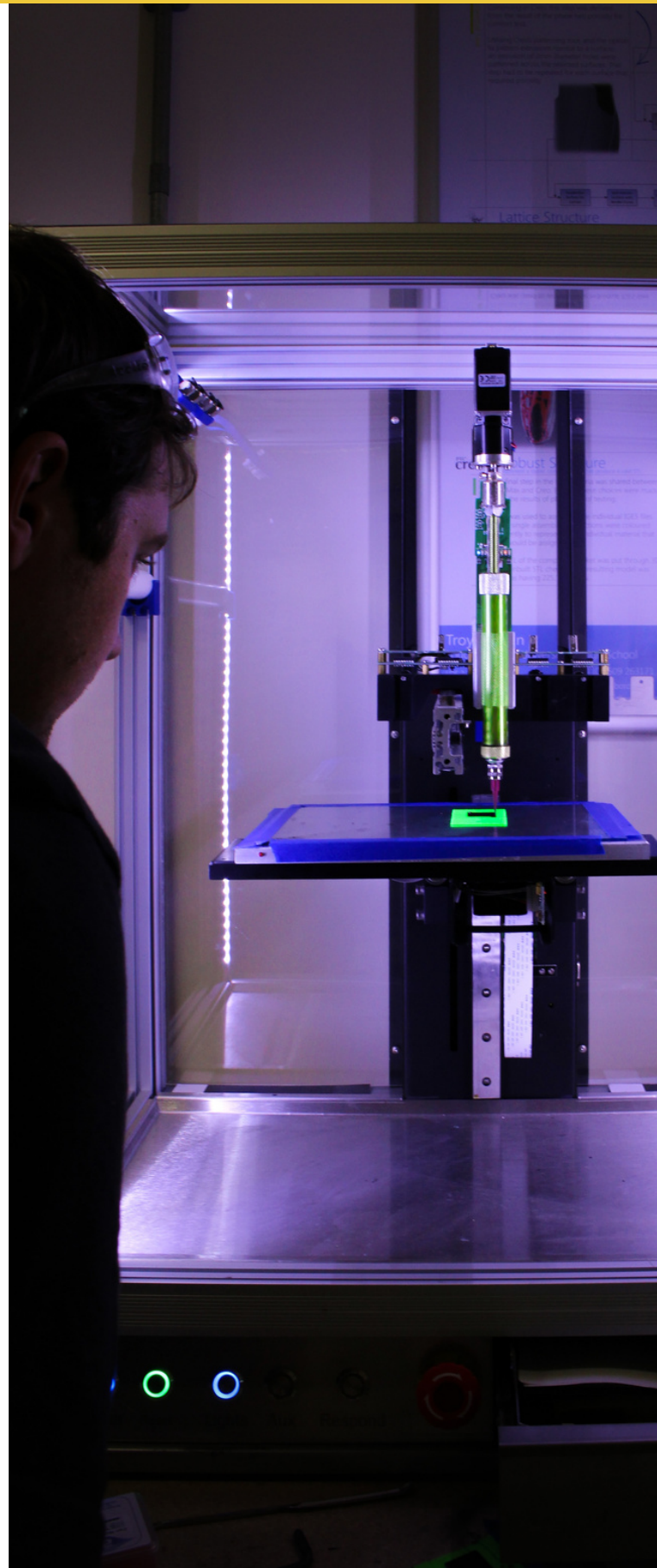




3D PRINTING GUIDE

3D PRINTER INTRODUCTION

3D printers are the fastest way to bring a concept into the physical world. Before 3D printers, you would have had to design your 3D model then contact a manufacturer to make injection-molded plastic parts.





3D PRINTING INTRODUCTION

At first, consumer 3D printers were costly and often unreliable machines, but now you can get a fully functional 3D printer for 300\$ or less and, with a little maintenance and know-how, will provide you with a high percentage of successful prints!

3D printing can seem daunting to a complete beginner and we hope that this guide will provide enough insight to help you get started on this exciting journey! First, we'll be discussing the hardware of a 3D printer. Understanding the hardware can help keep your printer in good condition and resolve bad prints. Hobbyists and professionals alike spend many hours adjusting their hardware and maintaining their machines. Next, We'll take a look at the process of producing a 3D model for printing, and then we'll discuss the very important slicing process, and how your slicer settings can make or break your prints. In this guide we will discuss free-ish or open source software, but we will not be covering resin or metal printers.

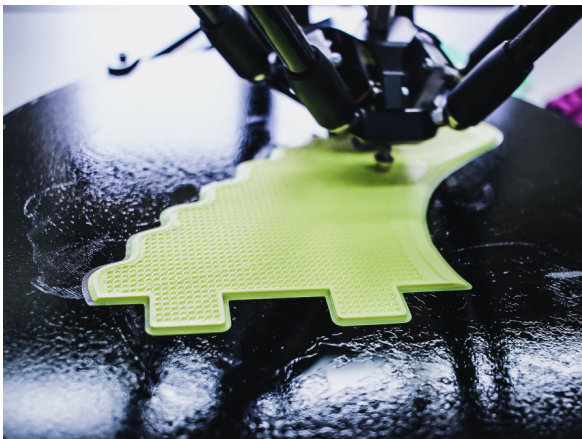
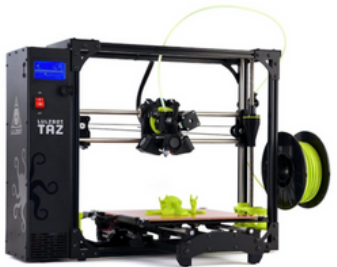
HARDWARE

Before we dive into the world of steppers and extruders, we need to ascertain the type of 3D printer we are using. There are two configurations of 3D printers on the consumer market: Cartesian and Delta. This is important to note because Delta printers will fail in ways that Cartesian printers will not, and vice-versa. The table below compares the two:

EXPERIMENTAL 3D PRINTERS

There are other configurations of 3D printers but they are experimental and not widely used. If you're interested, feel free to look into the world of polar and robotic arm printers.

