ALTERNATIVE TYPES OF PRINTING

There are 3 main types of 3D Printing – FDM, SLS and SLA. Within this handbook we will be discussing FDM, also known as Fused Deposition Modeling.

FDM

FDM works by laying down consecutive layers of material at high temperatures, allowing the adjacent layers to cool and bond together before the next layer is deposited. The resolution of FDM rapid prototyping is limited by the accuracy of the motors and user calibration as well as the print time available. Print time increases linearly as part tolerances become tighter.

FDM print tolerances range from 0.05 mm to 0.5 mm with the industry average coming in at around 0.2 mm. Some interesting benefits of fused filament fabrication include the ability to easily customize infills of the objects being printed. This means that it is very simple to print prototype models for fit and finish checks with low infill, or even hollow, in order to save on material costs.

SLA

Stereolithography (SLA) is another example of building a model layer by layer, but instead of extruding molten plastic through a nozzle, the process requires a liquid plastic resin called a photopolymer which is then cured by an ultraviolet (UV) laser. This process is even more comparable to a CNC operation as the SLA machine requires an excess amount of photopolymer to complete the print. An SLA machine typically stores the excess photopolymer in a tank below the print bed. As the print process continues, the bed is lowered further into the tank curing each layer consecutively along the way.

Once the final layer is complete the bed is raised out of the tank, exposing the solid 3D model. SLA has a higher resolution but longer print time than FDM

SLS

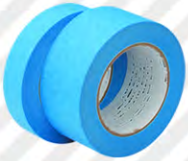
Selective laser Sintering is a very exciting technology which directly fuses particles together layer by layer through a high energy pulse laser. Similar to SLA, this process starts with a tank full of bulk material, but this time in powder form. s the print continues, the bed lowers itself for each new layer as done in the SLS process. Overhangs are naturally supported by the excess bulk powder material making those features much more simple to print than using FFF. The bulk material is typically heated to just under its transition temperature to allow for faster particle fusion and print moves. SLS is the preferred rapid-prototyping method of metals and exotic materials, however, there are additional costs in energy used for fabricating with this method which may reverse any savings realized in the material cost.

Bed Adhesion

Getting a print to stick to your bed can be a difficult task at times. Not only do many materials require a heated build plate, but some even require additional adhesions.

Glass Bed

Glass Beds have a very smooth surface which is preferred to acrylic. Due to the smooth the surface is, additional products are required for adhesion. We will often use hairspray to help. Glue sticks and other similar mixtures have been tried, but these often make the print too stuck to the bed, cuasing the removal of the part to be very difficult. If you do not have a heated bed it will be very difficult to only use hairspray and not experience any warping or lack of bed adhesion. Often individuals will use blue painters tape as a means to get the part to stick to the bed. We have done this in the past but the bottom surface quality is never as good. This also requires the tape to be replaced frequently.



PEI Bed

Polyetherimide (PEI) requires no additional adhesives such as glue or tape, but does require a glass bed to be attached to. PEI is a low surface energy polymer with a high glass transition temperature which resists deformation throughout successive heating and cooling cycles. It is used as the top surface to the build platform in order to enhance adherence to the bed along with the ability to cold release many 3D printable materials without resorting to destructive methods. Glass filled PEI further enhances these features by adding additional resistance to structural deformation during the heating and cooling phases.

Using a Brim

A brim is a when the printer lays lays lines around the base of the print which essentially act as anchors for the entire print. The amount of lines added can be chosen in the slicing settings.

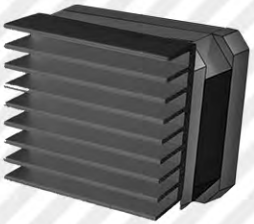
Brim can often be difficult to remove, but it is needed for any ABS print to stick to the bed properly and not warp. Since brim often adds lines to every part of the bottom layer, including holes in the middle of the print that are not needed for bed adhesion, you can actually add a close .01mm distance skirt to achieve the same function.

Other factors that can help bed adhesion is having the proper Z-Height as well as making sure the entire bed is level before printing.

COOLING

Cooling is an essential part for prints to be successful. Not only do motors and other parts need to be cooled to prevent overheating, but the barrel and print itself often require active cooling.

