

Beginner's Guide to 3D Printing

by Tyler Hudson

The first step in getting started with 3D printing is to decide on a printer that meets your needs. Many companies sell material extrusion (FDM-like) 3D printers, and this guide highlights four of them.

The following table shows some key features that customers should consider when choosing a printer. Build platform size is important for users because it limits the size of the parts that can be created. Many users like having a heated build platform and full enclosure because it keeps the part(s) heated during the build process, which can lead to higher quality builds. Leveling is an important feature to consider because it is critical to a successful print. Layer height refers to the minimum layer thickness that a printer can produce. The thinner the layer, the better the surface finish.

	Makerbot Replicator Mini	Solidoodle 4	UP! mini	LulzBot Mini
Price	\$1,375	\$599	\$599	\$1,350
Build Platform Size (mm)	99 x 99 x 124	203 x 203 x 203	121 x 121 x 121	152 x 152 x 157
Heated Build Platform	no	yes	yes	yes
Full Enclosure	yes	yes	yes	no
Leveling	not required	manual	manual	auto
Minimum Layer Height (Microns)	200	100	200	50
Available Filament	PLA	ABS, PLA	ABS, PLA	ABS, PLA, HIPS, PVA

Once you pick a printer that suits your needs, determine how you will obtain the 3D computer models for the parts you wish to print. Models can be found online on sites such as Thingiverse or GrabCAD. These sites allow you to search for models created by other users and download them to print on your machine. Some printers will accept STL and OBJ files, with STL being the most common file type. Other printers require g-code, which is standard for CNC machining. G-code provides step-by-step movement instructions for the printer. Many free g-code generators are available, and one popular generator is SLic3r. It works by allowing you to input your STL file and then the software outputs g-code for your printer.

Another route is to create the 3D model yourself. This can be done by using free modeling software such as SketchUp, 123D, Wings 3D, or Tinkercad. These programs are made for consumer use, with free versions available. Commercial products, such as SolidWorks, Creo, and AutoCAD, are available for purchase if you prefer software that is more advanced with more features.

Tips for advanced designing: Be wary of tolerances. From personal experience, I like to add around 0.004 to 0.006 inch to my dimensions to account for the width of the bead deposited by the 3D printer. The printer follows the exact shape of the model, so more material will be put down than required because the nozzle of the printer is very small. This is similar to how you need to account for tool diameters when using a CNC mill.

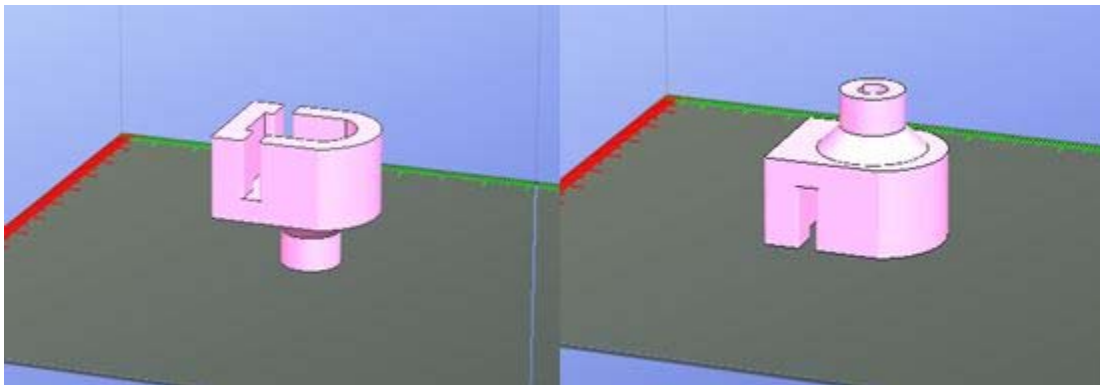
Getting started

Suppose you have purchased a 3D printer and you have a 3D model or two ready to print. Some printers will require some assembly, but instructions should be provided. It's best to set up the printer on a flat surface in a room that is well ventilated because a buildup of ABS fumes can be toxic.

Next, you need to make sure the build platform and nozzle are level. Each printer is different, but most printers will have screws or some sort of device that allows you to adjust the platform. Leveling the platform is one of the most important tasks to ensure quality builds. If it is not level, the plastic might not adhere to the platform, or the nozzle could crash into it. (A heated build platform helps to ensure adhesion.)

Next is to set the nozzle to the correct height. It varies from printer to printer based on printing material thickness, but it should be low enough for the material to be flat against the platform as it comes out. This means that the nozzle should be closer to the platform than the diameter of the nozzle. For example, if your printer has a nozzle that is 0.4 mm in diameter, the nozzle height should be less than 0.4 mm above the center of the platform.

Once the build platform is level and the nozzle height is set, it is time to import a 3D model. One can determine an almost-endless number of ways to orient a part, but only a few ways usually result in a good quality build. Ideally, you want to place the model in a way that will have the least amount of overhangs, and thus the least amount of support material. If you do have overhangs, it is best to make them exterior overhangs versus interior overhangs. An example is shown in the following.



The image on the left shows a good orientation for this part because it has an exterior overhang instead of interior like the image on the right. Exterior support material is much easier to remove than interior.

If the part has any critical features, the part should be oriented to minimize the amount of support material touching these critical features. Once you build a couple parts, these ideas will become more apparent to you.

Most 3D printers have options that you can change for the support material. For instance, you can define the number of support layers before the actual part itself begins to build, the angle of overhang, distance between support structures, and layer thickness. There is no absolute best choice for these options. You need to experiment with them until you find a combination that works the best.

Printing

After you have the model oriented in the best possible way, and the support structure options are set, you are finally ready to print. Once a build starts, it is best to not interrupt it because it can cause poor bonding of layers. That being said, a build can be paused once a certain number of layers have been deposited.

A part build can fail in several different ways during a build. One of the most noticeable is when the first couple layers do not attach well to the build platform. This is easily remedied by making sure the platform is level, nozzle height is correct, and platform is fully heated, if so equipped. Another common failure mode is warping or delamination of larger parts. This is usually caused by uneven heating throughout the build. An easy way to alleviate this is to enclose the printer, which traps some of the heat and allows the part to cool more slowly.

An additional possible cause of failure is feed entanglement. This can happen when the material remaining on the spool overlaps itself and binds, thus not allowing the printer to extrude material. This most commonly happens when the spool is running low and can be corrected by re-wrapping the spool or by switching to a new spool. If you notice that a build has failed for any reason, it is best to stop the build process as soon as possible in order to save material and restart.

Post Processing

Once the print is finished, the support material needs to be removed. The best way to do this is to remove large chunks at a time. Sometimes, it is possible to remove the whole support structure in one piece. If you have trouble removing the material, sharp instruments and needle-nose pliers can be helpful.

When the majority of the support material is removed, it's time to give your part a nice finish. Some sandpaper (100-220 grit) can be used to sand whatever support material is remaining and help give the part a smoother appearance. You can also dip a rag in acetone and rub it over the part to smooth out the transition between layers, assuming the material is ABS. This should be done carefully because too much rubbing can ruin the surface of the part.