3D Printing 101

Learn about how 3D printing works, and how its capabilities and limitations affects your models and designs

Wrong tutorial? See our full list of self-guided tutorials!

Introduction

3D Printing is an amazing technology that brings the power of manufacturing in the hands of everyday people. However, these printers aren't magic: they work according to scientific principles, which means that some objects print better than others. An understanding of how printers work will help you design and build more effective objects.

This tutorial is a "reading" lesson. The information you learn in this lesson will help you be a better "maker."

Contents

How 3D Printers Print

3d printers are pretty simple in concept. Every printer is loaded with a type of material, called filament, wound around a spool. This filament is heated just until it becomes gooey, and can be squirted through a tiny nozzle. The nozzle is attached to motors which move around and extrude the filament into specific patterns. As soon as the material comes out, it cools enough to turn solid.

To print a 3D object, printers lay down layer after layer of material. You can imagine it like building a house out of Legos: you start with the bottom layer first. When the bottom layer is done, you add a second layer, then a third, etc., until you have built the entire house. A 3D printer does the same thing, except instead of using Legos, it extrudes the material like toothpaste out of a tube.

3D printer filament is usually a type of plastic, either Polylactic Acid (PLA) or Acrylonitrile Butadiene Styrene (ABS). Some printers can print in other materials as well, such as dissolvable plastics, metals, or even chocolate.

At HHS, we stock PLA in a variety of colors. A spool of PLA is loaded into the printer, and the filament is fed into a nozzle. While printing, the nozzle heats to 230°C, which is just enough to soften and partially melt the PLA. At the same time, the filament is also being pushed through a 0.4 mm hole in the nozzle , so it comes out in a streamabout the thickness of the lead in a mechanical pencil.

By moving the nozzle around, the printer can very precisely draw an accurate layer of PLA. When the layer is complete, the nozzle moves up a tiny amount, and it started to work on a new layer.

Slicing

When you design a 3D model in Tinkercad, you design in terms of boxes, cylinders, spheres, and other geometric shapes. You don't worry about "layers" like a printer does. That's why 3D
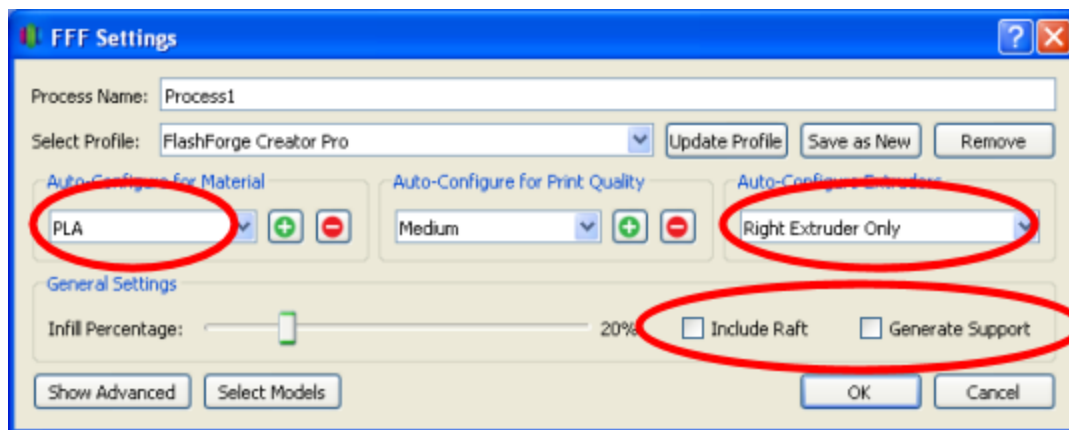
printers come with special printing software. This software takes your model and "slices" it. Slicing is just what it sounds like: it takes your model of a kitten, and figures out what each layer will look like from bottom to top.

At HHS, the printer software is called Simplfy3D. Simplfy3D is pretty smart, so you don't have to worry about slicing -- it happens automatically when you prep your model for the printer.

