4 and ticked 'ANSI', we can then click the 'OK' button. Depending on your computer, it may take a few moments but, once processed, you should now see two new objects in the file tree view – 'unfold' and 'unfold sketch'. These two new items exist outside of the active body containing our 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 in the file tree means we can inspect these items more clearly (**Figure 8**). 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 created a new project, moved to the Part Design workbench, and created a sketch in the XY plane. In the sketch, we drew the object in **Figure 9**. You don't have to faithfully recreate this sketch, but you can create something similar to play with these techniques.

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 15 mm 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.
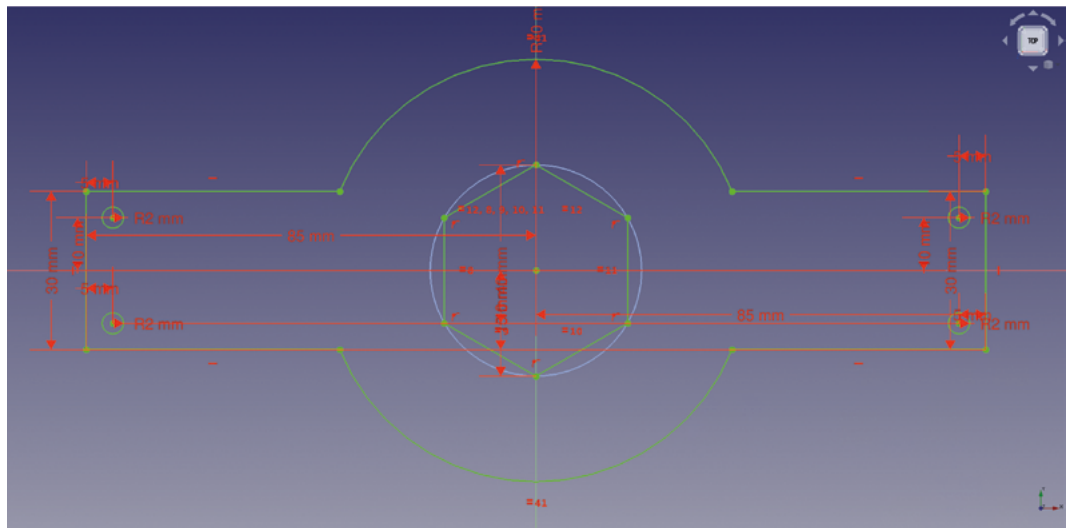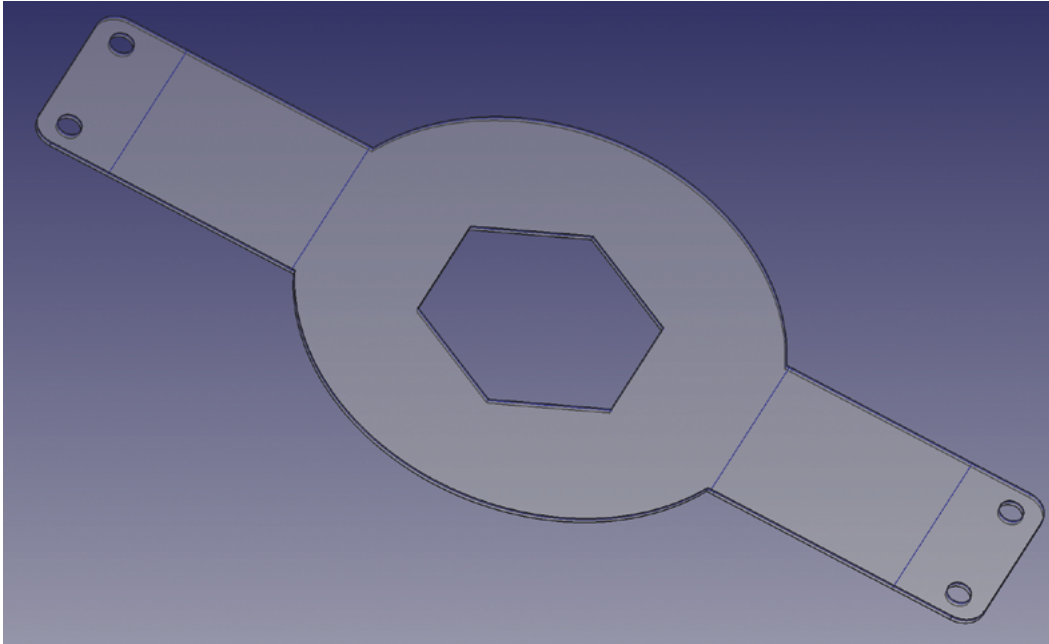
Figure 11
Our winged object, flattened, has retained all the additional features and folding points on the sketch and 3D object

Figure 10
Our first fold using a sketch line as a fold position

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 (**Figure 10**).

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
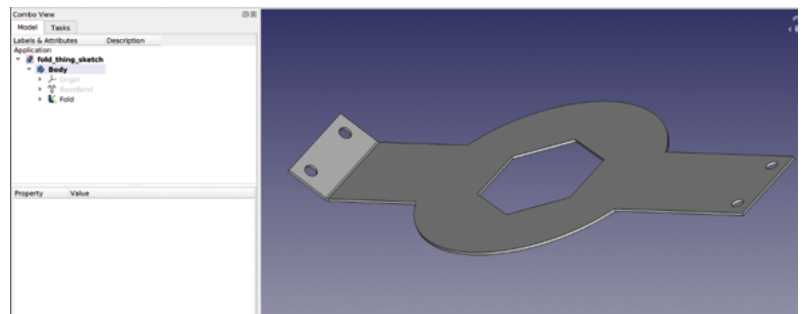
area of the circle rather than inside it. We repeated the process above for the other side of the object and, as a finishing touch, we added a chamfer to the upper wing corners using the chamfer tool in the Part Design workbench. You can see the completed item in **Figure 1**. One of the reasons we added the chamfers was to prove that even when you add geometry on other workbenches, the 'Flatten folded sheet metal object' will still work correctly and carries through all geometry on the designed object. In **Figure 11**, you can see that we have flattened our designed part, and everything is still present and correct.