Stepper Motor Cooling



While these motors are designed to run hot, there is a limit to how hot you want them to get for prolonged periods. One of the simplest ways to cool your components without disrupting the operation of the overall system is to add passive heat sinks. These heat sinks will increase the heat dissipation of the individual components by expanding the area available for convective heat transfer.

Barrel Cooling

The majority of **FDM** printers will have a fan on the barrel to prevent heat creep and frustrating nozzle clogs. When plastic material is being melted to extrude through the nozzle, that heat will often travel up the plastic filament strip. This heat will cause the filament to expand. If this expansion occurs in the middle of the barrel, the filament will get a large bulge, one too large to pass into the hotend or to be pulled out of the extruder. This often leads to a clog that requires the disasembly of the extruder. To prevent this, a barrel cooling fan is needed.

Active Cooling

Active Cooling refers to a fan actively blowing on the print as each layer is laid down. This process is needed for a high surface quality on **PLA** prints and similar materials This can actually be a negative to use on larger **ABS** prints due to its shrinkage rates. You have to know what material you are printing in and what settings that particular filament prefers.

Active cooling, and often the barrel fan, will require the firmware on your printer to be flashed with correct **PID** settings. Without this, an active cooling fan can actually cause the nozzle to cool - often enough to cause a print to fail.

DUAL EXTRUSION

Many printers come with the ability to use more than one extruder during the print. This can allow prints in multi-color or even dissolvable support material. While this sounds like a great addition to FDM printing, there are many headaches and problems involved.

Leveling the Nozzles

Leveling the nozzle can be one of the most difficult things involved with dual extruding. Each printer has it's own method of leveling, and some can seem impossible to perfect. Larger nozzled printers with larger layer heights are a bit easier, but a slight diference in their heights will lead to the lower nozzel knocking into the print. Some printers even require the entire hotend to be disasembled, the nozzles leveled, and then having the hotend reassembled.

Dissolvable Support Material

When dual extruding it is often suggested to use High Impact Polystyrene (HIPS) for support material. This is because HIPS is dissolvable in Limonene, a colorless liquid hydrocarbon. While we have experienced some difficulty with dissolvable HIPS, the idea is that you can increase your support structure fill amount up to 100%. This should remove all ugly undersides of prints as well as make post-processing much easier.

You can also print support in PVA (which is dissolvable in water) or in PLA which can be dissolved in Sodium Hydroxide.

Multi Color

Having dual or multiple extrusions also allows for multi-colored prints. An object can be designed with multiple colors or there are designs already available for download online.

Generally when printing with dual extrusions, priming towers and ooze shields are required in order to have a print with a good surface quality.

Extrusion Settings

Making sure your extrusion settings (or E-Steps) are correct is mandatory to have successful, clean prints. Under extrusion can lead to a print that doesn't finish and over extrusion can lead to very poor surface quality.

Under Extruding

Under Extruding will have prints with gaps in the top layers, they wont be watertight, and they will not be as mechanically strong as they could be. Overhangs are also very difficult to print when under extruding. However, you will not notice much of a surface quality problem (other than perhaps layer delamination).

Over Extruding

Over extrusion will cause prints to be too large, holes to be too tight, surfaces will look overstuffed and cause layers to be bulging out. Since the printer is over feeding the filament, each layer will likely be pushing into the layer below it. This leads to very poor surface quality prints. While this is true, overhangs are much easier to print when over extruding.

Correcting E-Steps

E-Steps refers to the amount of steps the motor should move forward when feeding filament, in a steps/mm measurement. In order to calibrate this, you will need to compare the amount of filament that should be fed through your extruder vs the amount that is actually being fed. This involves measuring out a certain amount of filament and then telling your printer to extrude that amount. You can then measure the difference between what should have extruded vs what was actually fed. Using a simple calculation can help you flash your firmware with the correct e-steps.

There is a very helpful video tutorial on this you can access by watching Thomas Sanladerer's YouTube Channel

FILAMENT OPTIONS

FDM filament and material options are growing every day. There are a vast amount of companies that now supply standard PLA and ABS, but there are now also those that produce material options with varying properties.

