MAKE | BUILD | HACK | CREATE

HackSpace

TECHNOLOGY IN YOUR HANDS

hsmag.cc | November 2022 | Issue #60

BUILD THE

INTERNET OF THINGS

Homemade projects to connect your life

Issue #60 £6
Nov. 2022

PINHOLE CAMERA

Inside the DSLR of the olden days

SHADOW CLOCK

Master time with a Raspberry Pi Pico

FLAT-PACK ROCKETRY

Build flying machines the IKEA way

TILES PLANTS KEYBOARDS LEDS

HackSpace
TECHNOLOGY IN YOUR HANDS

FORGE

HACK | MAKE | BUILD | CREATE

Improve your skills, learn something new, or just have fun tinkering – we hope you enjoy these hand-picked projects

PG 72
PHA FILAMENT
Are compostable prints a reality?

PG 74
BUILD A ROBOT
Interact with the world around you

PG 78

PICO W TAKEOVER
Control your RP2040 from a web browser

PG 82

FLAT-PACK ROCKET
A flying machine you can post

PG 68
SCHOOL OF MAKING
Start your journey to craftsmanship with these essential skills

68 **Slicing for 3D printing**

PG 88
PLANT STAND
Welcome to a steampunk future

PG 94

PINHOLE CAMERA
Low-tech photography

Getting started with slicing for 3D printing

Turn your digital designs into instructions for your printer

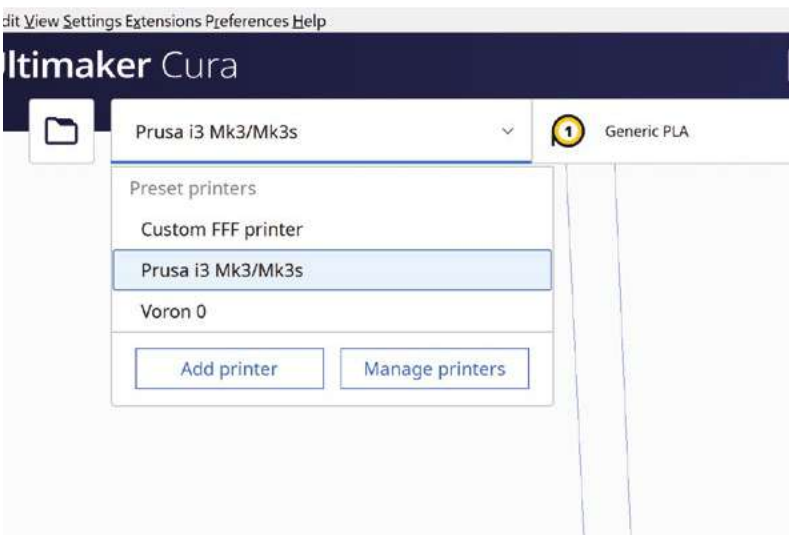
Ben Everard

@ben_everard

Ben's house is slowly being taken over by 3D printers. He plans to solve this by printing an extension, once he gets enough printers.

Most 3D printers (like most plotters, laser cutters, and CNC mills) understand a language called G-code. This is a text-based language that describes what they should do. Each line relates to a command, such as a movement or the setting of a particular parameter. Slicing is the process of taking a 3D design and converting it into a series of G-code instructions.

It's called slicing because, usually, 3D printers print a series of layers (aka slices), one on top of the other. There's nothing inherent about the hardware of 3D printers that requires them to print like this, but doing it this way simplifies the whole process.



Above You need to tell Cura which printer you'll be using

There are two common slicers for hot plastic 3D printers: Cura and PrusaSlicer. We'll look at how to use them shortly, but first, let's start with an overview of what we'll be trying to achieve.