It's been a pleasure to explore this workbench and, in the most part, we have found it very straightforward in use. We particularly like that the workbench can accomplish so much with so few tools, and this makes for a pretty straightforward experience. □

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.
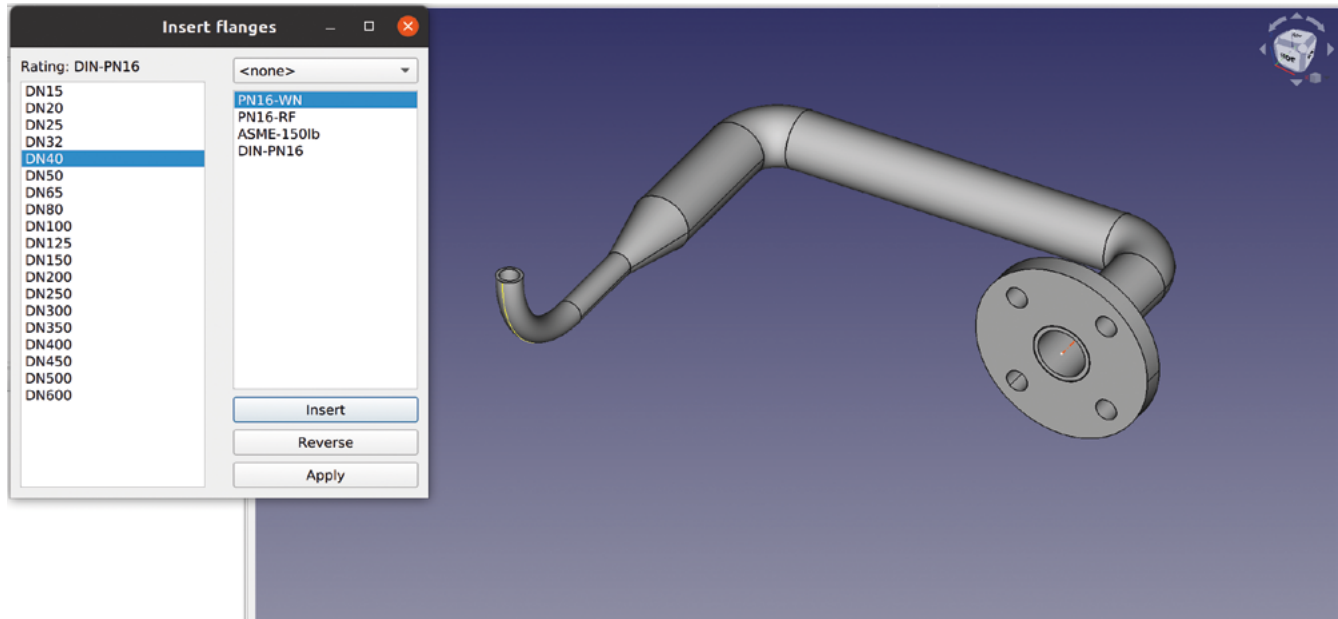
FreeCAD: frames and pipes

Let's look at a new workbench: the 'Dodo' workbench allows us to create complex frames and pipework from a collection of different profiles

While there's no denying that you could design sheet metal objects without using the Sheet Metal Workbench, it makes a sheet metal design much quicker to generate and work with. In this section we're looking at the 'Dodo' workbench, which treats working with framed objects and designs or pipework in a similar fashion.

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 in **Figure 2**. 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' and select 'RH', and then select the 'RH40×40×5' by highlighting it in the list below. 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. You can, of course, in the file tree view, make the cube object invisible to get a better view of our frame (**Figure 3**).

You'll notice that our frame has all the required sections but that they all overlap at the corners. We
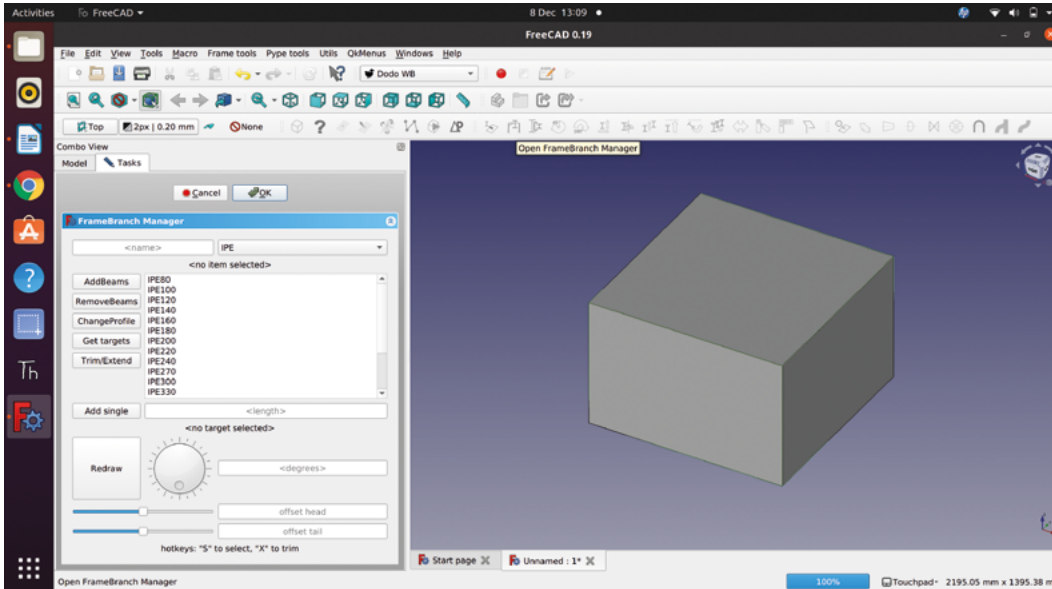
need to extend or trim these frame parts to make a frame that could be fabricated in real life. With the Open FrameBranch Manager still open, we can use the 'Get Targets' and 'Trim/Extend' buttons to help us do this quickly. First, select an inside face of one of the frame sections and then, holding 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 (**Figure 3**). 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 Sketcher workbench and create a sketch of a rectangle with an extra line spanning across the rectangle inside – our rectangle was 100 cm × 50 cm.

