

create: a primer on 3D printing

Ryan Eisworth
Houston Area Apple Users Group
July 2018

What is 3D printing?

- A type of additive manufacturing called Fused Deposition Modeling (FDM). Most 3D printers extrude plastic filament to build up a part on a work bed. This is in contrast to making a part on a mill or lathe where material is removed to create the desired part.
- There are some limitations inherent to all manufacturing processes. 3D printing is no exception.
- There are many material choices for 3D Printer filament material. The two major options are PLA and ABS. Many other specialty materials exist.

What is 3D printing?

- A 3D print is made up of many layers. With a consumer printer, these are usually usually around .010 to .020 mm thick. These layers are stacked on top of each other starting from the work bed to form the part. When the printer puts the next layer down on top of the previous layer, the heat of the material being extruded causes it to melt into the previous layer, forming a complete part.

What do you need to get started with 3D printing?

- A 3D printer (or access to one).
- Filament that the printer can use.
- A 3D model of the part you want to create.
- Software to turn the 3D model into something the 3D printer can use (called a slicer).

Getting a 3D printer...

There are a few maker groups and hacker spaces that may offer you the opportunity to use a 3D printer without having to buy one for yourself.

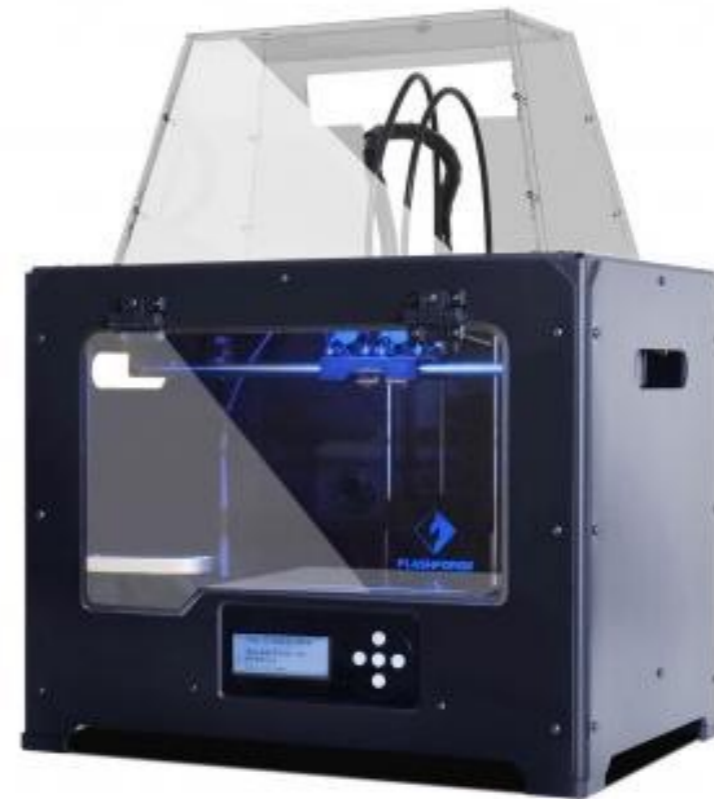
- TXRXlabs <<https://txrxlabs.org>>
205 Roberts St.
Houston, TX 77003
- Creator Space <<http://creatorspace.org>>
120 Park Ave.
League City, TX 77573

Getting a 3D printer...

- If you're looking to purchase your own printer, there are countless choices available at price points ranging from a few hundred to tens of thousands of dollars.
- In general, try to get a machine from a known manufacturer.
- The filament material you want to use will inform your decision. You'll need a machine with a heated bed plate if you want to print ABS.

Getting a 3D printer...

- My preference is for "enclosed" printers versus "open frame" designs.
- MicroCenter (5305 S. Rice Ave.) has a selection of 3D printers in stock and on display. They also have a wide selection of filament and repair parts stocked.
- My machine is a PowerSpec 3D Pro, originally sold by MicroCenter, that I bought off of Craigslist. It is a similar machine to the FlashForge Creator Pro.



Picking a home for a 3D printer...

- Make sure you have a good place to run a 3D printer.
- 3D printing involves melting plastic, which can release chemicals that are bad for you. You'll want to put your printer in a well ventilated place like a garage or utility room, where harmful fumes won't build up and be inhaled by living things.

Picking a home for a 3D printer...

- 3D printers are inherent fire hazards. A lot of energy is being pushed around to make enough heat to melt plastic in a controlled way. You'll want to pick a location away from flammable materials where you can monitor the machine while it is working.
- 3D printers are loud. You'll want to put the printer in a place where the sounds of the robot uprising won't disturb others.

