

3d Printing Guide

Welcome to the basics of 3D printing! 3D printing is a fun and rewarding hobby that can be enjoyed by people of all ages. It is a great way to get creative and make your own custom objects.

3D printing is a process that uses a computer to create a physical object from a digital model. The model is sliced into thin layers, and then a 3D printer prints each layer one at a time, building up the object from the bottom up.

The learning curve for 3D printing is relatively shallow. With a little bit of practice, anyone can learn how to use a 3D printer. However, there is a bit of a tradeoff between ease of use and cost. The more expensive printers are typically easier to use, but the less expensive printers require more tinkering and maintenance.

In general, you can choose two of the following three options: cheap, easy to print, and high-quality prints. If you want a cheap 3D printer, you will need to be prepared to put in some time and effort to get it working properly. If you want an easy-to-use 3D printer, you will need to spend more money. And if you want high-quality prints, you will need to invest in a more expensive printer and use high-quality filament.

Despite the learning curve and the tradeoffs, 3D printing is a rewarding hobby that can be enjoyed by people of all ages. With a little bit of effort, you can create your own custom objects, from toys and jewelry to functional parts and prototypes!

There are two main types of consumer 3D printers:
Filament (FDM) printers and Resin (SLA) printers.

Filament (FDM) printers use a heated nozzle to melt plastic filament, which is then extruded onto a build platform. The filament is typically made of ABS or PLA plastic, and it comes in a variety of colors and finishes. Filament printers are relatively inexpensive and easy to use, and they have a large build area. On the downside, the prints from filament printers can be less precise and have visible layer lines.

Resin (SLA) printers use a liquid resin that is cured by ultraviolet light layer by layer. The resin is typically made of epoxy or acrylate, and it comes in a variety of colors and finishes. Resin printers are more expensive than filament printers, but they produce prints with a smooth surface and ultra fine details. However, resin itself is hazardous to use, requiring special protection, and the prints require more post-processing.

How to print from START TO FINISH

(The first few steps are the same for both SLA and FDM)

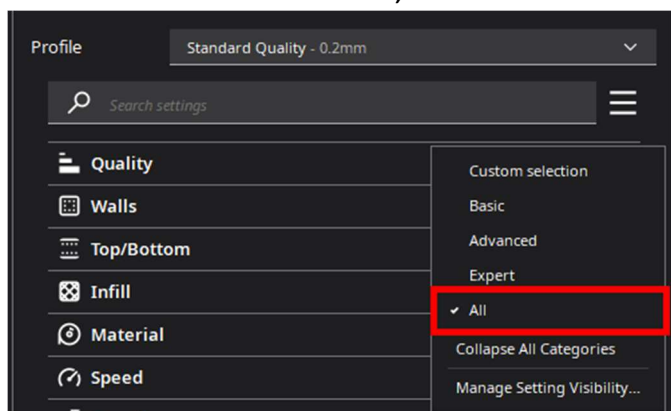
- Find/make your model
 - a. Either make a model using a 3d software such as Blender or TinkerCAD and export to stl/3mf file format
 - b. Alternatively, you can also get thousands of free files online from sites like Thingiverse, Cults3d, Printables, many others.
- Download slicing software
 - a. Cura is a refined and beginner-friendly slicer with many features that can still be used as you advance in the hobby
 - b. Prusaslicer is another very popular slicer with great features to fine tune your print, but is slightly more complex to start out with
- Slice your model (guide below), and after previewing your print, save the gcode to your sd card or usb drive
 - a. ALWAYS preview your print before saving, doesn't take much time and can save you a huge headache later if it didn't slice the way you wanted
- Level your printer bed (not totally necessary but a good habit to do this before any print)
- Preheat your printer bed (FDM) – leave it for 5 mins after it reaches the printing temperature so that the heat can spread evenly across the print bed
- Choose your gcode in your printer and start your print!