> **We need to extend or trim these frame parts to make a frame that could be fabricated in real life**
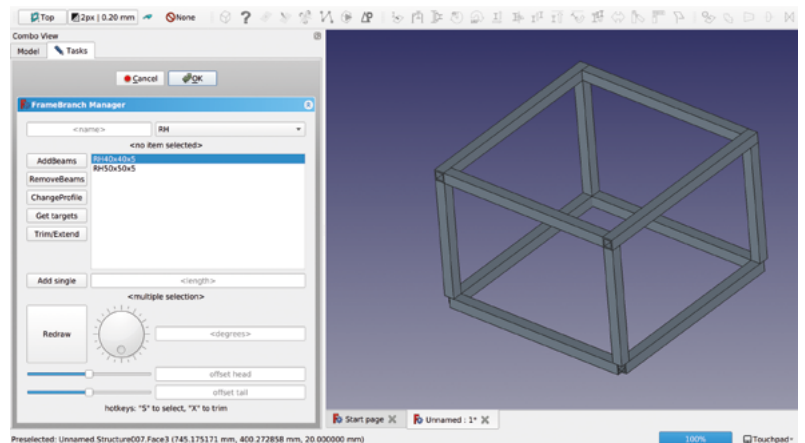
Moving to the Dodo workbench again, click to launch the Open FrameBranch Manager. Let's select a different profile – we went with 'IPE80', which is a common steel H-section used in buildings and structure fabrication. Again, click any edge of the sketch in the preview window and then click 'OK'. You should now see a frame appear made from the H-section. You can, of course, use any face of any of the IPE80 parts to set targets again to trim or extend other sections to. Another useful tool in the Open FrameBranch Manager dialog is the Rotation tool. Select any face of a section of frame and experiment by either moving the dial, or by typing an angle into the input box. →
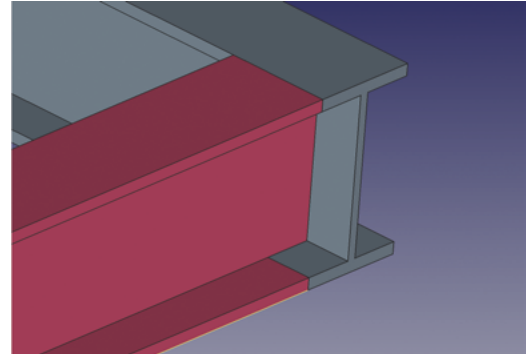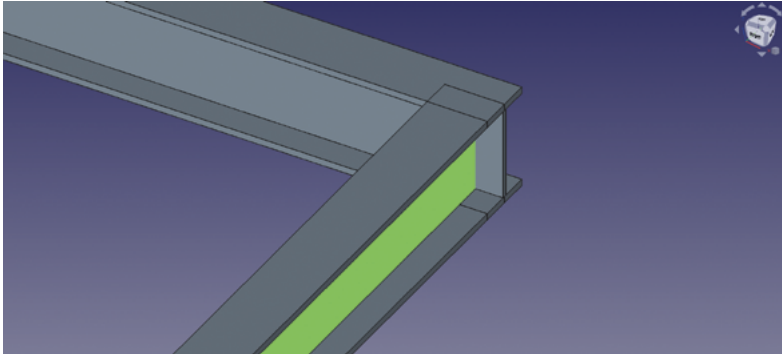
93

Figure 4 ◆
When H-frame sections overlap, they often create joints that aren't correct for the real world

Figure 5 ◢
Applying a workaround solution with the Part workbench 'Slice Apart' tool gives us a more correct H-section interface

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.
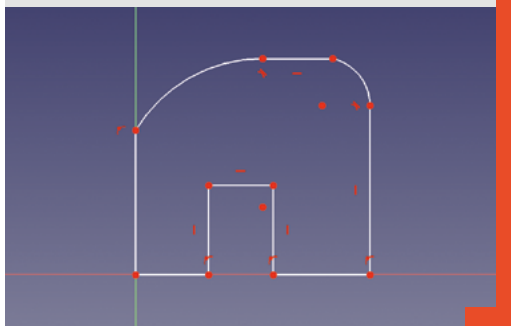
As you move, trim, and extend H-sections, you will probably start to see the limitation of the Dodo workbench. If you position a corner of our frame, similar to **Figure 4**, you will see that although the H-sections are in the correct position, the overlapping portion of the sections aren't trimmed. Of course, this would be impossible to realise in a real-world build. We've created a workaround that works with simple H-section overlaps; this can lead to quite a complex file tree, but it's still useful to look at here. Arrange a corner of the frame so that it looks like **Figure 4**, and then select the two H-section objects that form this corner in the file tree inside the 'Travatura' group – select these in an order so that the part you want to remove the overlap from is selected last. We opted to make the internal part the object we wanted to trim off the overlapping parts, which are two small sections on the top and bottom of the H-section. Next, move to the Part workbench and click the 'Slice Apart' tool, which is in a drop-down menu off a tool icon for the less common Boolean operations. You should now have a new group in the file tree called something like 'Exploded Slice' – inside this group exists a copy of our target intersecting part. In the file tree view, toggle the visibility of the original target part in the Travatura group to invisible, then, in turn, toggle the parts inside the Exploded Slice group to work out which parts you need to remove or make invisible. You should end up with an H-section part within the Exploded Slice group that fits the corner assembly perfectly as if ready for welding in a real-world build (**Figure 5**).