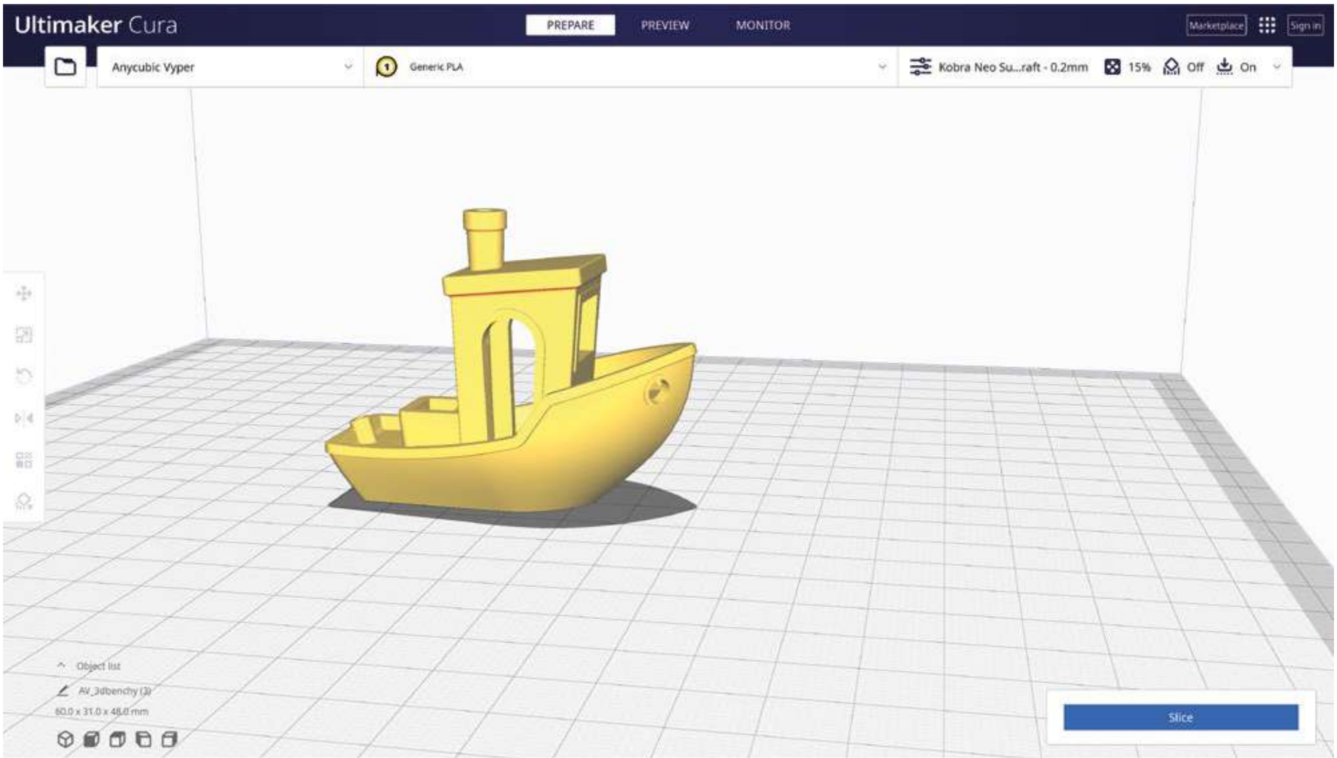
Usually, you start with a 3D design in an STL file, but some slicers can now read STEP files, so this might become a more common format in the future. Whichever format it is, you'll start with a file that describes the shape of the 3D object.

Since we're trying to get machine-specific instructions, we need to ensure that the slicer is configured for the exact model of printer we have. Some of this is fixed – for example, a printer will have an exact print area, and any attempt to print outside this will cause a problem. Some of it is more fluid – for example, there will be recommended speeds that you can print at – a well-setup printer may be able to go beyond this. All of these things should be set in the printer configuration that you'll need to load into your slicer. Generally, the 3D printer manufacturer will supply this, and you should have instructions for setting up a slicer with your printer. As you gain experience, you might want to tune it to your particular setup, but it's best to start with the manufacturer's recommendations.

Next, you must ensure you have the correct configuration for the particular filament you're using. Usually, printers will come with a recommended profile for PLA; if yours doesn't, or if you're using an unusual filament type, these are often fine to share between different printers.

There are some differences between different manufacturers of filament, and we'll look at how to tweak this for your particular filament a little later, but for basic use, the default profile should be fine.

Finally, there are some options that you might want to tweak on a per-print basis. The main one is



the layer height. This is the depth of each layer in the print. Smaller layers will print slower but give more detail, while thicker layers will print faster but with less detail. If you need parts to fit together, you might find that you need a thinner layer height to get the parts the exact shape needed.

Aside from the settings for the printer, you will also have some settings for the model itself. Once you import it, the first thing you need to decide is which way up to print it. Sometimes, model designers will ensure that the model automatically imports the right way up, but not always.

A few things to consider when deciding which way up to print a model:

- The print will be significantly weaker along layer lines, so if there's any force on the model, you want to ensure that it's perpendicular (or at least not parallel to) the layers
- Support material (see next step) will slow down a print and waste plastic, so it's good to minimise it

"It is possible to cut a model up into **smaller pieces and print them separately**"

- Any flat surface that sticks on the print bed will be the smoothest flat surface on your print
- A large flat surface on the print bed will help layer adhesion

Of these points, you can mitigate against all of them except the first, so that is perhaps the most important.