Cura Slicer Guide

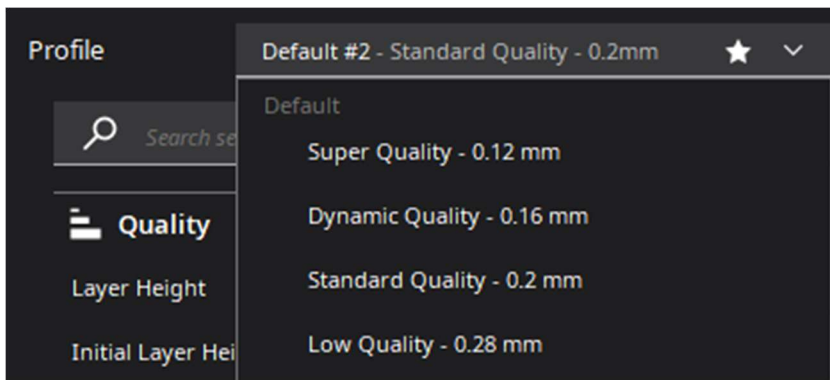
(SLA guide coming soon!)

The very first thing you will want to do with the Cura software is to unhide all the settings, as Cura hides most of them for beginners, but some are very important to have.

At the top right of the main toolbar in Cura, you will want to click on the settings dropdown, click the three lines menu button, then choose “all”

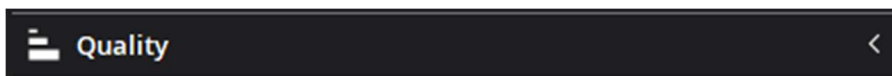


Profile:



Our preference is .2mm, or ‘standard’ quality for most normal prints, and 0.16, or ‘dynamic’ quality for higher quality models and stuff. This is setting your layer height. We would recommend starting with a default option, adjusting settings you know you are always going to use, then at the bottom of dropdown, ‘create profile with current settings’ to save and always use that instead of the in-built ones.

Quality:



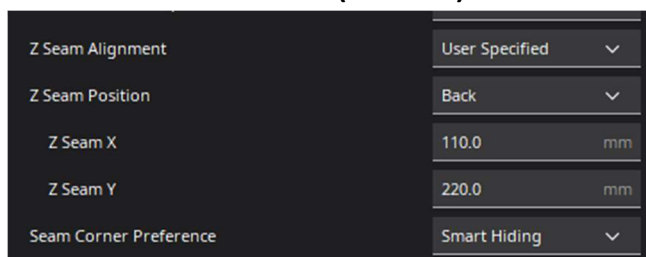
Don’t have to touch much here unless you’re really trying to tune a print, most of the time it’s calculated automatically based on the profile you choose.

Walls:



Thicker walls do more for strength than more infill for big prints. You want to have wall thickness be a multiple of the layer height in quality. Wall line count will be adjusted automatically. Default is usually fine.

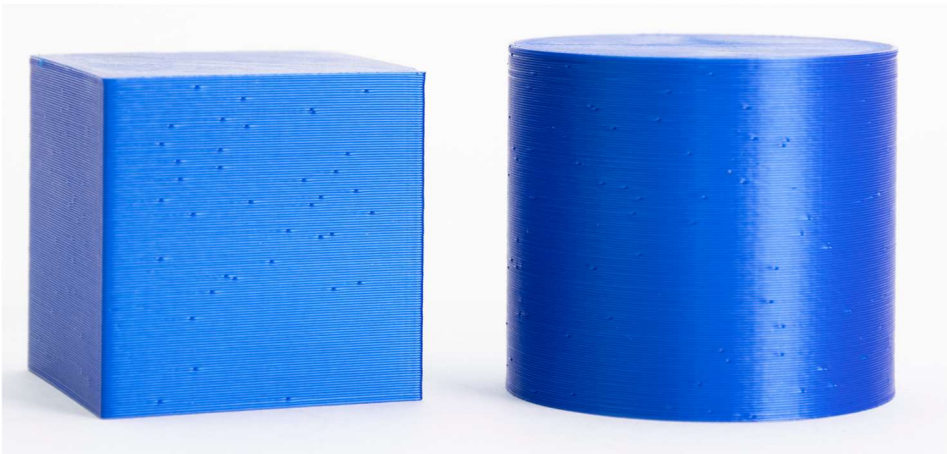
Z-seam (Walls):