Picking a home for a 3D printer...

- If you live in a small apartment, having a 3D printer around may not work out very well. That doesn't mean 3D printing is out of reach. Look for Maker Groups in your area where you can use a 3D printer without having to run it in your living space.

Anatomy of a 3D printer

- Bed plate
 - This is where your 3D print begins. The bed plate can be heated or non-heated. A non-heated bed plate limits your material choices.
- Extruder
 - The extruder takes your filament, heats it until it is pliable, and forces it out of a nozzle. This is one of the most important parts of the printer.

Anatomy of a 3D printer

- Gantry
 - The gantry moves the extruder around on the X- and Y-axis, allowing the extruder to "draw" with the filament.
 - There are a few other more complicated systems than a standard X- and Y-axis gantry but they are far less common.

Anatomy of a 3D printer

- Filament spool
 - The printer needs a way to hold filament. Many models will just have a dowel to hang a spool from, but some have a proprietary system that will only accept the manufacturer's filament cartridges. Avoid these.
 - Printers can use either 1.75mm or 3mm filament. 1.75mm is more common, so stick with that. You'll have more filament choices and better availability.

Getting started...

Printer setup is machine specific, but there are a few things you'll probably need to do regardless.

- Bed leveling: your printer's bed plate needs to be level relative to the extruder. This is usually accomplished by using a shim of a known thickness (like a sheet of paper) as a feeler gauge, and checking the gap between the extruder and bed plate at a few points.

Getting started...

- Filament loading: this usually involves entering loading mode via your printer's front panel, waiting for the extruder to preheat, and feeding filament into the extruder once the printer signals that it is ready.
- Bed plate preparation: it will be dependent on your printer, bed plate, and material how you'll prep your bed plate for the best first-layer adhesion.
- Calibration and setup: this part spills into setting up your slicer, which we'll cover later. You'll need to define things like infill, extrusion temperatures, and print speeds.

Bed plate preparations...

- Most printers with a heated bed plate will have a factory-applied polyamide (Kapton) tape. This is a heat resistant tape with a yellow-gold appearance. Do not remove or damage the tape or you'll have to apply a new piece.
- You can print directly onto Kapton tape, but I have better luck covering the Kapton tape with blue painter's tape, despite having a heated bed plate.
- Other options include printing directly onto glass clipped to the printer's bed plate, either bare or treated with a glue stick or hairspray.

Why do bed preparations matter?

- One of the first problems you'll run into when getting started with 3D printing is "first layer adhesion." You need the filament that forms the first layer of your print to adhere to the bed plate. If it doesn't stick, your print will fail.
- Once the printing is complete, the next problem is getting the part off of the bed plate without destroying either the part or the polyamide tape on the bed plate.
- Ideally you want your first layer to stick well enough to hold during the printing process, but still be able to pop off easily when the print is complete.

Even more on bed plates...