COMPARISON	DELTA	CARTESIAN
Build plate shape	Circular	Square/ Rectangle
Noisy?	Quiet as a mouse	Loud
Does the build plate move?	No	Yes
Size	Tall	Shorter





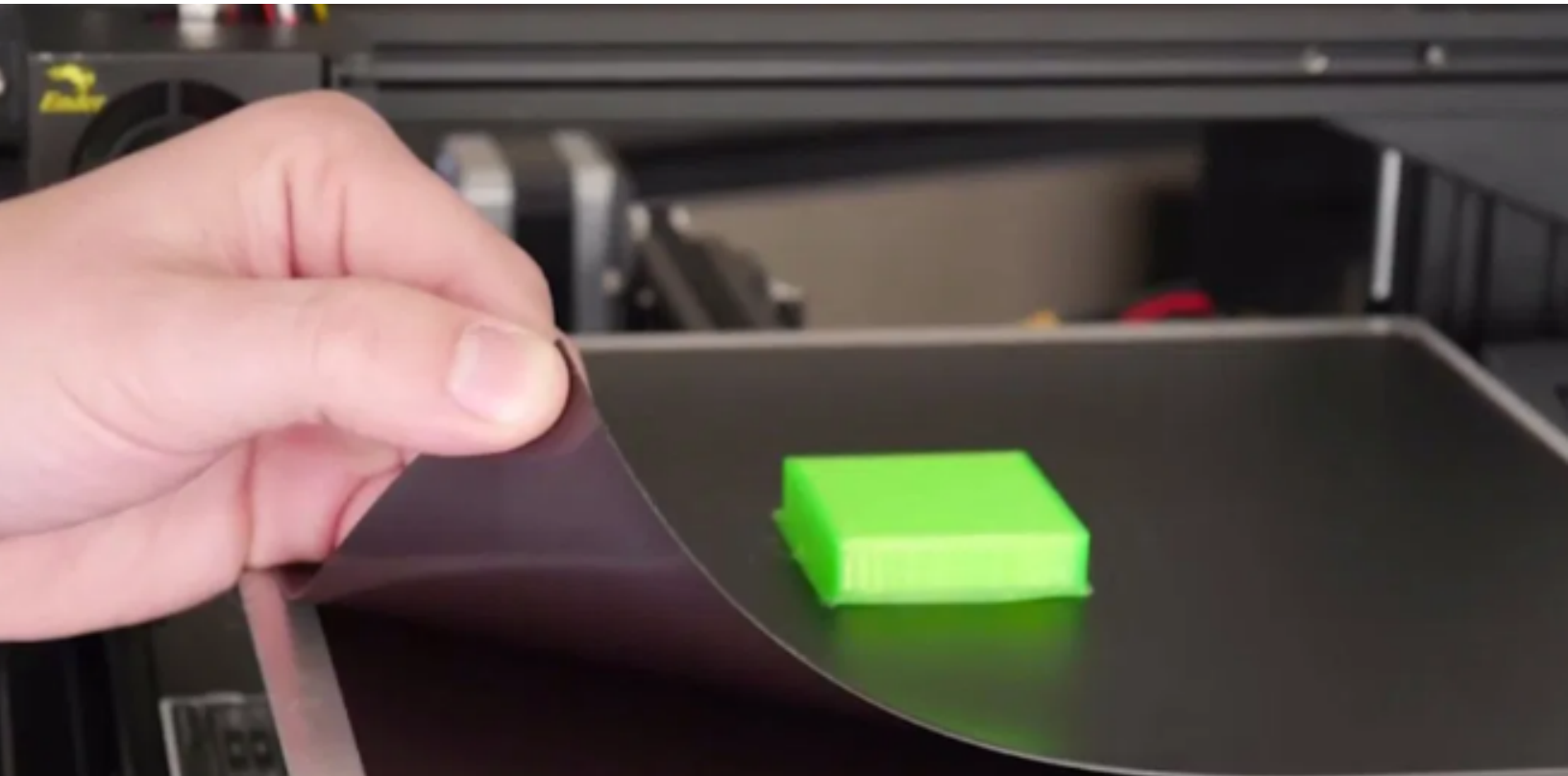
BUILD PLATE/PRINT BED

The build plate is the surface that your piece will be printed on. Delta printers have round stationary build plates, and Cartesian printers have rectangular build plates that move. Higher end printers have heated build plates. A heated build plate provides better adhesion but if you are experiencing issues, you may consider using some form of bed adhesive; sometimes, even a smooth glass heated bed isn't enough to keep a print properly stuck to the build plate. Purple glue sticks and hairspray are popular choices that are used to stick your print to the bed properly. Heated print beds are often made of glass but yours may have a rough or different thermally conductive build surface.