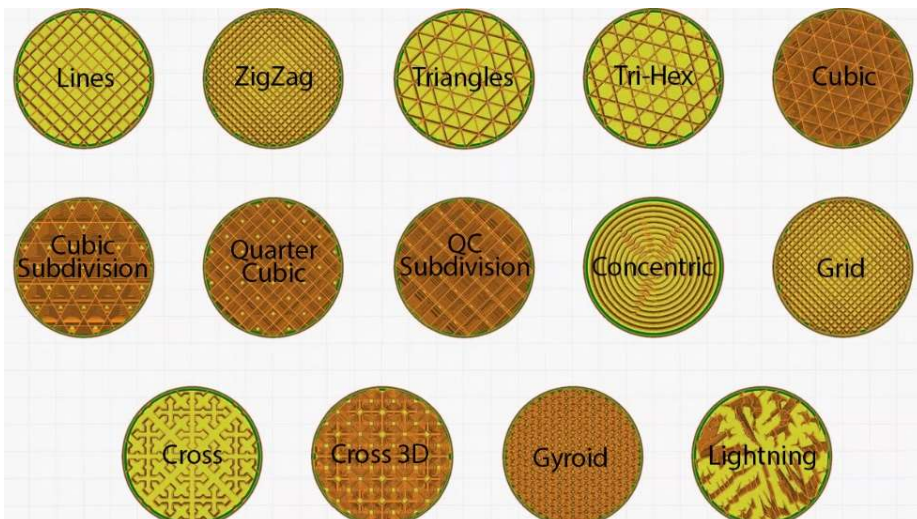
This is a very important setting, if you look at image here:



You can see the line, which is where each layer started. You can specify it in a specific place, but I usually leave corner preference at smart hiding and seam position at back (depends on print orientation) Relative is for printing a bunch of models when you want them to look the same; it will put the z seam in the same position for all the models on the bed. You can set alignment to random to have each layer start randomly, but layer adhesion must be tuned well, or you'll get spots all over the print (example):



Infill:



This is the most important setting alongside fine tuning supports. Infill sets the inner strength of print and makes it more structural. Anything above 35% is usually not needed. 10 – 20% infill is usually a good range unless you want the piece to be to be structural. In this case you can use 99 or 100%, but this will increase print time and filament use significantly. Using ‘denser’ infills like gyroid usually means you can use less % infill.

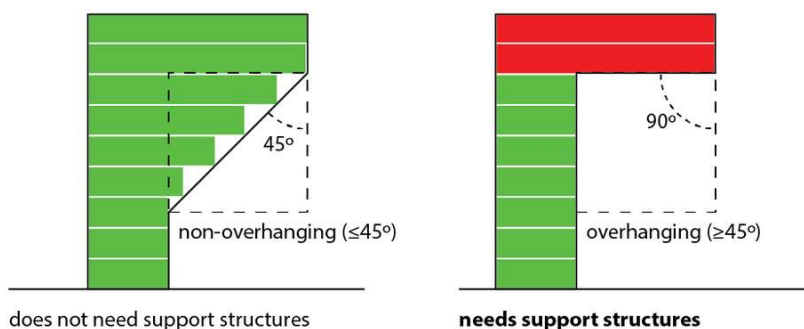
Gyroid is usually strongest, cool looking, and is one continuous volume so you could fill it with resin if you wanted. Cubic (top row rightmost) is usually fastest, along with zigzag, but slightly weaker. Concentric and Cross are made to be purposefully weaker for creating ‘flexible’ prints. Most others are unnecessary, but you can experiment as you wish.

Material, speed, travel, and cooling:

These can usually be left alone, unless you really want to tune some stuff while printing. Recommended temps for PLA are 200° for the Nozzle and 60-65° for the print bed. Increasing temperature can help with bed adhesion but has diminishing returns; if the bed is too hot, the plastic won’t be able to cool down enough to stick properly. Sometimes increasing retraction in travel can help with stringing.