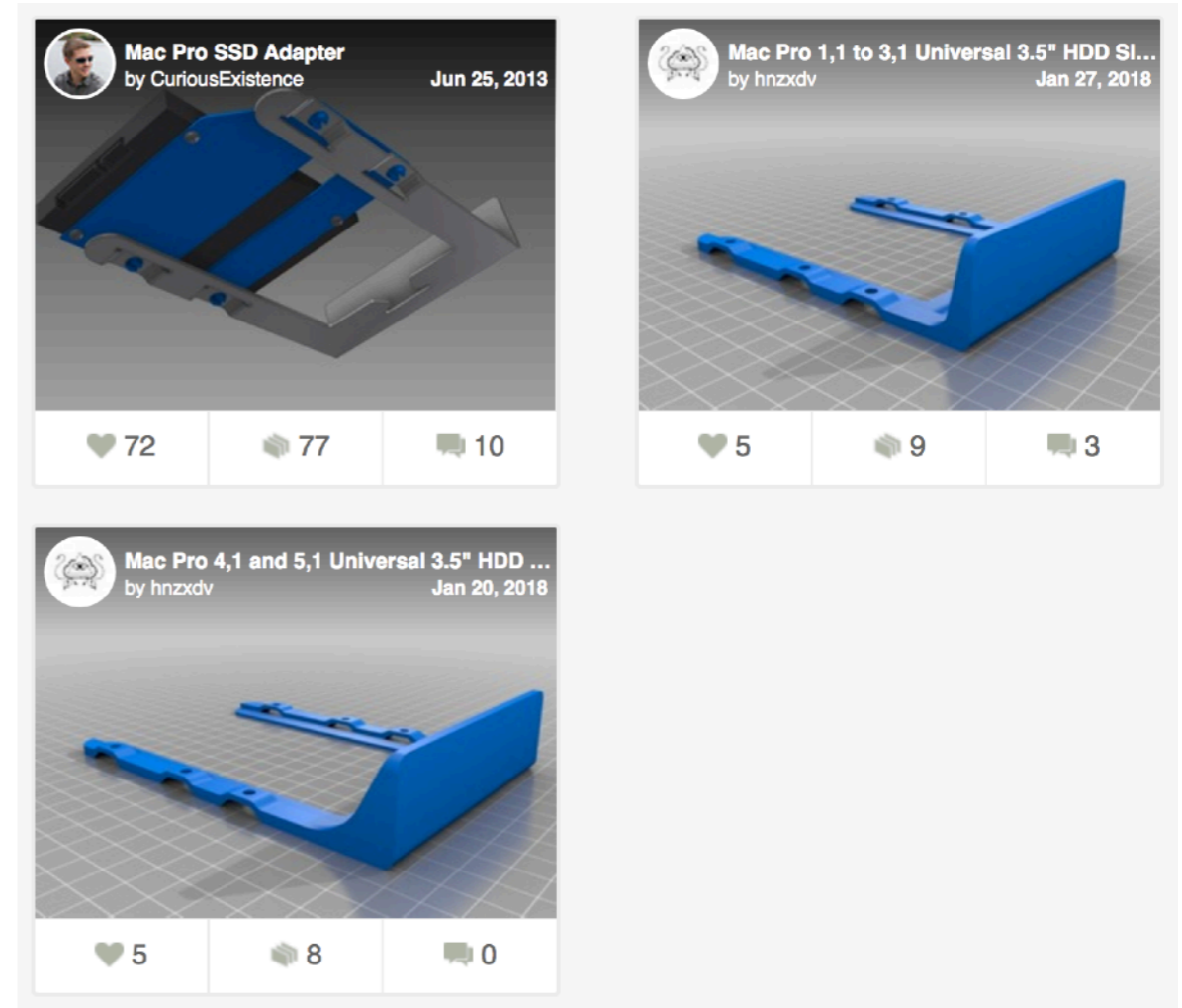
- One thing is certain: you can't go wrong by getting a heated bed plate. Even if you're just printing PLA (which doesn't require a HBP) having the heated plate can give you more options when it comes to solving first layer adhesion and part release issues. It also opens the door to printing with ABS and other types of filament.
- Personally, I tend to use blue painter's tape and the heated bed plate of my printer to help with first layer adhesion. Sometimes part adhesion is still a problem for me. I am interested in trying direct-to-glass printing, but haven't experimented with it yet.

3D models...

- Once your 3D printer is ready to go, you're going to need a 3D model of the thing you want to print.
- You can either find a model online that someone else has already drawn, or model the part yourself.
- For finding models online, the best resource I have found is Thingiverse <<https://www.thingiverse.com>>.

Thingiverse

- Thingiverse has thousands of free-to-download 3D models that are provided by community members.
- Categories range from toys to car parts and everything in between.
- Most uploads are in .STL format, ready to go into your slicing program.



Drawing your own 3D models...

There are many popular (and Mac-compatible) applications that you can use to draw your own 3D models.

- TinkerCAD (AutoDesk; Free; <<https://tinkercad.com>>) is a simple 3D design package that runs right in your browser and is free to use.
- SketchUp (Trimble, Inc.; Free, Pro Version \$695; <<https://www.sketchup.com>>) is a Mac and Windows application with an available free version.
- Fusion 360 (AutoDesk; \$25/month; Free License Available; <<https://www.autodesk.com>>) is a full-featured "cloud powered" native application with powerful functionality. Free licenses are available if you qualify (and you probably do.)

Free Fusion 360 licenses...

- From AutoDesk's site:
*"The free **Start-Up/Enthusiast licenses** allow you to access Fusion 360 with a yearly subscription after the trial period has ended. You can use this license if you are a small business making less than \$100,000 per year (or equivalent), or if you're a hobbyist using Fusion 360 for non-commercial purposes."*

Learning software for 3D design

- All 3D design software is going to have a learning curve.
- Of the three listed options, TinkerCAD will be the easiest to get into. SketchUp will have a bit more of a learning curve. Fusion 360 has the steepest learning curve, but is the most powerful of the three.
- Look for tutorials on YouTube and the software vendor's website to get started.
- You'll make lots of mistakes but it'll get easier.
- My best tip for 3D drawing: *change camera views often!*

Model considerations...