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 Sketcher workbench and create a new sketch in the XZ plane. Create a sketch similar to **Figure 6**. In this example, it's probably
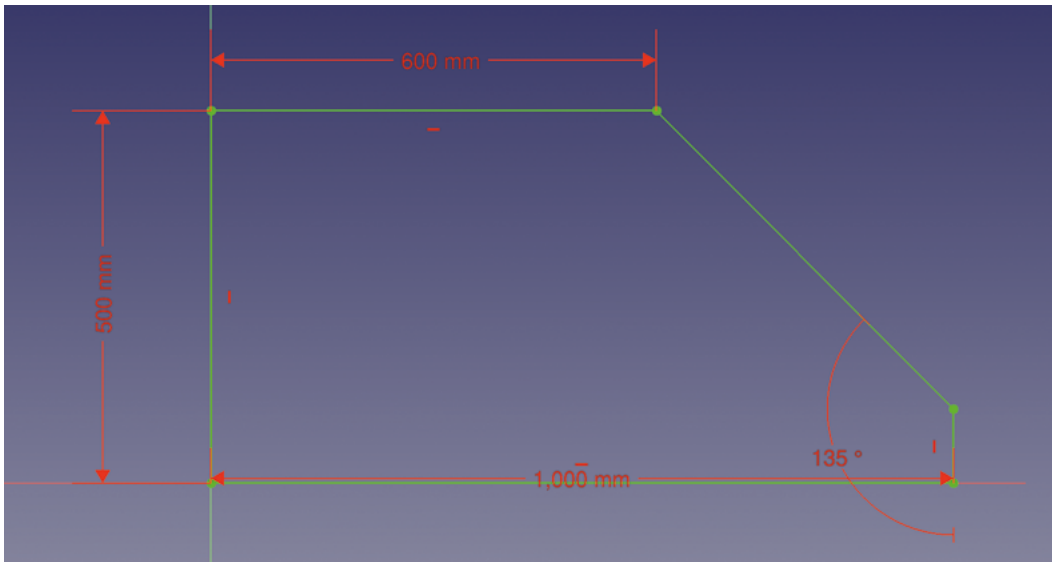
useful to fully constrain your sketch so that we know accurately where frame elements will be placed. Next, move briefly to the Part workbench and extrude the sketch into a large frameable object.

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. You can see this in **Figure 7**, where the front angled section hasn't been extended and the rearmost section has.

REMOVING OVERLAPS

Continuing to Trim/Extend, we can get the horizontal and vertical frame members to adjoin correctly, as seen in **Figure 7**. 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. You can see our sketch in **Figure 8**.

Next, 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. 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 →
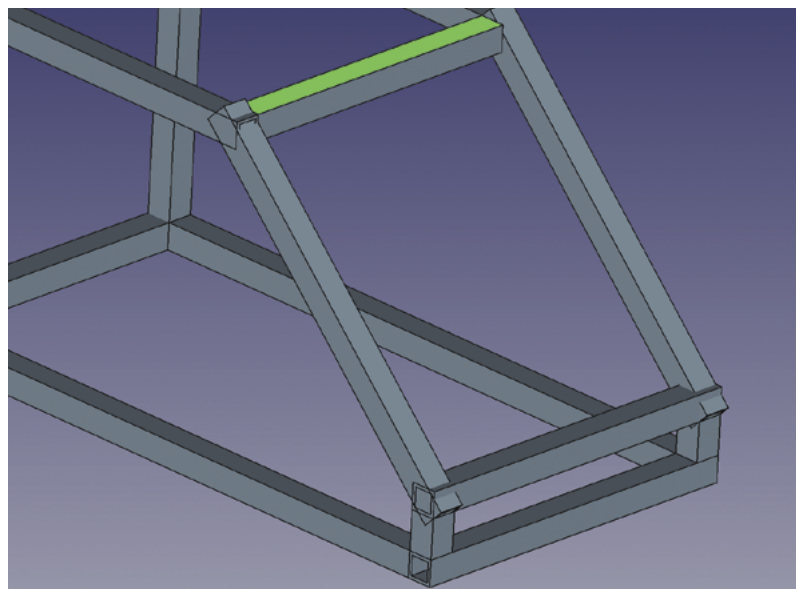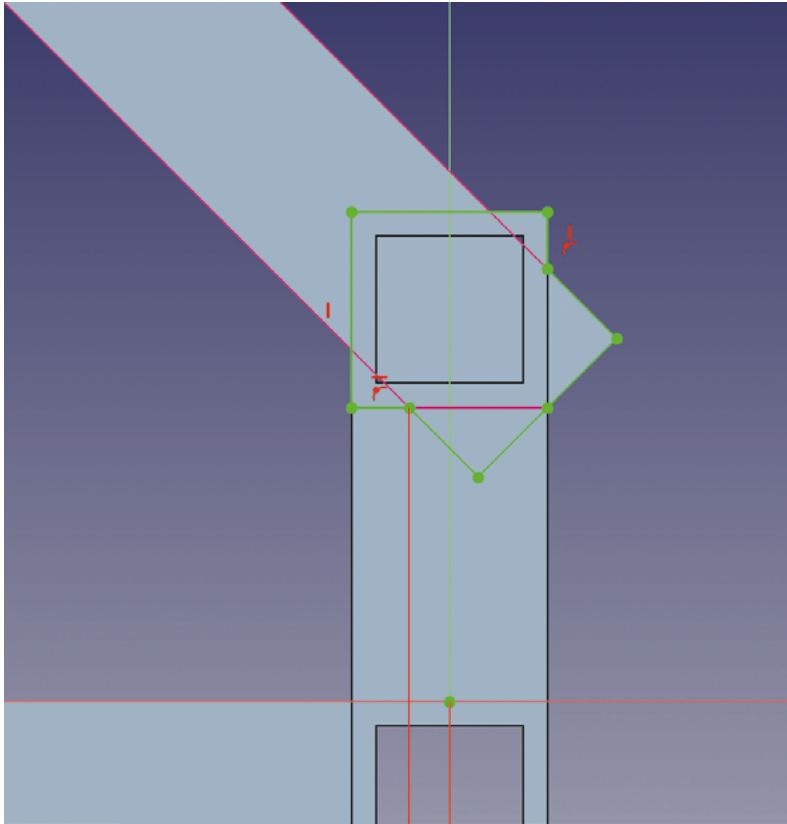
> **"** You can add pipe objects
> and other peripherals
> **by simply attaching them
> to an existing pipe "**

Figure 8 ◈
A sketch set up for
extrusion to create
an object that will
cut away and trim
the frame members
correctly

Figure 9 ◈
Using our workaround,
we now have the ends
of the angled frame
members correctly
trimmed