1.75mm vs 3mm

FDM extruders are built for either 1.75mm or 3mm diameter filaments. There is debate on which is preferred. The extruder for 3mm filament will have to turn its stepper motor less to extrude the same amount of material, due to the thicker diameter. This can lead to less grinding of the filament which will lead to less likelyhood that the filament will be grinded to the point of breaking. 1.75mm, on the other hand, is more flexible while on the spool and is easier to feed. 3mm also has a smaller tolerance percentage ranging from 2.8mm - 2.9mm (1.8%) vs 1.75mm of 1.7mm - 1.8mm(2.9%)

PLA and ABS

The 2 main types of materials used today are PLA (polylactic acid) and ABS (Acrylonitrile butadiene styrene). With very minimal shrinkage rates and a cheap price tag, PLA is the go to for parts that do not require stress. Since the shrinkage rate is so low, parts do not warp while printing PLA. This makes PLA easier to print than other filaments since it's not required to be in an enclosed environment for large parts. While PLA boasts a high tensile strength, it is also more brittle than most other materials we use with a very low impact resistance. It can crack or break when dropped and does not have many mechanical uses.

ABS has a very high glass transition temperature (105°C) which means that it requires a heated build plate for bed adhesion and to prevent warping and layer delamination. Due to the high shrinkage rates, larger prints also require an enclosed or controled environment with ambient temperatures of around 55°C. ABS is great for mechanical parts and has the ability to be acetone vapor finished.

Unique Materials

Over the past couple of years, the filament options for FDM printing have expanded to include an immense amount of choice. From flexible, to carbon fiber infused, to high heat resistant, to FDA approved - there are now options that can suite any need. The issues begin to arise when attempting to print complex models or not entirely understanding the preperations required. High heat resistant materials require an enclosed printer. Many flexible materials and nylons require a PVA Water solution be added to the bed. Some materials just cannot print parent support material. By using these unique materials and understanding the limitations, some amazing things can be printed.

G-CODE

G-code is a language in which people tell computerized machine tools how to make something. The "how" is defined by instructions on where to move, how fast to move, and what path to move.

Layer by Layer

FDM printing can be thought of as 2D printing over and over again, layer by layer. This process is done by feeding the printer a .gcode file. The printer then reads this G-code by telling the printer how to move in the X-Y direction for each layer, how much filament to feed, at what temperature to print, and how much the printer head should move in the Z-direction after completing that layer. The process is then repeated.

Slicing

In order to get a G-code, one must create a "slice" of a 3D file. This is done by using a slicing program. There are free programs such as

Cura, Slic3r, or the Makerbot program that comes with it's printers (Makerbot actually uses .x3g instead of G-code). A paid program called Simplify3D is great for personalization of settings, but it can be a bit overwhelming at first. There are numerous basic settings for each material such as temperature, speed, and layer heights and there are advanced settings for infill, support, retraction and more. Tutorials, blogs, and our complete PLA and ABS profile settings guide on our website can help, but at the end of the day, trial and error is always required. This is because different printers using different hotends with different materials made by different manufacturers which can all lead to changes in settings.

Once happy with the settings chosen, it is a matter of exporting the part as a .gcode to whatever device is used for printing (SD card, Octoprint, etc)

Simple Commands

There are some simple commands to know for easy use of your printer from the terminal

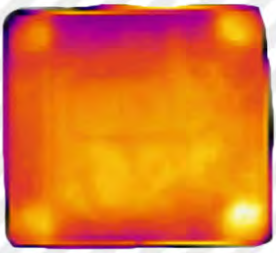
```
G0: Rapid Linear Movement (can be applied to X,Y,Z,E,F,S)
Ex. G0 X20 moves the printer head 20mm in the x direction

G92: Set Position (can be applied to X,Y,Z,E)
Ex. G92 E0 resets extruder

G28: Move to Origin (can be applied to X,Y,Z)
Ex. G28 X Z Go to home for X and Z axis

M0: Unconditional Stop

M302: Enables cold extrusion
```

HEATED BUILD PLATE

Many basic printers do not come with a heated build plate, but it is necessary to have one if you would like to print in materials with high mechanical properties.