Printer Settings

Because Simplify3D is so powerful, it controls most of the fine details of nozzle temperature, motor speeds, cooling fans, etc. However, there are some settings you need to understand, so that your model comes out the way you want it. If you completed Tutorial #1: Intro to Tinkercad and 3D Printing, you adjusted some of these settings when you clicked "Edit Process Settings."

Filament Material ("Auto Configure for Material" option)

The printer needs to know that you are using **PLA filament**. This should be the default, but you should always check. By knowing you are using PLA, the printer will heat the nozzles and the print bed (the platform the object sits on) to just the right temperatures.
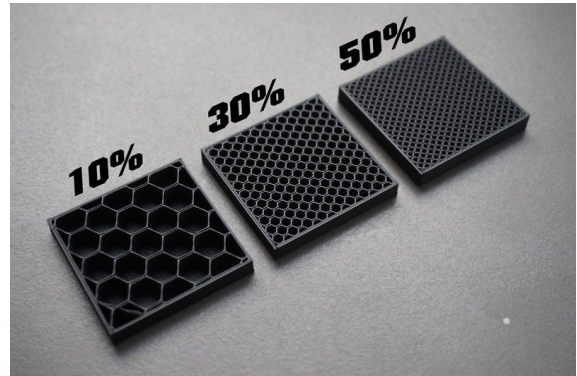
Nozzle ("Auto Configure for Extruder" option)

The printers are HHS are FlashForge Creator Pros. These are "dual extruder" printers, which means it can have different filaments loaded into each of two different nozzles. Ideally, this gives the printer the ability to print two colors at the same time, but this is *very* complex to model, and will be held off until a later, more advanced tutorial.

For now, you just need to tell the printer which nozzle and spool to use, based on which color you are printing. In the back of the printer, you will see two spools. Be sure to set this to whichever spool (**left or right**) is holding the color you want.

Infill ("Infill Percentage" option)

It is usually inefficient and wasteful to print solid objects. Instead, the slicer part of Simplify3D makes your model a hollow shell, with widely spaced internal supports added in. When done, the object and feels looks solid, but it actually has a lot of space inside.
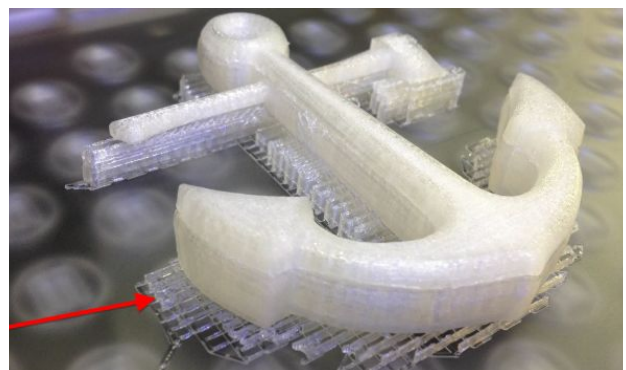
These automatic internal supports are called infill. The default for infill is usually about 15-20%, which means 80-85% of the inside of your model will be empty space. In most situations, this is perfectly fine. However, if you are printing something that has to be both very thin and very strong (like a plastic screwdriver?), you might need to bump the infill percentage up.

Raft ("Include Raft" option)

A raft is a large, flat, smooth set of layers that is printed below your first layer. The raft is designed to help your model stick to the print bed. Sticking to the bed is critical. If your model starts to slide around the bed, stuck to the nozzle, then instead of a beautifully printed little penguin, you will get an enormous ugly blob.

How well your model sticks to the bed depends in part to how much if it touches the print bed. If you were printing, say, a rectangular box, you'd have a big flat section touching the bed, and it will probably stick fine. On the other hand, if you were printing a miniature coffee table, with only four little table legs touching the bed, those legs could easily become unstuck. Printing a raft first gives those legs a layer of plastic to stick to.