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. In **Figure 9**, 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 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
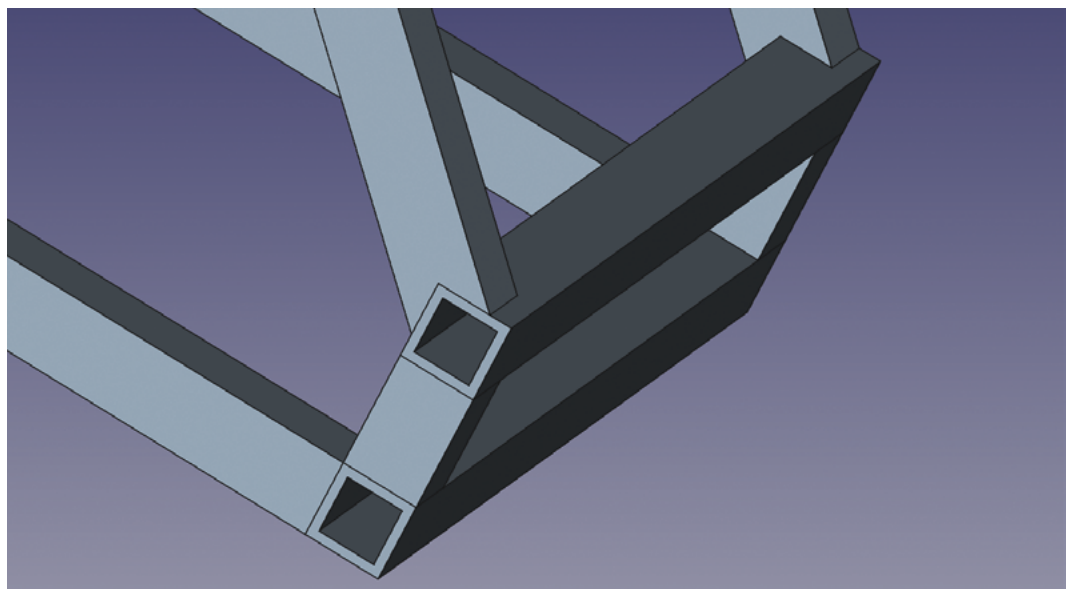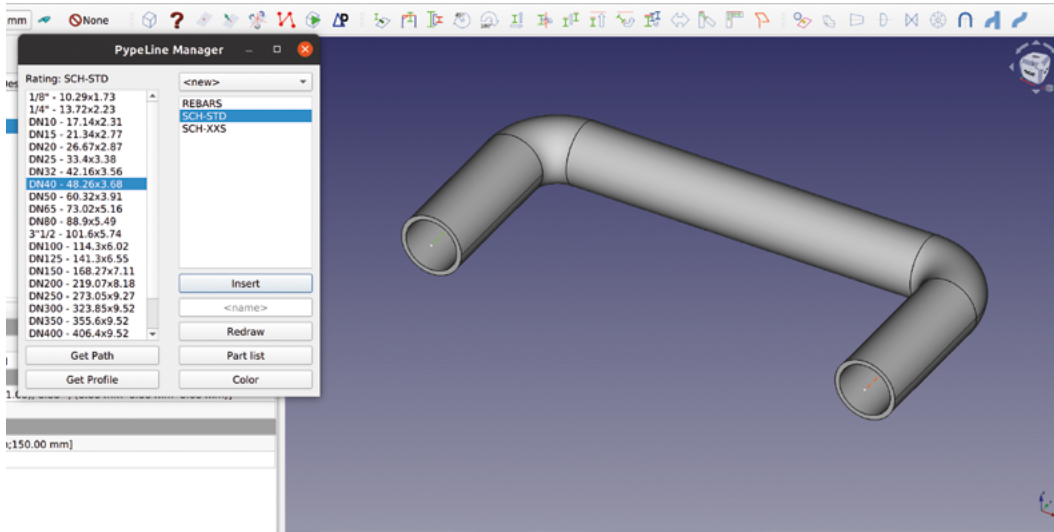
automatically create a pipe that follows the complete sketch path with the 90-degree curved sections added (**Figure 10**).

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. In **Figure 11**, 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. You can see the reduction, manually added pipe, and the angled curved elbow we have just added in **Figure 1**.

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 (**Figure 1**).

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 on page 42. ▫

Figure 10 ◩
Creating pipework that follows a sketch is simple using the PypeLine Manager tool

Figure 11 ◈
Adding a reduction part to our simple pipe example
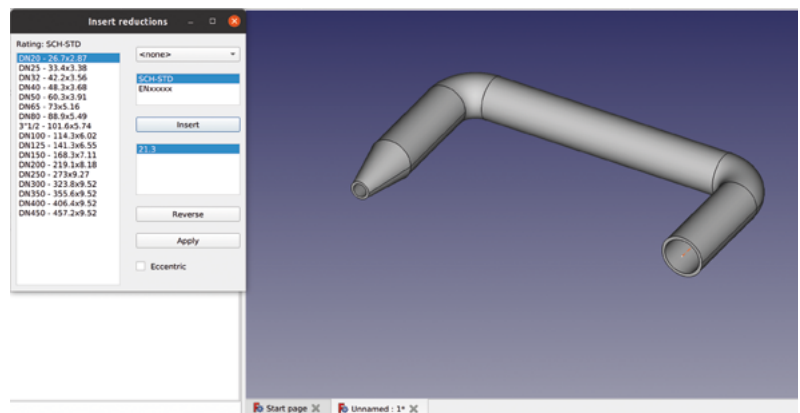
FreeCAD, summing up!