- When drawing or considering what to draw, remember the limitations inherent in the 3D printing process. Think like a 3D printer and like the software that's going to have to slice your model for the 3D printer. As the part is built up from the bed plate, each new layer needs something to adhere to.
- Examples of things that 3D printers can potentially have trouble with are spans (places where material is expected to go across two points), overhangs, and vertical holes.
- A good slicer can do wonders to help minimize these issues by adding supports or changing printing speeds.

Visual examples of 3D printing limitations...



Image: 3dhubs.com

This sample print demonstrates the issues of printing overhangs. After roughly 45-degrees, layers start to separate because there isn't enough support material for subsequent layers to adhere to.

Visual examples of 3D printing limitations...

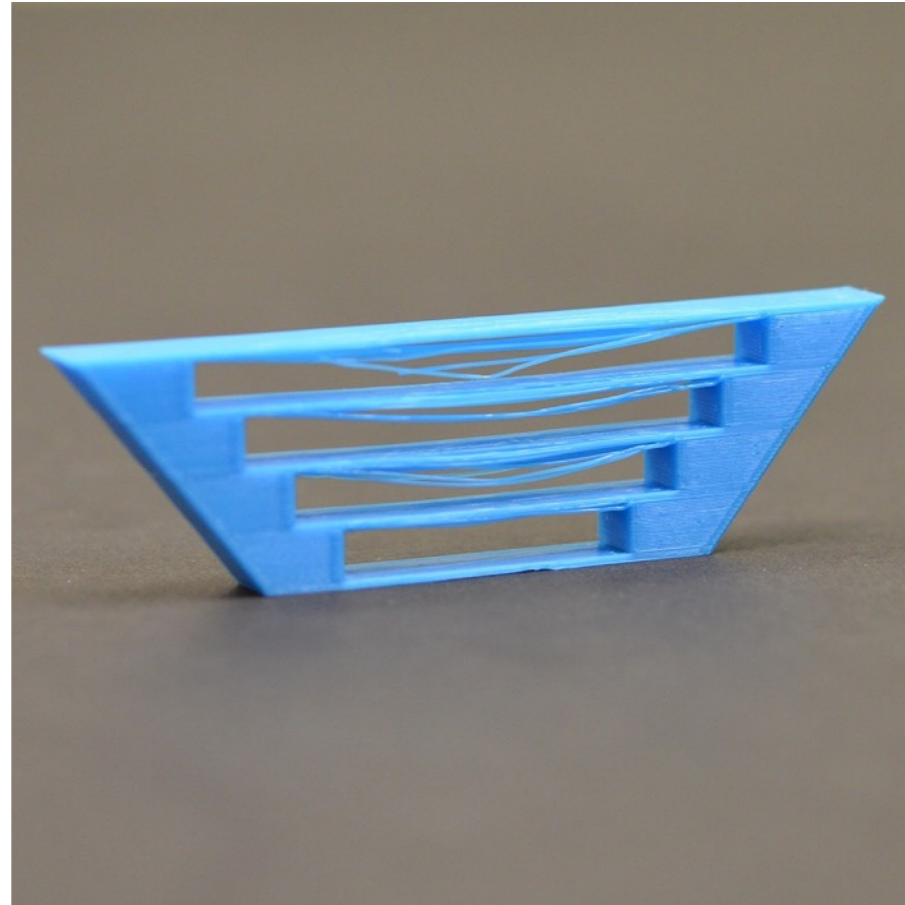


Image: Simplify3D.com

This sample print demonstrates the issues of printing bridges or spans. As distance of the span increases, the print starts to sag.

Supports to the rescue!