If you have a heated bed with a thermally conductive surface, it's best to replace it with

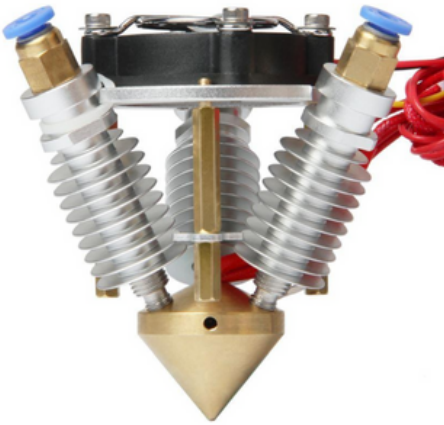
glass if possible. Even if you're careful, you may accidentally rip the surface of the build plate when removing prints. Some printers offer removable, replaceable, and even magnetic layers which cover the build plate. With the current cost of available printers, it is encouraged to choose models that offer a heated bed.

IF YOU ADD A GLASS BUILD PLATE TO YOUR PRINTER DON'T FORGET TO ADJUST THE CALIBRATION OF YOUR PRINTER TO COMPENSTATE FOR ADDITIONAL BED HEIGHT!



HOT END

The hot end is the part of the printer that heats up to melt your plastic. This is a common point of failure for printers since the hot end can get clogged or damaged over time. It's a good idea to keep a few replacements on hand in case something goes wrong. The size of your extruder will determine what thickness filament you use.



You can experiment with different metals and sizes of extruder, though 1.75mm brass is the most common. Your nozzles may need replacing over time and some exotic filaments require certain types of nozzles including stainless steel instead of brass. Please be sure to check the recommended nozzle type based on the type of filament you are using.

EXTRUDER

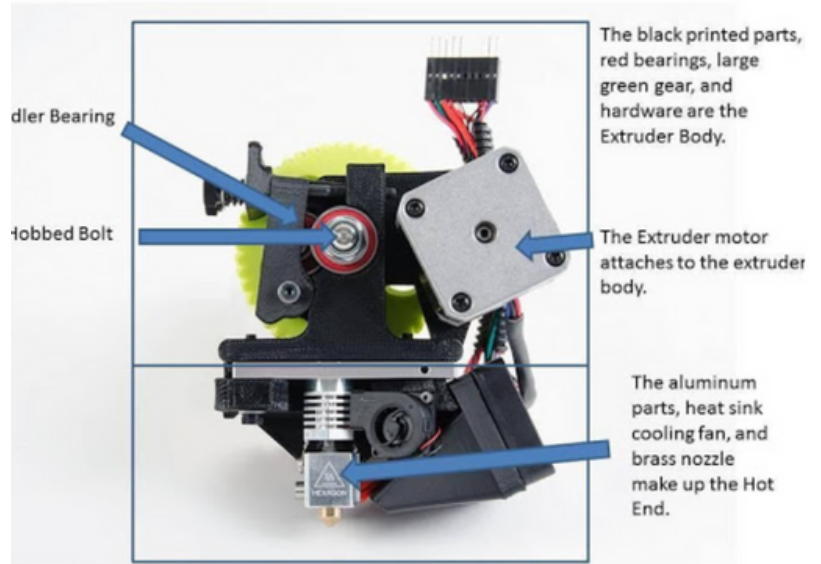


Photo courtesy of Matterhackers.com

There are two types of extruders available (Bowden and Direct) and the extruder is potentially the most important part of the 3D printer. It is the part responsible for making sure the hot end gets the necessary material and is often referred to as the “cold end”. The extruder motor pushes the filament into the hot end. This is often where you push a lever and spring to load in filament. Be warned that some extruders may have a hard time with certain plastics. How fast your gears are moving determines the flow rate of plastics.

FANS

3D printer fans cool filament once it has been deposited by the hot end. This helps the plastic harden so you'll have a cold print instead of a hot plastic puddle. Normally 3D printers will let you control the fans on the console. If anything happens to your fans, it's very easy to replace them as they are brushless DC motors.



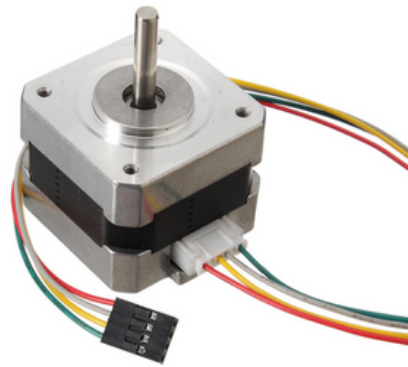


CONSOLE

Most 3D printers have a console. Your console will let you change the settings of your printer and start prints. Some consoles are internet based, like Duet3D, and others are all onboard, like the Ender 3. The most important function of your console is the bed leveling, which is essential for producing a good print. The second most important function is the ability to manually set the temperatures for your bed and hot end, which you'll have to adjust depending on your filament.

STEPPER MOTORS

Stepper motors are often used to move the arms, extruder, or build plate of your printer, or move filament towards the hot end. Steppers are used for their precision. Your 3D printer's motors may be connected to a rod or pulleys with a timing belt.