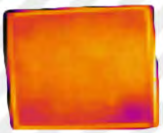
Glass Transition Temperature

The glass transition temperature (Tg) is the temperature region where the polymer transitions from a hard, glass material to a soft, rubbery material. Prints will often deform around this temperature, but it is needed for the bed to heat to just around a materials glass transition temperature in order to have good bed adhesion. You can get away with printing **PLA** on a non-heated bed, but any material with a high Tg will require a heated bed.



Temperature Gradients

Thermoplastics are very sensitive to temperature fluctuations, specifically when they are being rapidly extruded at temperatures well above their glass transition temperature. When there is a significant diference in temperature on one side of the **HBP** compared to the other, we call that a temperature gradient. These temperature gradients and environmental fluctuations are the leading factors that lead to part failures due to warping. Temperature gradients above the **HBP** can also lead to an effect known as "delamination", where the print remains adhered to the bed, but individual layers above the **HBP** surface begin to weaken and separate from each other. So if you want to have strong parts that print reliably, a good starting point would be to ensure your **HBP** is as uniform and gradient-free as possible. Since we have been researching the effects of **HBP** thermal gradients for quite some time, we decided earlier this year to take a shot at fixing the issue. We incorporated a very special bed stack into our beta **3DGenie** (**3DG**) automated **FDM** printer with the aim to eliminate significant gradients at the print suface (pictured to the right)



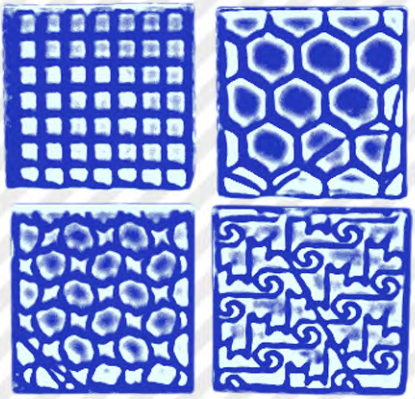
Infill

Infill refers to the structure that is printed inside of a closed object. If you wanted something printed hollow, you would print with a 0% infill. 100% will fill the entire object with material, causing the weight and time required to print to increase.

Standards

An infill of over 25% is not needed for most structures. This is one reason that 3D printing is useful in aviation. Drones and other similar structures can have strength yet reduced weight. In order to have a print with a clean top surface, larger prints will requrie an infill minimum of 10%.

Infill Paterns



Within certain slicing programs the user is able to create unique infill paterns in order to test the strength of each at different fill percentages. Decorative patterns such as Moroccan stars and Catfill show poor performance and should only be used if they are exposed and are part of the design. The real debate is between Linear, Diagonal and Hexagonal patterns. Varying combinations of infill percentage and pattern can influence strength, material usage, and print times. Sometimes, in order to improve the mechanical performance of a 3D printed part, it comes at the expense of print speed, affordability, and quality.

Test Results

Through trial and error we have found that the three most useful patterns are diagonal, linear, and hexagonal. Triangular structures can also be used to speed up the print, but when it comes to strength vs material used, those three seem to have the best results.

Jogging and Homing

Jogging your printer means that you are moving the carriage and bed along their given axis. When printing it is always best to make sure that your gears and axis are properly calibrated and sliding smoothly. This means that you should periodically check and confirm where "home" is for your printer. This can save a major headache when you prevent a long print from failing 20 hours in.

Finding Home

No matter the style of printer, each FDM machine will need to find "home" before each print. This refers to hitting an end stop in order to know where to begin. Since this is a mechanical process, it is always good to both visually and physically check to see if the X,Y, and Z axis are starting correctly.

It is also very important to make sure you have a Level Build Plate and properly check the Z-Height (as mentioned later in this handbook)

Lubricating Rods

If you are printing frequently you will want to make sure to occasionally lubricate the rods. When jogging your carriage or build plate you will often be able to tell if there are any rough areas that are not moving smoothly. You can do this with your hands when your stepper motors are turned off. If there are any areas that you can tell is not sliding right, add some white lithium grease or similar lubrication product.

Stepper Malfunctions

Each axis in a cartesian printer requires its own stepper motor. This motor can wear out over time just like any other part on your printer. Screws may become loose and the pulley may not spin the proper amount of times in what the printer thinks is a full rotation. This will not only lead to XY Axis Shifts, as mentioned later, but it may make the print too long or too short in one direction, along with the under/over extrusion issues that come along with that. Periodically check your steper motors and their pullies to see that everything is jogging smoothly.