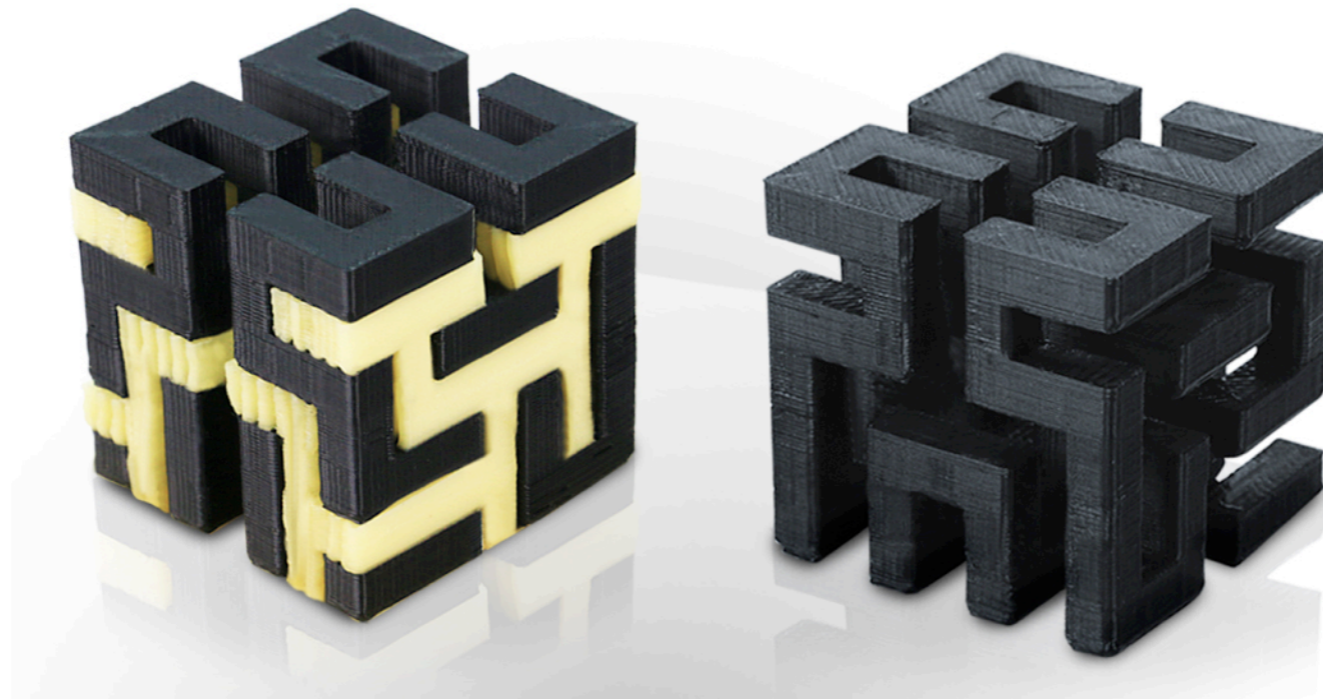


Image: AirWolf3D.com

To solve these problems, slicers will add supports (shown in yellow above). These help support the print during the printing process, and are broken away after printing. Above, the yellow material is AirWolf3D's water soluble support material. These supports don't need to be broken away. They can be dissolved in water.

Slicers

There are tons of slicers out there. Some are free or open-source, and some are commercial.

- Slic3r (Open Source; Free; <<https://slic3r.org>>) is a free program with a native Mac app, and offers a GUI interface.
- Cura (Ultimaker; Free; <<https://ultimaker.com>>) is another free option with a native Mac app and GUI interface.
- Simplify3D (Simplify3D; \$149; <<https://simplify3d.com>>) is a commercial option with a native Mac GUI app. This is my preferred slicer after trying many others.

Which slicer should I use?

- My opinion is to start with Slic3r or Cura while learning the basics of 3D printing.
- I started out with Slic3r but switched to Simplify3D when I had trouble successfully printing a very small gear.
- Simplify3D seems to work great for me, and the supports it creates break away very easily. I'm a fan.

Slicer options...

When you bring a 3D model into your slicer, you'll need to set up a few parameters.

- If your printer has dual extruders, you'll need to tell it which extruder to use.
- You'll have to specify your print temperature. An advised range will be on the packaging for your filament, and this is filament-dependent.
- If you have a heated bed plate, you'll set a temperature for that, and depending on the slicer, you might have options to change the temperature after a certain layer.