We look at where you may like to explore next as part of your ongoing FreeCAD journey!

n this final part of our guide to FreeCAD, 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 guide 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. In **Figure 1**, 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 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 (**Figure 2)**. 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.
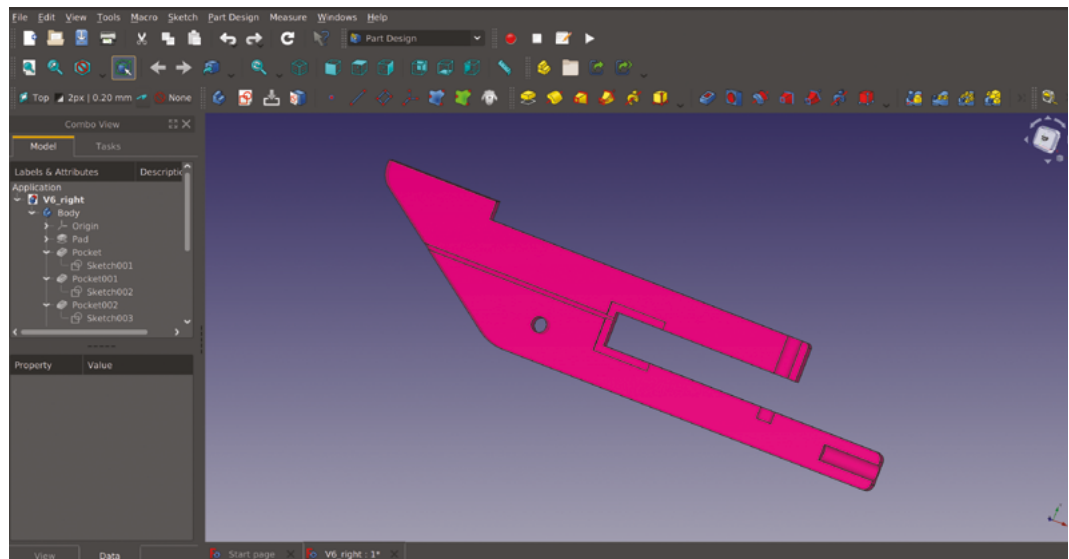

Figure 1
Changing the stylesheet options in FreeCAD can customise the colours and also change the tool icon sizes
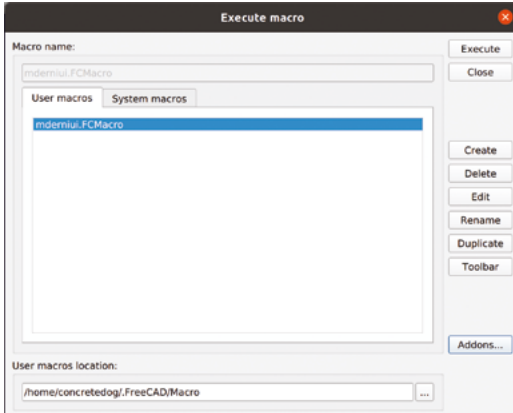
In turn, click Macros and, in the Execute macro window, click Create (**Figure 3**). Paste in the macro script copied from the Addon manager, and then close the tab and click Save and give it a name when prompted. Return to the Execute macro window, highlight the macro you created, and click Execute to run the macro. Then, 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

> **Essentially, once a macro is recording,** it builds a script of all the actions you perform until you stop the macro recording

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
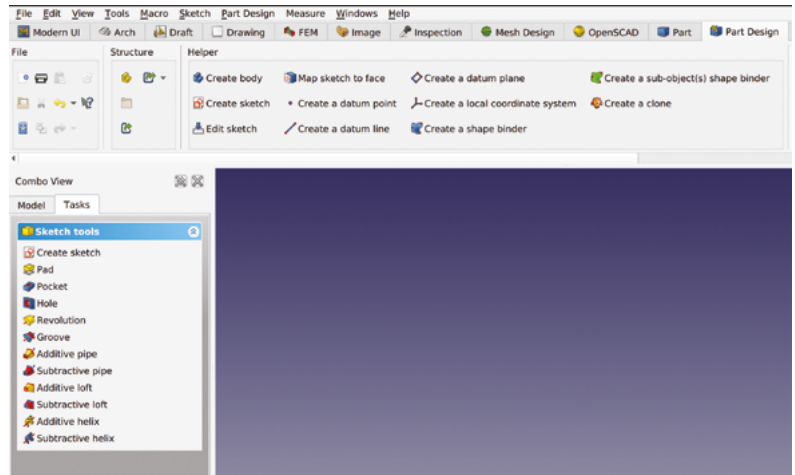
require a little editing and tweaking. It's been described to us more than once that macros are the gateway to writing your own workbench!

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 →

Figure 2 ◆
Available via the Addon manager, Modern UI changes the design layout of FreeCAD and uses tabs for each installed workbench

ROTATING

Sometimes you might want to make your 3D model move, and, as ever in FreeCAD, there are numerous ways to achieve this. At first glance, you might jump to installing the Animation workbench from the Addon manager, but currently this only supports older Python 2-based versions of FreeCAD. It's worth keeping an eye out for if it gets any updates in the future. A current popular approach to animating assemblies is to use the Assembly 4 workbench. We haven't looked at this in the series, but we did look at an assembly workbench, A2plus, in issue 43. Whilst you are exploring and learning these approaches, a super-simple way to create a rotating movement of your part or project is by using the turntable tool. Located at Tools > View Turntable, when you launch the turntable dialog, you'll see a couple of simple parameters: you can adjust the angle of the view of the part and you can set the speed of rotation. You can also set the view to full screen and also set a timer for the length of time the part rotates for. Click the play button and your rotating part springs into life. You can use the turntable function in combination with a screen recorder application to make rotating video clips.
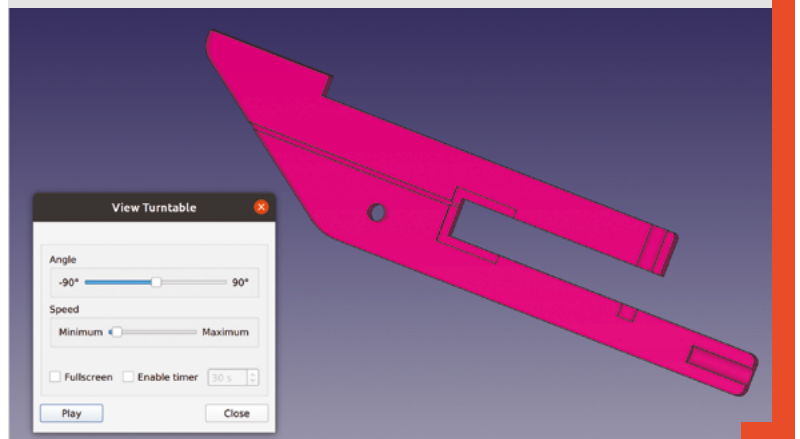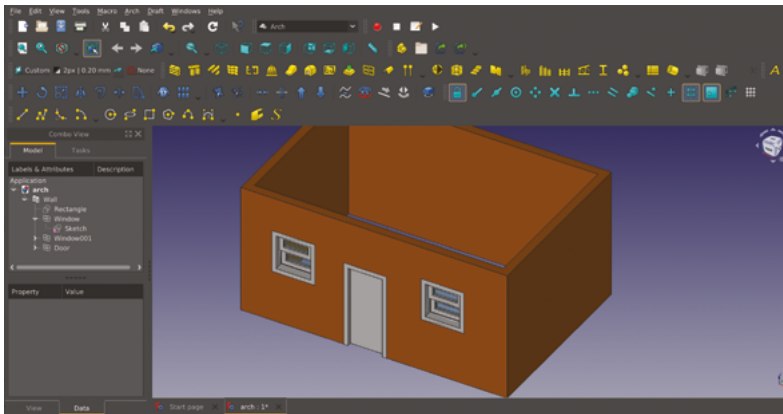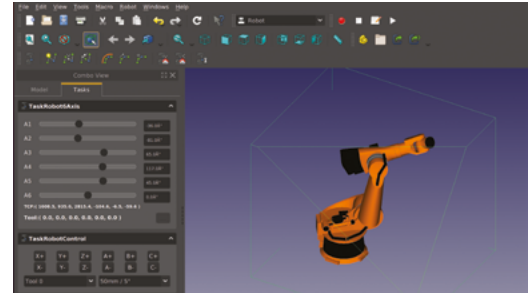
part of this guide, but we wanted to highlight a few examples that you might like to go on to explore.