KNOWLEDGE OF SURROUNDINGS

When we say to know your surroundings we don't mean to understand demographics or updates in the 3D printing industry, we mean to know the details and variables of the area where your printer is located. Ambient temperature, humidity and available power are all just some of the things you should be aware of before printing.

Ambient Temperature

The temperature of the air surrounding your printer can drastically change the quality of your manufactured part. As mentioned in Filament Options, certain materials such as ABS requre a higher ambient temperature to prevent warping and layer delamination. Often the ambient air in our facility can become so low that we are required to enclose racks of printers even for PLA prints.
You will need to notice when there are extreme temperature fluctuations in your building especially when not using an air conditioner or heater for stability.

Humidity

Humidity can effect the polymer you are using to the point of causing a failed print. Humidity effects the shrinkage rates of filaments which can effect the surface quality.

Ambient humdity does not only effect the filament during the print, but where you store it. Most materials want to be stored in a very low humidity area of 40% or below. We have a dehumidifier running 24 hours a day where our filaments are stored. Most filaments come in a vaccuum sealed pack which you should not open until ready to use. The type of material will depend on how much it is effected. Nylons will saturate quickly and will likely pose problems when printing if not stored in proper humidity.

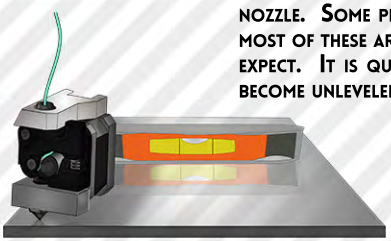
Whether you print in a dry hot climate or a cold humid climate will drastically change your setting and quality. Cold winds or a printer directly under an aircondition vent can lead to never ending failed prints. You have to know your surroundings if you ever expect to turn out reliable results.

Level Build Plate

A common printer failure will be when the build plate is not level. This actually happens so often that we recommend leveling the build plate before any long print. There are some FDM options that say they are able to automaticaly level the printer bed, but even these will require frequent calibration.

Unlevel Bed

When we say "level build plate' we mean that each corner (and center) of the printer bed is equal distance from the nozzle. Some printers claim to automatically level, but most of these are not nearly as calibrated as one would expect. It is quite surprising just how often a bed can become unleveled.

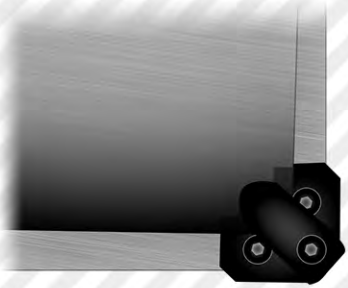


Beds that are unlevel will make large prints impossible. This is why we recommend doing a bed leveling test before any important or large print.

Thick Glass Build Plate

We have found that working with a thick glass build plate is the easiest and most user friendly. Aside from thin glass being more likely to crack or dent, a thicker build plate is easier to ensure a level build area. This is because at high temperatures it is more likely for thin glass or acrylic to bend or warp, leaving parts of the build plate further from the print nozzle than others. We prefer to use 1/4" thick glass instead of the 1/8" that many printers come with.

Proper Bed Clips



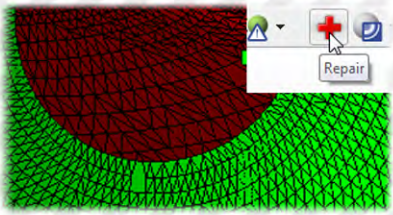
Having bed clips that are easy to manipulate is essential in having a level build plate, especially after upgrading to a thicker glass. It is best to have clips that have two access points that allow easy tightening and loosening with a strong spring. This allows you to level the bed during the initial bed leveling test and allows you to actually level during the first layer of the print.

Model Errors

One very common mistake that happens during the print may not actually be an issue with the printer or software – it may be with the model itself. Since we are often sent files that we did not design, it is mandatory that every model be checked for the ability to be printed.