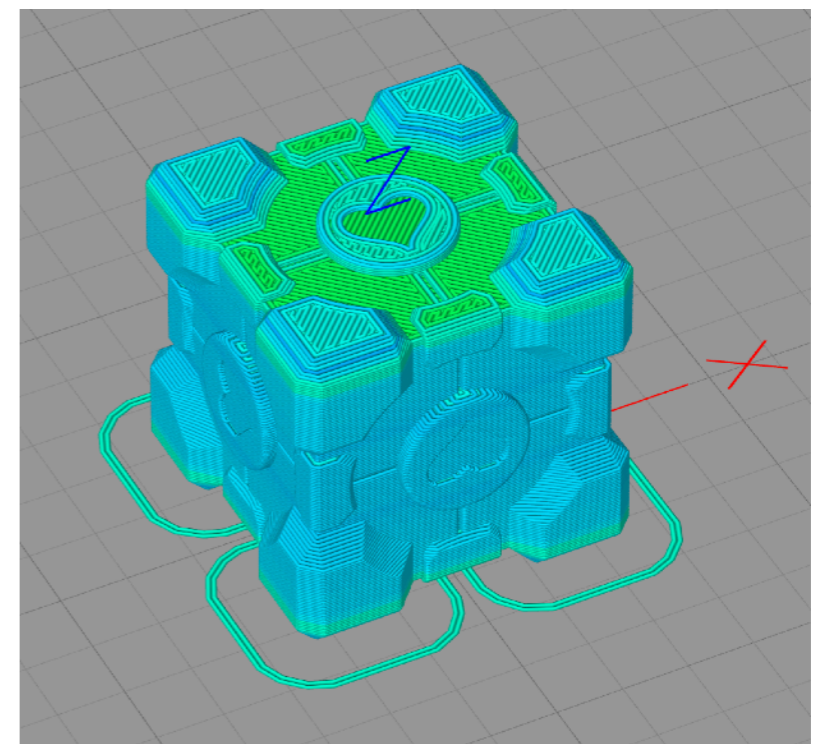
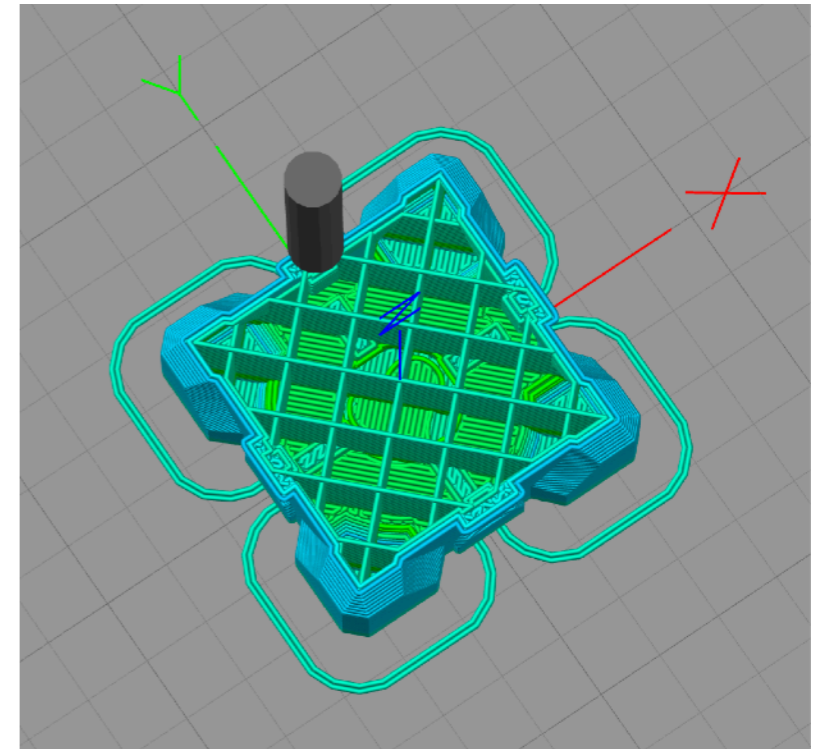
Slicer options...

- You'll be asked for your filament diameter. It is best to use a caliper to get an accurate reading of filament diameter in a few places along the filament and average them together to use that value.
- This value directly affects the amount of material extruded during printing.
- If you don't have calipers, use the diameter of the filament (usually 1.75mm or 3.00mm), but this is the first setting to check if you have under- or over-extrusion problems.