If you can't satisfy your needs with any orientation, it is possible to cut a model up into smaller pieces and print them separately, but this can introduce problems where warped prints don't fit together as they should. →

Above
The main window of Cura lets you position the object where you'd like it

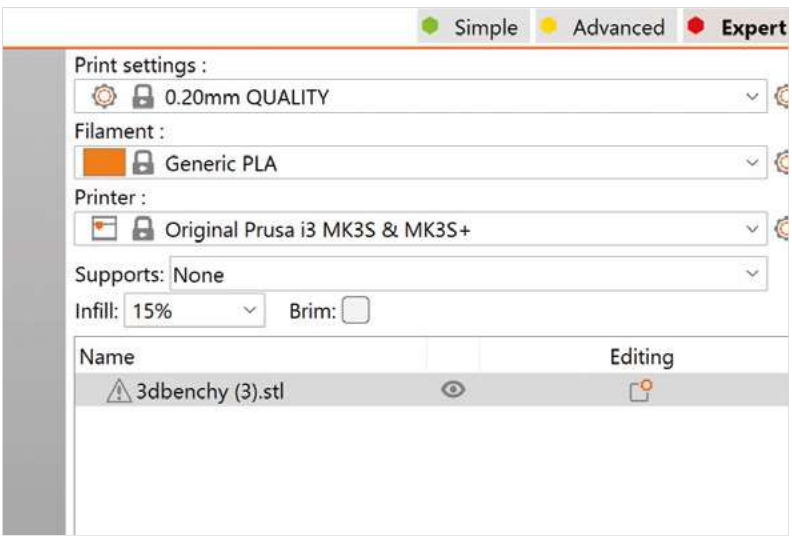
The final two things that you may want to consider are whether you need supports and whether you need a brim. Your printer should be able to handle a certain amount of overhang (where a layer above protrudes beyond the layer below) and bridging (where a top piece is supported on either side but not in the middle). However, if things go too far, you'll need some support.

Support material is a structure printed alongside your model to support overhangs and other bits that don't have model below them to print on top of. Once you've printed, you can snap or cut away the support material, leaving you with just the model you wanted.

There aren't any hard and fast rules for when you'll need support other than the fact that it's not possible to print in thin air. The safest option is to let your slicer add supports if it thinks they're necessary, but it will be very conservative, and you'll have far more supports than you need. With experience, you'll learn what your printer can handle, and sometimes it's good to try prints that you're not sure about without support, just to see if they'll print.

The risk of not having supports where you need them is either a messy print, if you're lucky, or a completely failed print.

Below
This author has a bad habit of forgetting to change the filament setting to the one in his printer



If you only have a small amount of print in contact with the print bed, you might want to add a brim, which is a bit of print one layer high around the edge of the print to help it stick down.

Let's look at how to do this for the ever-popular Benchy model in both Cura and PrusaSlicer. You can download the **3dbenchy.stl** file from **hsmag.cc/Benchy**.