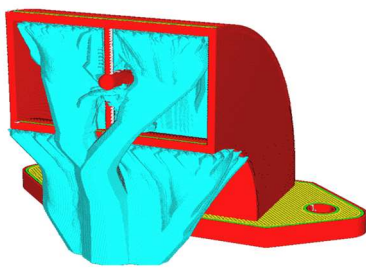
Support:

One of the most important settings for more complex prints. Supports help the printer with large overhangs:

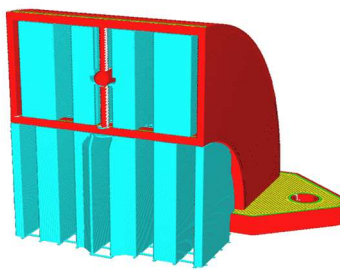


There are two types of FDM supports- normal and tree/organic.

- Normal supports are easier to print and less likely to fail, but much harder to take off. They are also much less efficient in terms of filament usage.
- Tree supports are incredible new supports that automatically determine the best areas that need to be supported while using the least amount of extra filament.



TREE SUPPORT



NORMAL SUPPORT

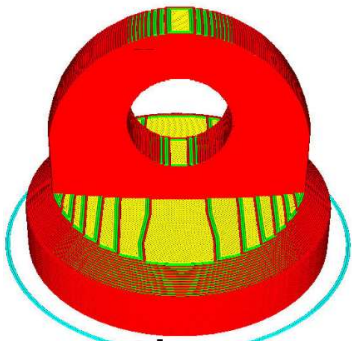
Support Overhang Angle(Support):



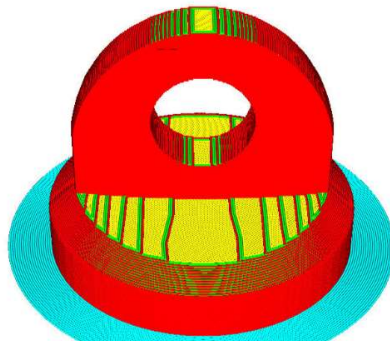
Support overhang angle is the most changed value, adjusting how far out the overhangs can be before needing supports. 45 is the lowest you should need to go, but we usually recommend 50-55° so that there are not too many to deal with. With prints that you're more confident in, but the slicer still thinks you need supports, you can adjust up to like 65-75° to get very minimal supports.

Build plate adhesion:

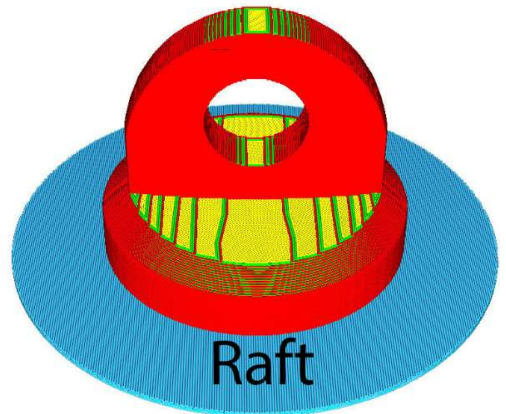
There are three types of bed adhesion. Skirt, brim and raft.



Skirt



Brim



Raft

Skirt – Thin loop of filament usually 2-3 lines thick

Brim – one layer of plastic all around the base of the print – usually 15mm margin by default

Raft – Four-layer base that completely covers the build plate underneath the print. The model will be printed on top of a smooth layer of plastic instead of directly on the build plate.

The more you use adhesion, the better the model will stick to the build plate but it will use more filament and take more time. Rafts on bigger prints (over 6-7 inches in diameter) take 1-2 hours by themselves. Skirt is shortest, for use when you're confident with adhesion. It is usually best to use at least a skirt to make sure the

bed is level before the print starts, and to remove old filament or clumps on nozzle. No reason to set to none, skirt is fast and minimal filament loss.

Ignore dual extrusion and mesh fixes - defaults are fine.

Special modes:

Spiralize external contour/Vase mode:

Good for simple, hollow big prints, usually the model will say if vase mode is preferred. This will set wall width to nozzle width usually, as walls will be one layer thick. This mode will print the whole model in one smooth movement, a large spiral with no z-seam. Prints fast, and you can usually set print speed higher on the printer during the print, which will adjust everything you need, extrusion, speed, travel, etc.



Experimental:

Usually do not touch much here unless you know exactly what the setting does, but 'fuzzy skin' is fun to play around with to add texture to your prints (example)