Slicer options: Infill

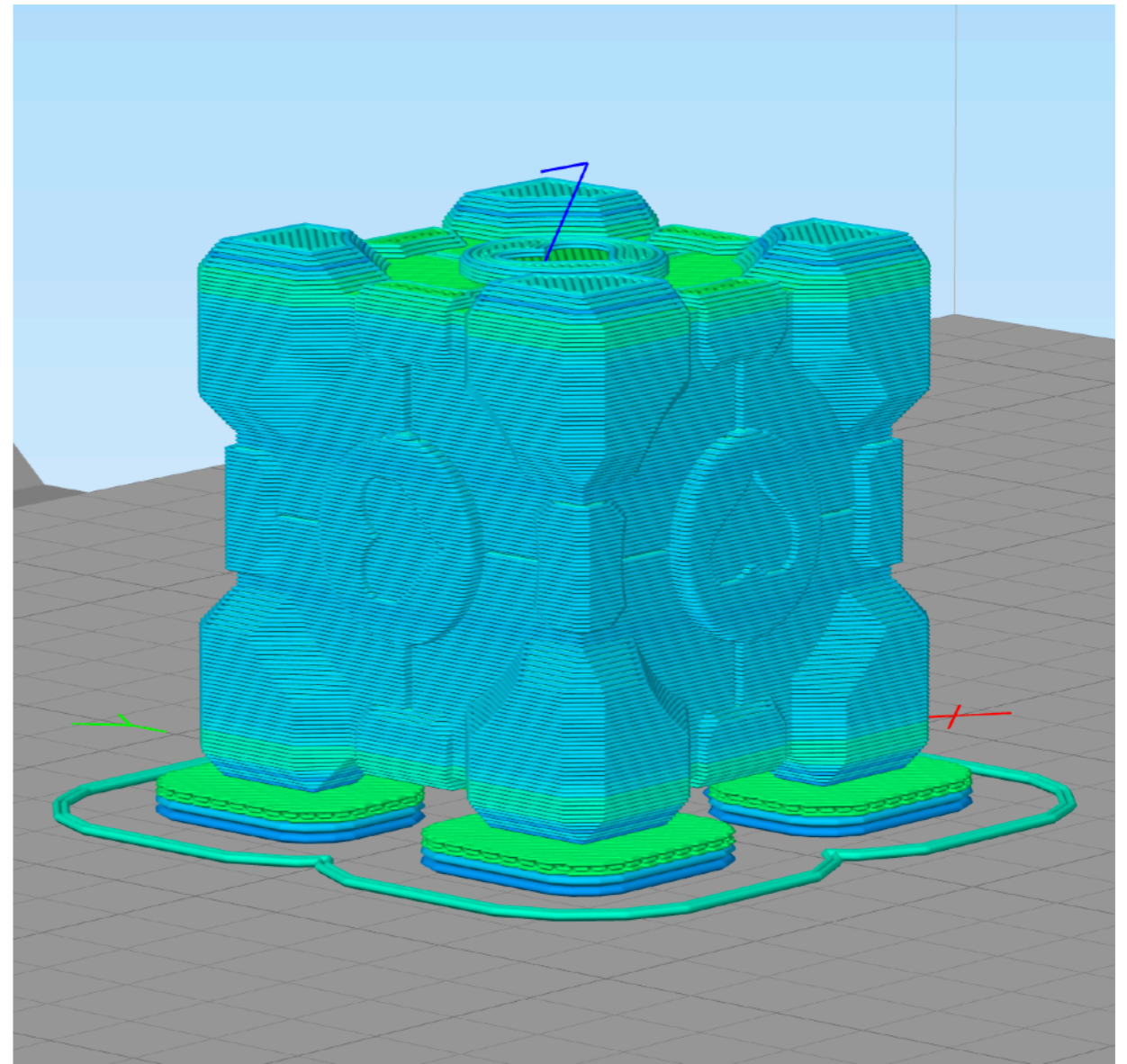
One of the most important options in the slicer is infill.

- Your 3D printed object won't be completely solid. The infill percentage controls the amount of material inside the "solid" parts of the model.
- Infill correlates to the strength your 3D printed part will have.