BOWDEN TUBE

Though not all 3D printers have bowden tubes, most tend to have them. If your printer doesn't have one, that means you have a direct drive extruder where the extruder gearbox will be directly on top of the hot end. However, most printers have the extruder far from the hot end. In those cases, the filament is stored in a teflon tube that leads into the nozzle. Your bowden tube has couplings that you can push in to remove the tube, which is handy if you have a blockage.

FILAMENT NOT FEEDING PROPERLY?

Make sure your filament strand isn't broken anywhere and be sure to check the gears for filament residue which could be causing your filament to slip!



PLASTICS

99% of the fun of 3D printing is exploring the variety of filaments on the market. 3D printing companies are getting pretty wild with their filaments and they offer sandable wood pulp PLA, plastics that feel like stone, and even filament made of beer brewing byproducts! However, before you dive into the wide variety on the market, you should understand the basic plastics and how they act.

THE MOST COMMON PLASTICS

ABS: ABS (Acrylonitrile butadiene styrene) is a plastic that has somewhat fallen out of favour for a variety of reasons. ABS smells extremely bad when printing, as it releases carcinogenic styrene gas when heated. When ABS is heated, it expands quite a bit and shrinks considerably when cooled down. It requires a heated print bed at a whopping 100-110C. It's generally made from petroleum, and most recycling facilities do not accept ABS plastics. However, it is extremely strong

FILAMENT SPOOL HOLDER

You hang your filament on it. Some 3D printer models offer a spool holder that can double as a carry handle if the printer needs to be moved.



and impact resistant, making it useful for functional prints. It does have a major caveat, it can be compromised by solvents. This can also be seen as an advantage, as the layer lines typical of a 3D print can be vapour smoothed away, or pieces can be welded together with acetone. **NOZZLE TEMP:** 230C **BED TEMP:** 110C

PLA: PLA (Polylactic acid) is the most common plastic used for filaments. It's very easy to print with, and is by far the most common filament used by hobbyists and educators. It doesn't release gas like ABS and actually smells rather nice when printed. Heated beds are not a requirement since it doesn't warp too much when cooling. PLA is a corn based plastic, meaning that with some effort it can be composted, but not recycled. It isn't as strong as ABS but with the right settings it can be used for functional prints. **NOZZLE TEMP:** 200-220C **BED TEMP:** OPTIONAL 50-65C **PETG:** PETG (Polyethylene terephthalate glycol-modified) is the less popular rigid plastic for printing. It is

COMMON PLASTICS CONTINUED

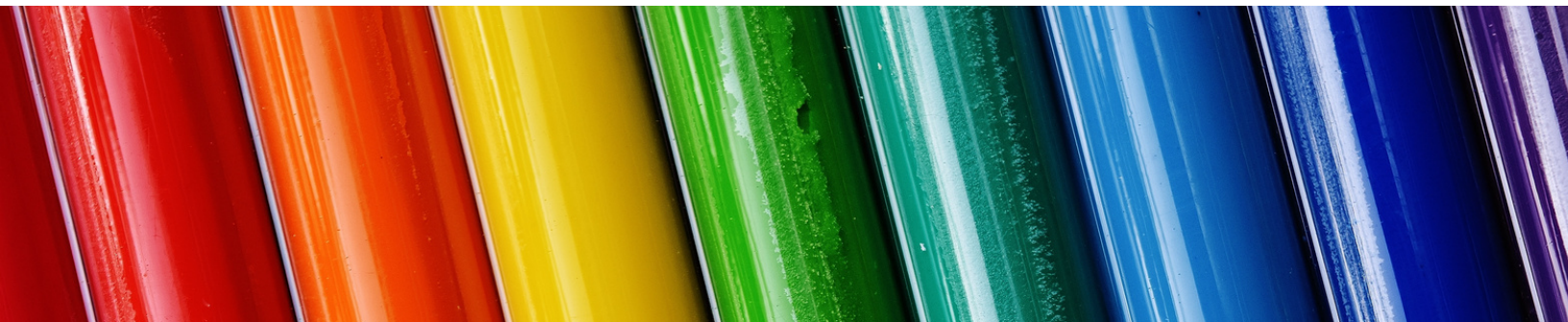
also one of the most common plastics in the world, with polyester fabric and plastic water bottles being composed of the PET family of plastics. This means that your failed prints can be recycled and are considered food safe. PETG doesn't warp, and sticks to itself well, making it stronger than ABS in certain circumstances. Unfortunately, support material is difficult to remove from PETG and it requires a heated bed. NOZZLE TEMP: 220-240CBED TEMP: 70-75C

TPU: TPU (Thermoplastic polyurethane) is a flexible filament, making it useful for wearables and shock absorption. It has elastic properties, which means it can stretch over a phone! TPU is petroleum based and cannot be recycled and is not food safe, so it is limited in that regard. Another downside is it's difficulty, as it doesn't like to stick to the print bed nor does it look great. Even a new reel of TPU can have issues with stringing. It needs to be printed slowly to have good results. However, it is much easier to print than other flexible filaments on the market, and a heated bed isn't required. NOZZLE TEMP: 190-215CBED TEMP: OPTIONAL 55-65C

FILAMENT CARE

Waterlogged Filament

All 3D printing filaments have a shelf life. Over time, the plastics become brittle and print at a very poor quality. This is because the plastics used in filaments absorb water from humidity in the environment over time, a property called hygroscopy. When waterlogged filament is printed, the water in the plastic is boiled off in the hot end. If you listen closely when this happens, you can hear sizzling and often results in bubbles or pimples because of the boiling water. ABS and PLA are mildly hygroscopic. They can be kept in open air for around 6-8 months until they start to degrade dramatically. PETG is slightly more hygroscopic than PLA/ABS, but not as bad as TPU or nylon. TPU can go bad in a matter of weeks, while nylon degrades in hours. You can slow down the degradation of filament by storing your filament in a box with desiccants, and you can dehydrate waterlogged filament in a food dehydrator, or in an oven with great caution.