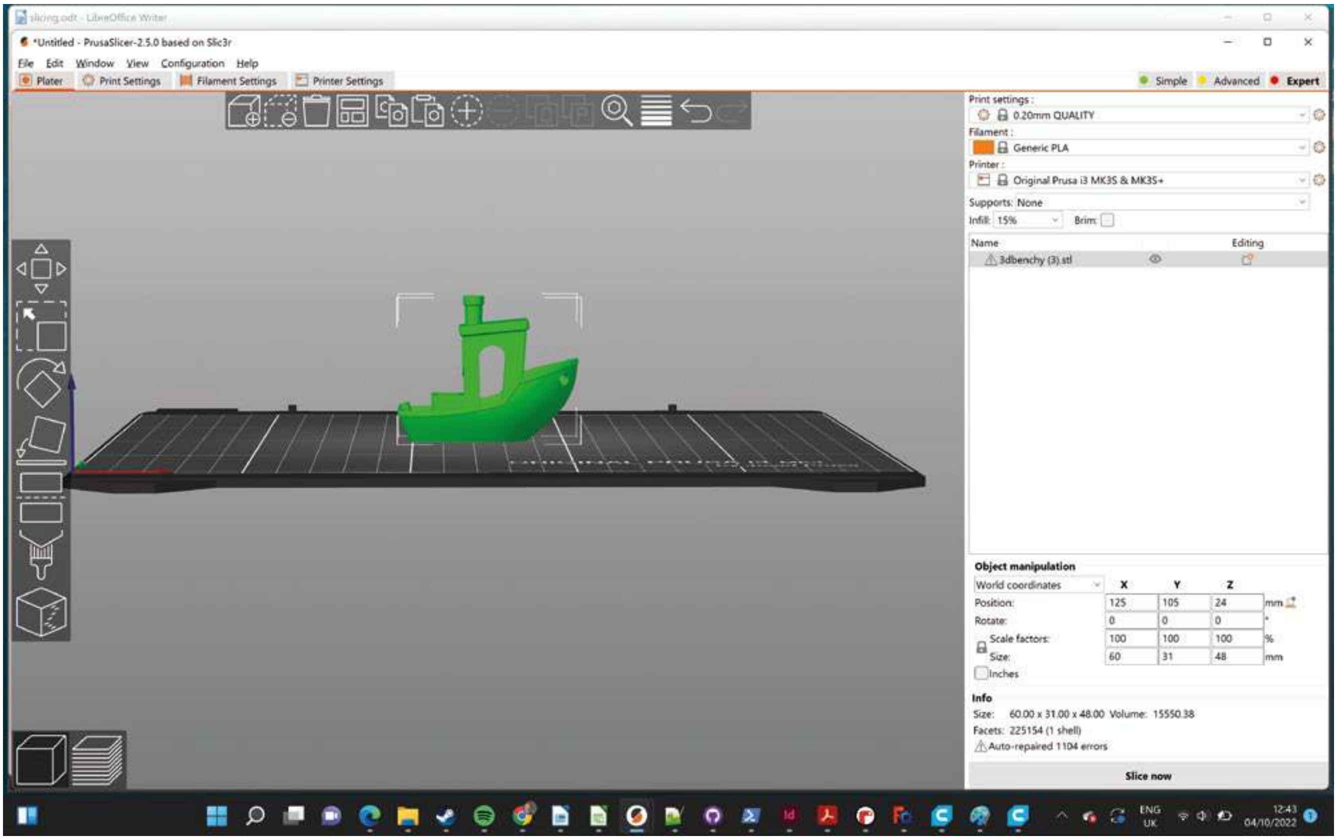
"The risk of not having supports where you need them is either **a messy print, if you're lucky, or a completely failed print**"

CURA

First, open Cura and start a new project if you don't already have an empty build plate (File > New Project). Add the Benchy model with File > Open and select **3dbenchy.stl**. This will load the model onto the build plate.

You can view the build plate from different angles by clicking the boxes in the lower left-hand corner.

If you click the model, it will get a blue outline to show that it's currently selected. The most important thing with a model is to ensure it's the right way up. Benchy should already be the right way up, but it's worth getting familiar with the controls. Hover the mouse cursor over the toolbar to see what each tool does. We want 'Rotate'. There should be three options: 'Free rotate', 'Lay flat', and 'Select face to align with build plate'. We rarely find the 'Free rotate' useful – at least for getting things the right way up; it's sometimes useful for positioning items diagonally on the build plate. 'Select face to align with build plate' is usually the best for lining things up. Click this (the square will get a black outline), and then click on the Benchy to line this up with the build plate. We want the boat the right way up, but it's worth getting familiar with this tool so you can line other models up.



PRUSASLICER

Starting with a blank project (you can go to File > New Project if there's already one loaded), import your Benchy STL with File > Import > Import STL and select your Benchy STL. This should appear on the build plate.

Benchy will appear on the build plate highlighted in green. Currently selected objects are green; unselected objects are orange. Click anywhere to deselect Benchy – it will turn orange. You can click and drag anywhere on the build plate to spin it in 3D to see how the object looks.

Click the object to highlight it green, and this will activate the toolbar on the left-hand side. You can hover your mouse cursor over each button to find out what it does. The most important of these, at least when you're getting started, is 'Place on face'. This will let you select which face you want on the build plate. If you select this tool, all the possible faces that could be placed on the build plate will be highlighted in translucent white. In the case of Benchy, it's already the right way up, so you are ready to go.

The print settings in the top-right control how the file will be sliced. Make sure that you have selected the correct layer height, filament, and printer. You can also select supports, infill, and brim. We don't need supports for Benchy, but you can select either supports everywhere or supports on the build plate only. A brim will help objects with a small footprint stick to the build plate. An infill of around 15% is fine for Benchy.

With all these set, select 'Slice now' to slice your model. This changes the interface slightly – in the bottom right-hand corner, you'll see the option to view the 3D editor or the preview. Up until now, we've been working in the 3D editor, but now we're in Preview mode. If you want to change anything, you need to switch back to the 3D editor. If you change anything that affects the slicing, the preview will go blank, which can be a bit confusing, but just select 'Slice now' again to regenerate the preview.

The orange vertical line can be used to select which layers are visible in the preview. This can be useful for taking a close look at areas where you're not sure about support material.

If you're happy with the result, press 'Export G-code' to save the G-code to a file ready to send to your 3D printer.

Above
All the key settings are listed on the right-hand side