Wall Thickness

One extremely common model error is having a wall too thin or non existant. A printer will only print the wall of a print if it is thicker than the nozzle diameter. And with that said, we always suggest walls be at least 3 times thicker than the diameter of the nozzle. It is best to figure out if this is going to be a problem while in your slicing program. Take the print layer by layer to make sure there are no gaps. The same is true for the thickness of the bottom and top walls.



Holes in Model

Individuals will often try to save time by converting a 2D object into a 3D field and expect this to print properly. While these 3D objects look good to the eye, the printer will not recognize the walls as anything but non-printable 2D faces. One easy way to tell if this problem exists on your print is to drag into a program like Netfabb and check for anything highlighted red that can be repaired.

There can also be actual holes in the model just as there are flat faces. Often these can be very hard to notice on a slicing program and you won't even know until the printer head gets to that section.

Angles and Limitations

As explored in Understanding Limitations, there are some things that are just not printable on an FDM 3D printer. A mostly enclosed structure that requires support may not have an exit for that support material to be removed. Extreme angles and lack of flat surface touching the build plate can cause ugly surface quality on most materials.

Nozzle Clogs

Since FDM printing involves melting polymers and feeding them through a very small hole, Nozzle clogs are one of the most common and one of the most annoying problems to occur with a 3D printer. Repair can be as simple as pushing a hard alan wrench through the barrel, some can be as time consuming as dissambling the carriage and torching the material out.

Heat Creep

As mentioned in our Cooling chapter, a barrel fan is needed for almost all FDM printers to succeed without heat creep causing a nozzle clog. While feeding material out of a hot nozzle, the heat will often creep up the filament. This causes the filament to expand inside the barrel, often creating a lump. This lump will cause a clog and will make the removal of the filament from the barrel near impossible without disassembly.

PID Tuning

PID refers to a proportional integral derivative controller that is used in 3D printing to control the temperature of the nozzle. When the PID settings are off a fan can drastically change the temperature of the hotend. The barrel fan or the active cooling fan can drop a hotend from 210°C to 160°C if the PID settings are off. This rapid cooling will make the nozzle too cold for the filament to feed. This means that the extruder gear will slowly grind into the filament until it snaps. You can update your PID settings by reflashing your firmware.

Tips to Avoid Nozzle Clogs

Keep Everything Clean - Cleaning a bed and wiping down a nozzle after and before each print can go a long way to preventing a clog.

Feed Filament Down Roughly 1/2 inch before pulling out - When filament is just sitting in a barrel, some minor heat creep can occur. While not enough to cause a clog while printing, it can be extremely difficult to remove. This can be prevented by feeding the filament past this point, and then pulling it out.

Use only one material in each printer -This can be very difficult for those with only 1 or 2 printers, but switching materials on a printer can lead to clogging. Switching from ABS to PLA may not get rid of all of the remaining ABS material, which can clog during the print.

Proper Printer Speed - Printing too fast on a material can cause grinding until the filament snaps, and printing too slow can increase the heat creep risk.

Orientation of Print

Along with errors and complicated models, the orientation of a print can drastically change the time to print, material used, and surface quality. It is recommended to print on a flat surface and make sure to not have any extreme overhangs

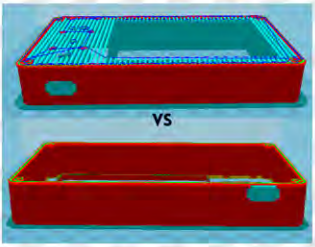
Flat Bottom

It is extremely beneficial to have the surface that is touching your buildplate be flat. A flat surface allows for strong adhesion to the build plate, easier support removal, and better surface quality.

When support is needed on an area that is very close to the build plate, it can be very dificult to remove.

Support Material

The amount of support material needed can drastically change depending on the orientation of the print. Support material essentially works as scaffolding for any angle more extreme than 45° from the bed. It is normally best to print in the orientation that requires the least amount of support material in order to save time and money. The print quality itself is also much better when oriented in the fashion with the least amount of overhangs.



Overhangs

As mentioned above, it is best to keep extreme angles oriented in a way that will need the least amount of support material. This should be taken into account whenever designing a part for 3D printing. There are numerous 3D designs that look great on the computer but print very rough due to extreme angles and overhangs.

Post-processing

Post-processing refers to working on the print after it has already been printed. We do not mean support and brim removal, but rather acetone vapor finishing, combining large prints, and even painting.