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. In **Figure 4**, we have 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. Don't be fooled by our first attempt, though: 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 **hsmag.cc/RobotTutorial**, you can import an industrial robot arm. In **Figure 5**, 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 outside of this FreeCAD tutorial series way back in HackSpace magazine issue 25. 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 (**Figure 6**).

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

CROWD IN

One of the many ways in which people can contribute to the FreeCAD community is by helping translate FreeCAD so it can be used in many different languages. This is a large task but is being well achieved due to the use of 'Crowdin'. The Crowdin website really simplifies crowd-sourced translation. You simply set up an account, click a language you want to help with, and you are presented with every part of FreeCAD text split into little areas. Click an area and then try and translate a few terms or sentences. You can also view and comment on other people's translation attempts which means, with enough people, the translation becomes more and more accurate. This peer reviewing aspect of the Crowdin platform also means that your translation doesn't have to be perfect – often there is no perfect translation and, as such, all attempts help inform the best option. Once a language is translated, the FreeCAD developers can then pull that language into the platform for an upcoming release. Check out the FreeCAD Crowdin page here: **crowdin.com/project/freecad**.
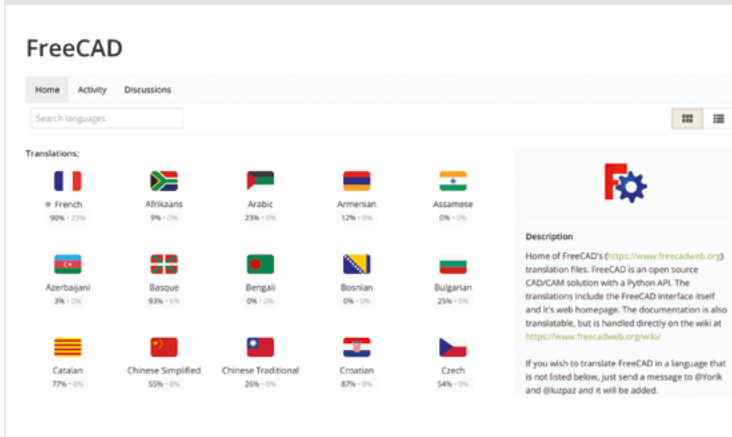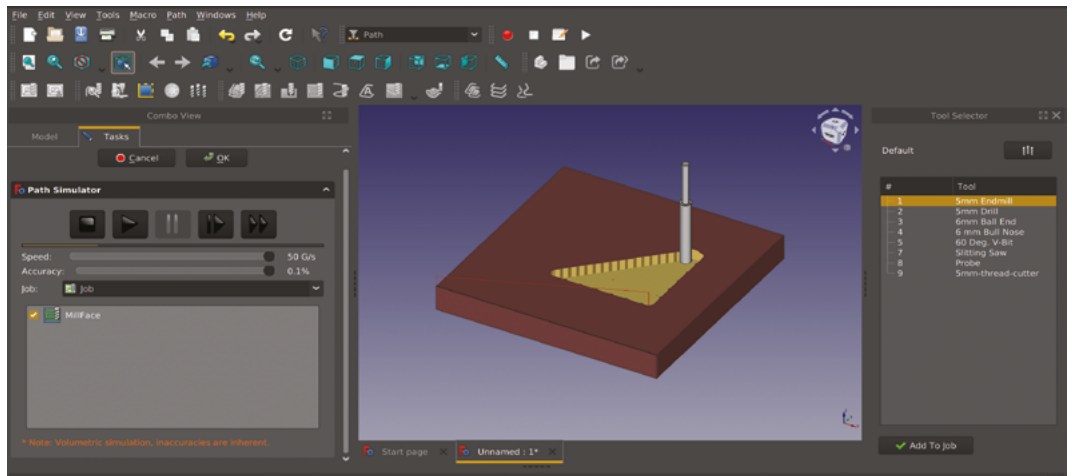
Figure 6 ◈
The Path workbench is a powerful CAM tool enabling toolpaths and G-codes to be created and simulated for CNC machines

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 (**Figure 7**).