SOFTWARE

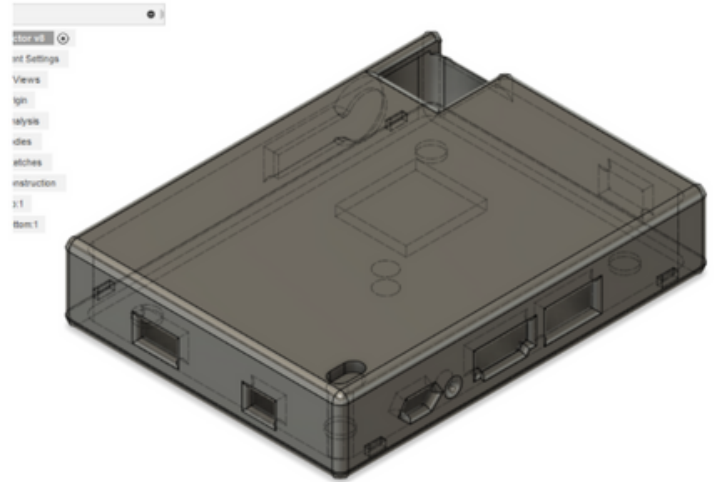
Before you can get your 3D printer moving, you'll need two different kinds of software: a slicer and a 3D modeling program. We'll talk about the 3D modeling programs first and some design considerations.

3D MODELING SOFTWARE

3D modelling is by far the most time consuming part of the 3D printing process. Before you pick up 3D modelling, you'll need to choose software that suits your needs. Broadly speaking, there are two kinds of software you'll use for 3D printing: mesh modelling and CAD modelling.

CAD MODELLING

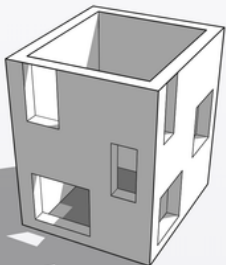
If you need accurate dimensions for functional parts, you'll want to look on the CAD side of 3D modelling. CAD is based on primitive lines and shapes which are refined with different tools to create a model. Ratios and exact measurements are key to the modelling workflow, and organic models are very difficult to make. The advantage of CAD style modeling is that you don't have to manage the numbers of polygons on your design. CAD software has many options for new users and experienced users, with the most basic ones being Sketchup (<https://www.sketchup.com>) and Tinkercad (<https://www.tinkercad.com>).



Above: The viewport of Fusion360. A case for a miniature projector has been modeled accurately and is ready for slicing. Note how there are no vertices.

GETTING THE SIZE JUST RIGHT :)

Focus on getting the proportions right in your 3D modelling software and you can set the size and dimensions later in your slicing software.



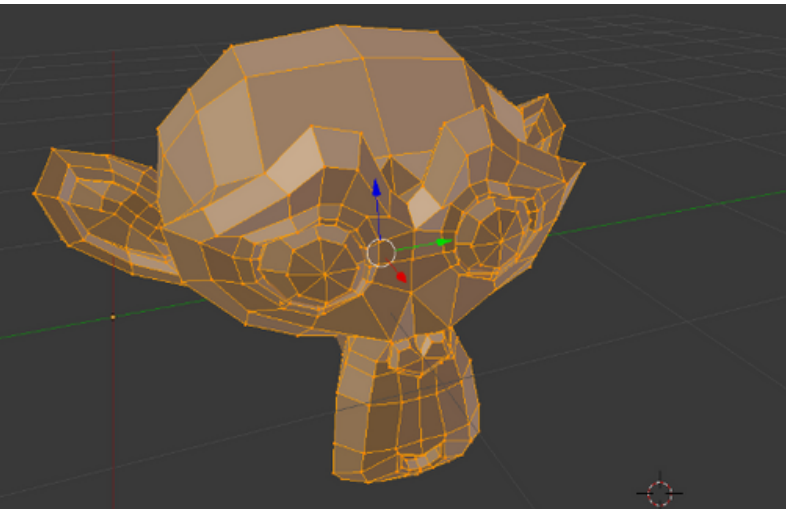
CREATE
Visualize your Ideas

The joy of drawing by hand. The ease of super-smart software.



MESH BASED MODELLING

If your focus is on character modelling and organic shapes, then mesh based modelling is for you. Mesh based modelling has a slightly steeper learning curve, but there is less to learn overall compared to CAD style modelling. Mesh modelling uses three components to create 3D models: faces, lines and vertices. This allows you more control over every aspect of your design. The major downside of mesh modelling is the lack of dimensional accuracy. Most mesh programs use arbitrary units with no correlation to real life unit sizes. Sculptris is one of the easiest mesh-based programs, while Blender is considered by some to be the most difficult.