Smoothing the Surface Quality

There are a couple of ways to improve the surface quality of a print after printed. Since **ABS** is soluable in acetone, if the part is made of **ABS**, or some form of **ABS** combination, you will be able to give it an acetone vapor bath. This will allow the part to become more water tight and give it a shine that is similar to injection molded pieces.

PLA can be smoothed with a brush on gel such as **XTC**. This is a bit more time consuming and less effective than an acetone vapor bath.

Pre Post

To properly apply an acetone vapor bath, it is easiest to have a slow cooker and a vacuum. You can do an overnight acetone vapor bath at room temperature, but results have been much better in a slow cooker. It can take a little while to hone in the amount of time required, but when done correctly, parts should only take around **2** minutes to give a good acetone vapor finish. Once that is complete it is best to vacuum purge the parts to remove any excess internal acetone and improve strength.

Combining Prints

One annoying thing about printing can be the size limitations of the build area. The largest standard **FDM** printers for sale are right around **1** foot^3. That doesn't mean you can't split larger prints up and combine them after printing. **ABS** can be welded with **ABS** cement and **PLA** can be combined with strong super glue. Cracks can then be filled with a **3D** printing pen to increase strength. Once strong, the seams can be smoothed out with bondo or spackle.

Once the entire part is strong and the cracks are smoothed out, most materials work great with spray paint.

QUALITY OPTIONS

Quality refers to the surface quality of the print and not necissarily the strength. The best way to increase the quality is to decrease the layer heights and use a small nozzle, but there are a couple of other settings when slicing.

Layer Heights

Since FDM printers work layer by layer, the smaller these layers are, the more detailed the print and smaller the tolerences. Most standard printers can comfortably print 200 micron layer heights, others can get down to about 50. The smaller the layers, the longer the print is. A 100 micron layer height print takes just about twice as long as one with 200 microns.

Layer heights are also limited by the nozzle size. A printer with a .4mm nozzle cannot print 400 micron layer heights. A good ratio is to not have layer heights above 3/4 the diameter of the nozzle.

Nozzle Size

As mentioned above, the nozzle size can effect the quality of the print as well. The smaller the nozzle, the more detailed a print can be in the XY direction. If there is a wall on the print that is only .5mm thick, it will be impossible to print on a .6mm nozzle, while a .25mm nozzle will have two complete shells. If there are ever any holes on your print or text is not coming out gret, it may be worth while to check if a smaller nozzle will get the job done.

Settings

There are settings within your slicer that will also help with the quality of the print. PLA and other materials should have an active cooling fan for the surface to be smooth. Lowering the infill overlay percentage will prevent veiny surfaces. It is also good to run the printer slow with a slow acceleration speed, especially on the outer shell. But remember as mentioned in the Nozzle Clogs chapter, slowing the extruder down could require some temperature changes in order to prevent a clog.

And as mentioned in the previous chapter, post-processing is also another great way to improve the quality of your print.

RETRACTION

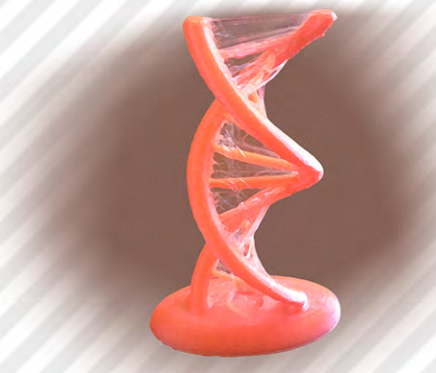
The extruder retracts filament in order to prevent excess ooze when moving between sections of the print. Issues can arise on certain filaments since this can increase grinding. When retraction is turned off, prints can get a very hairy look and will leave cobweb like strands connecting sections

Settings

The settings involved with retraction are the speed at which the material travels, minimum distanced traveled before retracting, minimum amount of material extruded before retracting and Z hop when retracting. Z Hop refers to the amount the extruder raises when retracting. This is set to a minimal amount so that small prints are not knocked off.

Each material will require different tweaks to these settings. Materials that are more prone to grinding and snapping will require slower retraction with more caution applied in each of the other areas.