When we looked at the Mesh workbench on page 74, 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 (**Figure 8**). 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
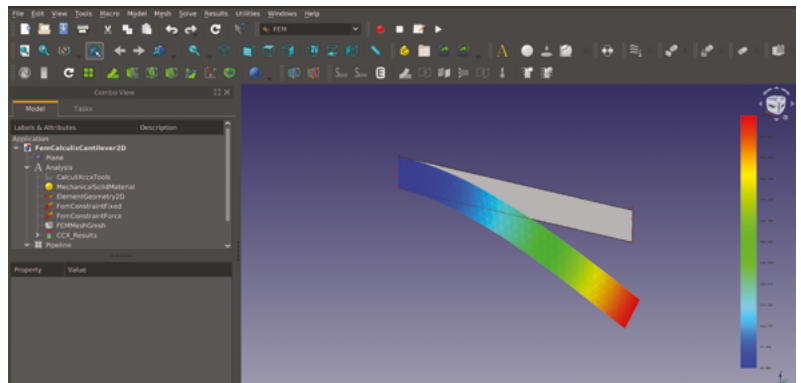

Figure 7 ◈
One of the included FEM workbench examples. This example analyses the deflection of a beam under load

> *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**

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 and 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 (**Figure 9**).

Are you a developer? FreeCAD is always interested in attracting more developers to contribute to the software. The majority of the FreeCAD →

QUICK TIP

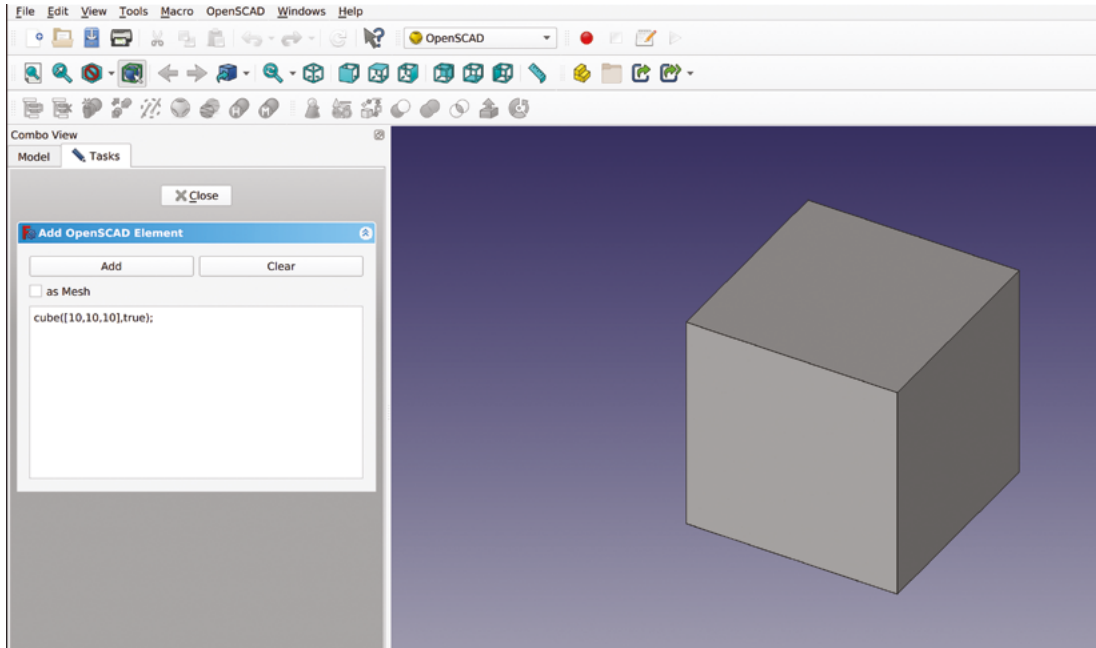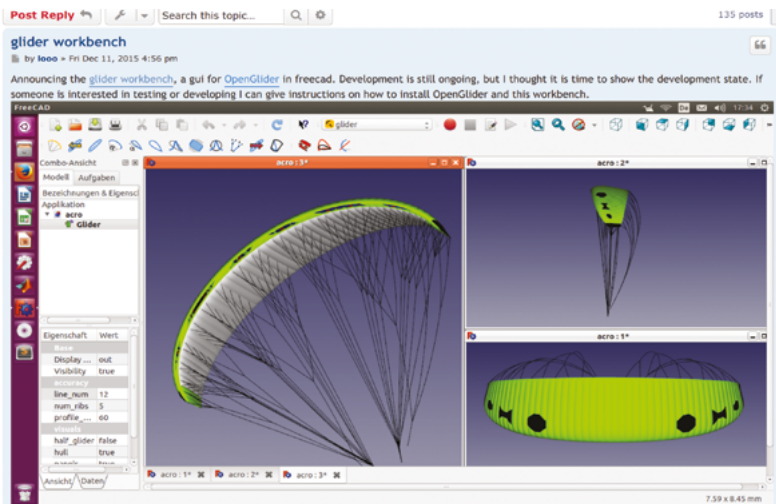Remember that FreeCAD is made and maintained completely by volunteers. Be kind!

Figure 8 ◈
The OpenSCAD
workbench offers all
the functionality of the
excellent OpenSCAD
environment built
into FreeCAD

Figure 9 ◈
The Glider workbench
is not under active
development, but
is still is being used
by a small number
of users to produce
amazing projects

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 (**Figure 10**). 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

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.

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 guide, 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,

Version   0.19
Revision number   24291 (Git)
Release date   2021/04/15 09:17:08
Operating system   Ubuntu 20.04.3 LTS
Word size   64-bit
Branch   (HEAD detached at 0.19.2)
Hash   7b5e18a0759de778b74d3a5c17eba9cb815035ac

Figure 10
The 'About FreeCAD' tab allows you to copy details of your machine, FreeCAD version, and operating system

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 guide 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 we've had preparing this guide 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 the tutorials that comprise this guide for HackSpace magazine, and we hope that you have enjoyed it too. 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. ◻

> **YouTube is absolutely crammed with tutorial videos,** and pretty much any area of FreeCAD is covered

Thank you for reading FreeCAD For Makers

FreeCAD is a free-to-use, open-source design program that's packed full of features to make your next project faster, more accurate, and more fun to build.

We've brought together the best FreeCAD projects, previously published in HackSpace magazine, to help you get the most out of this immensely useful software.

Read HackSpace magazine, the magazine for the modern maker

Buy in paper format or download the PDF for free: **hsmag.cc**

Get a Raspberry Pi Pico W with a print subscription: **hsmag.cc/subscribe**

Buy magazines and books from Raspberry Pi Press: **store.rpipress.cc**

Raspberry Pi PRESS

raspberrypi.com