Above: Blender's viewport. The monkey is comprised of vertices that form edges and quadrilateral/trilateral faces.

FILE FORMATS

Once you're finished with your 3D model, you'll need to make sure the file format is compatible for slicing, the next step in the 3D printing process. The most common file formats are OBJ and STL. Both of these files can be looked at in Windows 10 if your OS is up to date. If your 3D modeling software's default file format isn't STL, such as blender's .blend format or Fusion360's .f3d, you can typically export it to a different format from the File tab.

SLICING

It is impossible to print an STL or OBJ file. It must be converted to G code before it can be printed. G code is a set of instructions for the movement of the stepper motors to follow to print out your model layer by layer. That's why it is called a slicer, as it slices your model into 2D lines and layers these images, or slices, one on top of the other which then corresponds to the movement of the print head. G code also contains instructions for setting hot end and bed temperatures as well as the strength of your model, the resolution it prints at, if rafts are needed, and what supports are required.

SLICING SOFTWARE

Some 3D printers come with their own slicer, but most don't. If you have a 32 bit computer, or use a chromebook, you will have to use a browser based option like Astroprint. If you have a 64 bit computer, you have a wealth of options to choose from. One of the most popular free slicers is Cura created by Ultimaker (<https://ultimaker.com/software/ultimaker-cura>), which is the one this guide will use for the purpose of demonstration. There are many options available, including Matter Control, created by Matter Hacker (<https://www.matterhackers.com/store//mattercontrol/sk/MKZGTDW6>), so you will need to try some to see which one you prefer.

SLICER SETUP

Once you've installed Cura, you'll need to add your printer profile. If you have a mainstream 3D printer, it will be pre-loaded in the software. If your printer isn't on the list, you can input the settings manually.

Next, you'll need to set your filament. Cura has a lot of branded filament options as well as generic settings. If you're using TPU or another less common plastic, it's best to avoid the branded options and follow the temperatures written on your spool of filament.

SLICER SETTINGS

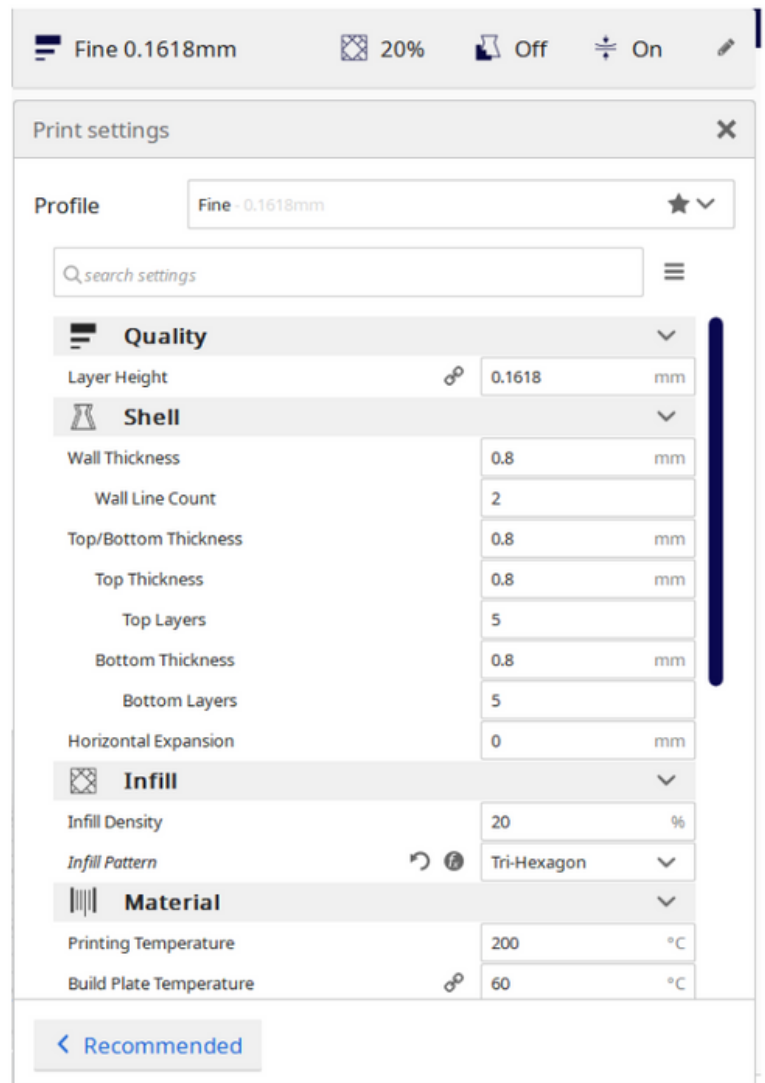
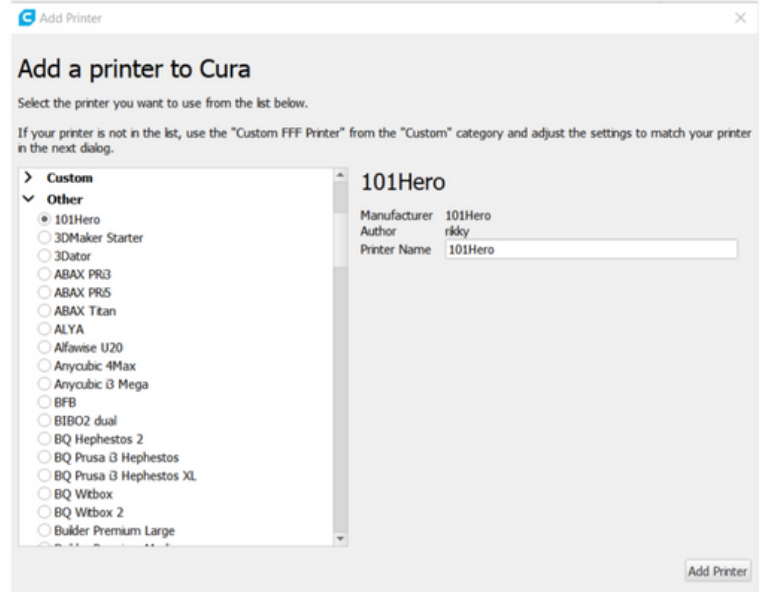
Under the settings tab in Cura, you'll be confronted with a lot of options, all of which can make or break your print. Hobbyists dedicate a lot of time adjusting their slicer settings to get the best print. This guide will touch on the most consequential slicer settings.