If your model has a large flat area on bottom, you probably don't need a raft. If it has only a small area touching the bed, you might need to include a raft.
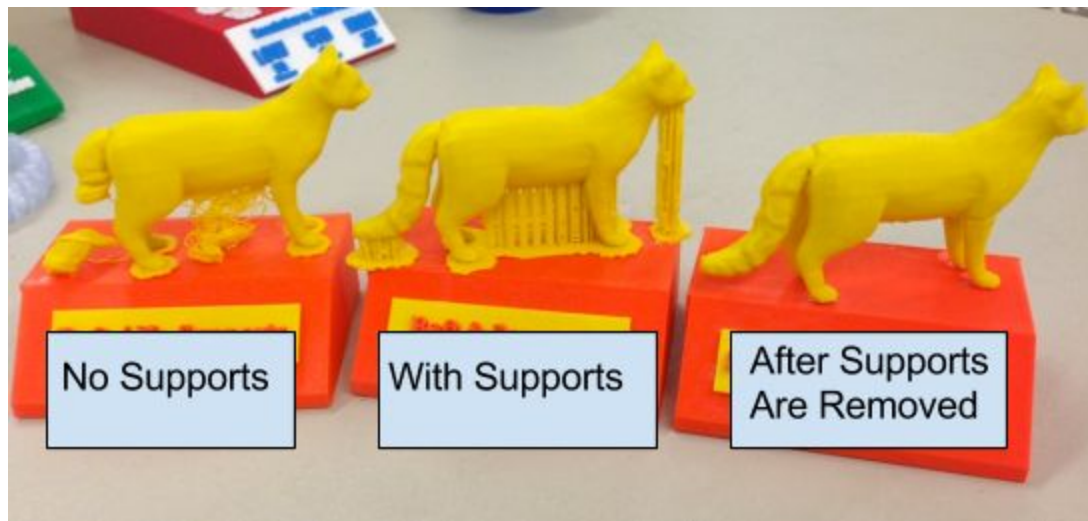
Rafts are constructed to break off of your model easily after you remove it from the printer.

Supports ("Generate Supports" option)

Imagine that you want to print a big, vertical capital H. The printer will print the bottoms of the each leg, layer by layer, rising up from the bed like two vertical columns. When the printing is halfway up, It is time to print the crossbar of the H, connecting the two columns. And here's the problem.

How does the printer print across a big gap of air? If it tries, the filament will droop into the gap, and make a mess. The printer actually can bridge small gaps without much problem, but large gaps just won't do. That cross-bar needs supports.

The photo below illustrates this well. The cat on the left was printed without supports. Look how the tail and stomach fell apart while printing. The cat in the center has supports added in, and the cat on the right is the same thing, but after the supports were removed. Much better!

Simplify3D can intelligently add supports to your object. These supports get added in as wiry, easy-to-remove scaffolding to hold up pieces of your model. The downside, of course, is that you have to break off the supports afterward, and that might leave little rough edges on your final product.

By the way, what's a better way to print that big capital H? Print it lying on its back, of course - then no supports are necessary!

Designing with Printing in Mind

Understanding how 3D printers work provides us with a few things to keep in mind when printing:

- **Plan your sizing**.
  Our FlashForge Creator Pro printers can print an object that fits in an 8.9 x 5.7 x 5.9 inch box (225 mm wide, 145 mm deep, 150 mm tall). Also, be aware that larger objects take a longer time to print. An object the size of a AA battery will take about 45 minutes, and an object the size of a Rubik's cube will take closer to 3 hours.
- **Avoid overhanging pieces** (when possible).
  Sometimes this can be accomplished by printing the object on its back, side, or upside down. If overhangs can't be avoided, you will have to either let Simplify3D generate supports automatically, or design your object as multiple pieces that you can glue together later. For example, a simple ping-pong ball-sized sphere is notoriously difficult to print well, because the bottom third is all very steep overhangs. It is very common in 3D printing to instead print two hemispheres and glue them together afterward.
- **Rotate your model for printing**.
  Before exporting your STL file for printing, consider whether rotating your model different directions will help reduce the need for supports or a raft.

Phew! Nice job learning about 3D printing! Now that you know about how the printers work, let's put your skills together and design something from scratch! Head over to the next activity, [Challenge A: Personalized Keychain](#)!