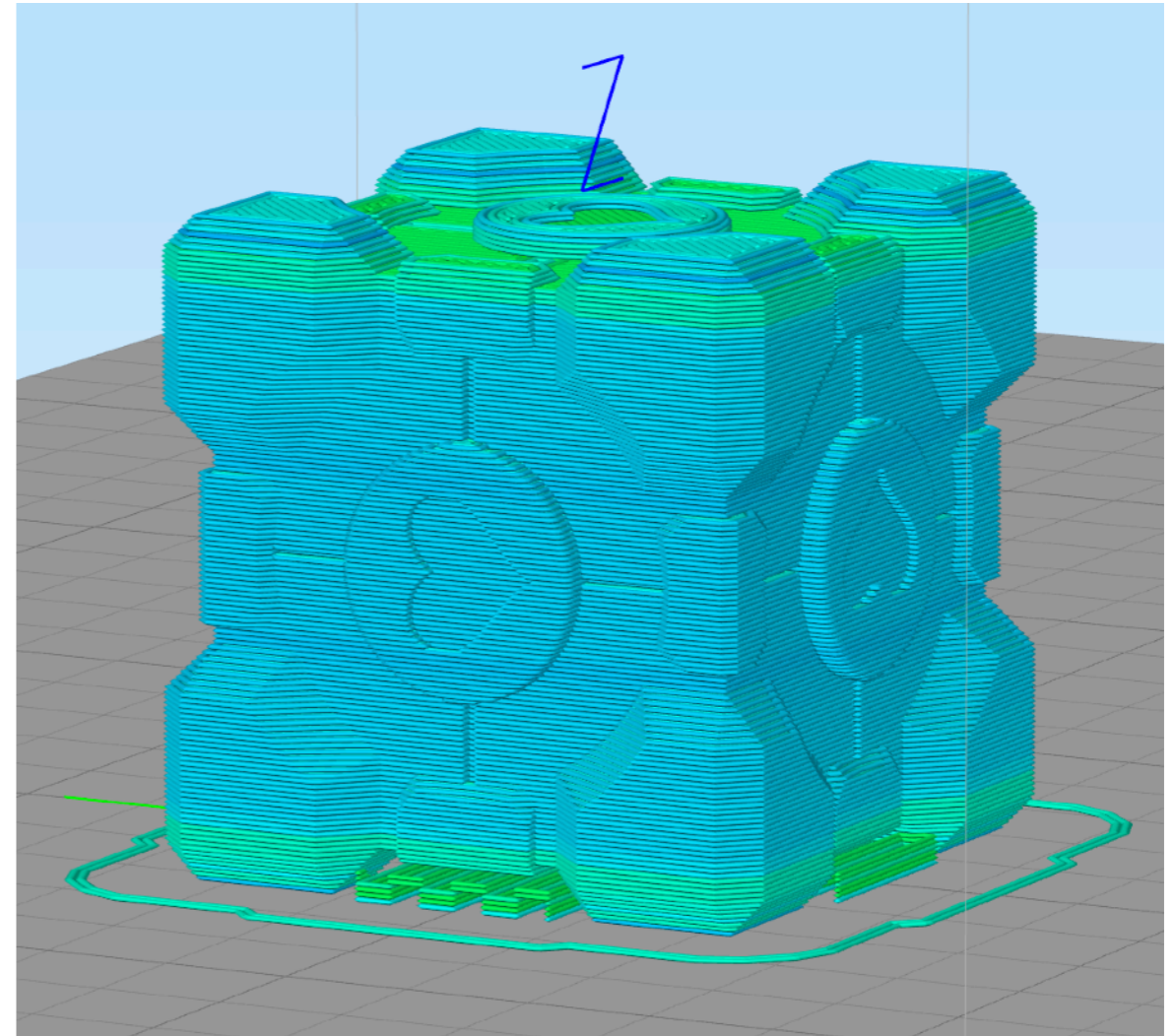
Slicer options: Rafts

Rafts give your 3D print a 3D-printed surface to start from instead of starting from the bed plate. This may give you a better surface finish for the bottom of your model and help with first layer adhesion.



Slicer options: Supports

- One of the most important settings in your slicer are supports. Supports will help keep your overhangs and spans from drooping, ruining your print. If you anticipate problems, turn supports on in the slicer.



From slicing to printing...

- After you've finished setting up your slicer, you'll use it to generate toolpaths for your printer. The toolpaths form the commands the printer will receive to move the extruder and bed plate around, turn on heaters and maintain temperature setpoints, run fans, and more.
- Your printer may have an application you need to use to drive it over USB, or you might need to save the toolpaths to an SD card and select the file from the printer's LCD.

Once you press "Go"...

- After you've loaded your toolpaths into your printer and give it the command to start the print, it will begin to preheat the extruder and, if you have one and are using it, the heated bed plate.
- After the extruder preheats, parts of the printer will start to move around. The printer will begin extruding at the first layer.
- *Watch the first layer carefully.* If your print is going to fail, there's a great chance it'll fail on the very first layer. Make sure you have good adhesion and the filament isn't curling up around the extruder or floating off in space.

During printing...

- Keep an eye on your printer. Don't leave it unattended. House fires are bad.
- If you notice a problem with the print, cancel the print from the printer's control panel.
- Don't stick your hands around the printer during printing. The extruder is very, very hot (twice the temperature of boiling water or more). If you stop the print and need to clear filament that is stuck to the extruder, use a tool, not your hands.

After the print is done...

- If you've used a heated bed plate, wait a while after the print is done for it to cool. This will make separating the printed part from the bed plate easier.
- Break off any rafts or support material.
- Enjoy your 3D printed part!

If things don't go well...

- During your first few prints, you'll be tweaking settings in your slicer and printer to get the best results. Follow Simplify3D's [print quality troubleshooting guide](#). It has tips applicable to all slicers, not just Simplify3D.
- Ask for help online. Reddit's 3D Printing subreddit and the forum for your slicer software or specific printer are great places to ask for help. There may also be a subreddit specific to your printer if it is a common model.
- Keep trying to make improvements. Don't give up!

Question and Answers