LAYER HEIGHT

The layer height is directly tied to the resolution of your print. If you're uncertain if your print will turn out, you can turn the resolution down to draft quality(0.2mm), though be warned that finer details might not be printed. The lower the resolution, the faster the print. You can also consider going to 0.3mm.

SAVE SOME PRINTING TIME!

Be sure to double check your infill as you rarely need 100% infill density. Most prints can get away with using anywhere from 10-30% which will save you printing time!



CONSIDERATIONS

In this section we consider the set up considerations, infill patterns, temperatures adhesive plate, support to keep your 3D printer in running shape and finally getting ready to print.

INFILL

3D printers don't often print solid pieces of plastic. It would take far too much time to print and it would be a waste of filament. Instead of it being hollow or completely filled, a geometric pattern is printed to give structural support. If you're using a flexible filament, the infill will affect the squishiness of the print. There are many infill patterns to choose from with different properties. Some infills, like tri-hexagon, print faster but also may leave cosmetic defects in your print where the wall meets the infill depending on your selection of Infill overlap which is a setting that determines how much infill overlaps with the innermost shell lines. Of course, your infill pattern will be a matter of preference. If you're looking for a safe bet, tri-hexagon and grid are the most common and are fast to print. The infill density determines the size of the infill pattern, as well as the print time. If you're not concerned about your print being under mechanical stress, feel free to leave the density at 20%. The only time I have ever personally used an infill higher than 20% was for a TPU skid plate for a skateboard. Since the part was going to be dragged along pavement, a higher density was needed.

TEMPERATURES

temperatures are easy to set, simply choose a generic plastic from the filament settings, or follow the nozzle temperatures on your filament reel. Often filament reels don't list a heated bed temperature, so you'll have to do some research and experimentation to determine what works for you.

Z HOP WHEN RETRACTING

This setting is for stringy or stretchy filaments. In plain english, a Z Hop Retract is when your printer pulls the extruder head up vertically when the extruder is pulling the filament backwards. This can prevent stringing from the hot end and keeps the hot end from dragging across your print.

SUPPORT

Since your machine prints layer by layer, and you can't print onto thin air, you'll need to generate support structures for any overhangs. Some 3D printers can handle overhangs better than others. Some printers may tolerate overhangs larger than 45°, some being able to print 60° overhangs comfortably; however, that will depend on nozzle size, fan type and settings, and even the fan shrouds. When your model is displayed in Cura, surfaces that need support will be coloured red.

When a support structure is printed, it must be removed by breaking the thin pieces of plastic after printing. These can leave scars on the surface of your print. If your design can accommodate it, avoid using support unless necessary. By default, Cura uses Linear type supports. In the experimental settings, you can try using treelike supports. Instead of printing rectangular shells to support overhangs, it generates a structure of tree branches to hold the piece up. These have the advantage of minimizing contact with the surface of your print and saving filament. Feel free to experiment!

FINISHING THE SLICING PROCESS

Once you're happy with your slicer settings, you can hit the slice button in the bottom right-hand corner. Cura will give you an estimate of how long the print will take, as well as how much weight in plastic it will use, and how many meters of filament. If you hit the preview button, it will show a simulation of the printing process. Once you hit "Save to file", it will give you a ready-to-print G-Code file.

GETTING YOUR PRINTER READY TO PRINT

Before you load your G-Code into your printer, you'll need to make sure your printer is ready to print. If you don't have a heated build plate, you should clean off your plate and put down a fresh layer of adhesive. If you have a heated bed, make sure to wipe off any leftover filament, and make sure your filament can't get tangled in the reel. After this you're ready to print! Look online if you're unsure of how to load a G-Code onto your printer

TROUBLESHOOTING

3D printing is all about managing a delicate act of cooling, heating, and precise motor movements. If the balance is disturbed, it will be visible from minor cosmetic defects to your printer spilling out spaghetti like strands everywhere. In this section we'll go over the various printer settings that can be tweaked if you come across a problem.

Z OFFSET

If your nozzle is so low it's not extruding and scraping your bed, or it's too high up and giving you a sloppy print that falls apart, adjust the Z offset. This will offset your printer's starting vertical height.



Above: A bad first layer caused by a Z offset issue. The distance between plate and nozzle is too high. Source: mrbnatural10 on 3D Hubs

PRINT SPEED

Some prints will be sloppy no matter what, and the only solution is to slow things down. Slowing down your print is often used as a last resort, after you've exhausted your other options. Increasing the print speed is generally not advised unless you are very sure of the success of your print

3D PRINTERS CAN BE TRICKY

3D printers require care & maintenance as well as perseverance and resilience!

PRINT SAMPLES

Not all 3D prints are of equal quality. In this section we review print samples and share terms and considerations about the quality of your printed designs.

ECHOES/GHOSTING

If you print your design and notice faint “echoes” of features on the outside, it means that your 3D printer is vibrating and the vibrations are shifting around the plastic as it hardens. To prevent this, you can keep your printer on a soft surface that can absorb some of the energy. This does not work for Delta printers generally, so you may have to adjust travel speeds and acceleration speeds in your slicer to slow things down. If that doesn't work. You may need to check belt tightness and other mechanisms.

BLOBS AND PIMPLES

If the surface of your print is blobby and bumpy, you're dealing with some over-extrusion issues. This happens when too much plastic is coming out of the nozzle at once. Sometimes this can be caused by your retraction (when the extruder pulls back the filament) settings. You may have to increase retraction distances to stop the blobbing. If the blobs are at the end of a line or layer on the surface, you will have to turn coasting on to relieve some of the built up plastic in the nozzle.

ELEPHANT'S FOOT

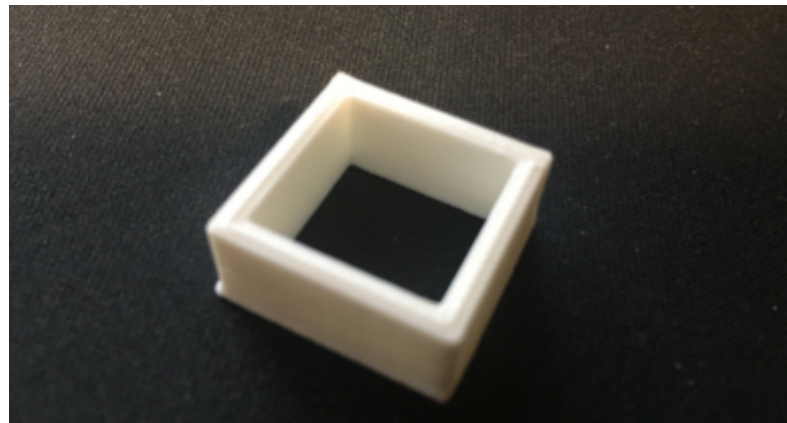
This is a problem that is easy to fix! If the bottom of your print is flaring out, it means your temperatures are too high, either from the bed or the nozzle. Reduce your temperatures and make sure you're printing at recommended temperatures. If your temperatures are ok, you may have a more serious issue relating to your fans or temperature sensors.



Notice the right cube shows ghosting.



Pimples on a print. Source: Steve Lowry, Gear Best blog



The bottom of the cube is flared out from high temperatures. Source: user441767 on stack exchange

SAFETY CONSIDERATIONS

3D printing is great but users need to use caution. High heat, moving parts, and certain materials can cause safety concerns.

HIGH HEAT

3D printing temperatures of the hot end can reach heats well above 200 degrees Celsius. Great care should be taken when dealing with 3D printers for this reason. The hot end should never be touched while heated and during operation. If touching the hot end is absolutely necessary then users should only attempt to do so with a tool which does not conduct heat.

Some cases which may necessitate manipulating the hot end during printing may involve a print that is doing well but you notice filament is building up near the hot end which could cause future problems and your print to fail. In this case a scraper with a non heat conductive handle could do the trick. This should be done quickly and the scraper should focus on the filament rather than the hot end itself.

MOVING PARTS

With a many stepper motors, belts, bearings, and gears all working together to print your model, please keep you hands and other items clear of all moving parts.

ABS FUMES

When heated, ABS plastic will release toxic fumes. ABS should only be used when necessary and only when proper ventilation and aeration is available. Some 3D printer models are available with enclosures which stop the printer movement if the door is opened and starts the cooldown process for the hot end automatically. Keep in mind cooling down the hot end can take a long time so opening the door on these models does not mean it is safe to touch the hot end!

CONCLUSION

These are only some of the issues that you may encounter with your 3D prints. Because 3D printers encounter a lot of wear and tear, they will need to be maintained often. If you're ever unsure why your prints are of poor quality, feel free to ask online communities for help! 3D printing forums and subreddit are full of experts who are eager to help new users. It is important to be patient, persevere, and experiment. Your goal should be to improve the quality of your prints. Be sure to share your successes online as well, 3D printing is all about sharing and collaboration

Written by Zoe Ritchie
Co-authors Caleob Maher Watson
& Ethan Zatt

Let's Connect

If you're interested in learning more about 3D printing modeling and projects let's connect and **#MakeSomethingBrilliant**

EMAIL: INFO@BRILLIANTLABS.CA
TEL: 506-442-9059
BRILLIANTLABS.CA