Problems



Too little retraction leaves hairy, cob web looking prints with a lot of cleanup and an ugly print. This can also lead to very thin walls if the printer does not extrude enough after retraction. Finally, an extruder may ooze too much when having too little retraction.

Too much retraction can cause the surface quality of a print to decrease because the extruder may over compensate when starting the print again, leaving a small buldge where it returns. The printer will then have difficulty moving over that section on the next layer. Too much retraction can also lead to grinded filament, as mentioned above.

Following printing guides for specific filaments, such as those we provide on our website, along with trial and error with your specific printer extruder setup, will help you to get these settings right.

SUPPORT

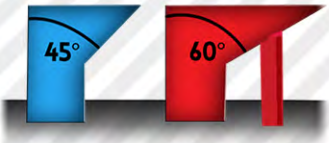
Support refers to the scaffolding material that breaks off of a print required for extreme angles. Since prints build layer by layer, the filament needs something to be laid upon without the material drooping. Each material requires it's own support settings and have their own angles that can be printed without support.

Settings

As mentioned in the Orientation of Print section, support material is needed for extreme overhangs. The angle at which the printer will generate support material is chosen within the slicing program. The amount of material (fill) generated in the support can also be chosen. When using dissolvable support and dual extruders ,you are able to take the angle down to 0° and fill up to 100%, causing the printer to create a solid block of support material. The higher the percentage and lower the angle, the longer the print and more material that is required. This is why it is beneficial to design parts that require the least amount of support.

The distance from the print in the XY and Z directions refer to the amount of space between the print and the support material. The closer the material, the cleaner the underside of the overhang, but the more difficult the support is to remove.

When printing with a single extruder these settings will have to be tweaked depending on the material used. PLA can handle angles as high as 55° or 60° and only requires a 15% support fill, printing at a distance of .8mm in the XY directions and .16mm in the Z direction. ABS, on the otherhand, requires a higher support fill of 23% and is required at 45° and above. The distance from the print should be .7mm in the xy direction and .13mm in the z. These settings will be tweaked depending on your printer and extruder. Some materials just cannot print support and have a sucessful clean print, making the design of the model that much more important.



Removal

One of the most annoying parts about 3D printing is support removal. With settings drastically closer and in a higher support fill than those described above, you will often need to resort to sanding machines and time consuming labor. Removal should be fairly easy on PLA and ABS with a set of pliers, but even with the best setings, the print will be cleaner without the markings of removable parent support material.

TEMPERATURE

Getting the temperature right on your build plate and hotend is crucial for any print to be successful. Each material requires its own settings for both. The devices used for temperature reading are called thermistors and thermocouples.

Settings

The settings for both the extruder and the hotend change for each material. **PLA** is printed at **180°-220°** celcius, and while they do not require a heated bed, it is extremely recommended to heat yours to around **60° C** (its glass transition temperature) to prevent warping and prints being knocked off. **ABS**, on the other hand, has to be printed from **230°-240°C** and require a heated build plate of around **110C**. Large **ABS** prints also require ambient temperature be around **55°C** to not warp or delaminate.

Unique materials have unique settings. Some nylons and flexible materials require a coat of a **PVA**/water solution be laid to a **45°C** bed for proper bed adhesion. Trial and error, as well as reasearch, are required for a successful print in each material.

Thermistors

Thermistors are the most commonly used temperature reading device in **3D** printing and come stock with almost every hotend. Thermocouples are able to be more acurate at higher temperatures, but require specialty electrionics that not many boards accept.

Thermistors are very thin and brittle objects. This is why it is key to be precise and careful with the assembly of the thermistor to the hotend. Make sure to have the cords tied tight without any chances of being tugged on by any obstruction mid print. As always, be sure that no wires or parts of the thermistor are exposed. Some thermistor-hotend setups have the head of the thermistor smashed into the hotend via a screw and washer. Being a mechanical and frail part, this will eventually give you troubles and require maintenance. You will see a "Min Temp" error for the extruder when a thermistor is not reading. We prefer to use third party thermistors on our **E3D** hotends that screw into the hotend and come with strong insulation. These have a much longer life, require much less labor to install, and allow you to remove the hotend without tweaking your wire configuration or removing the entire extruder.

UNDERSTANDING LIMITATIONS

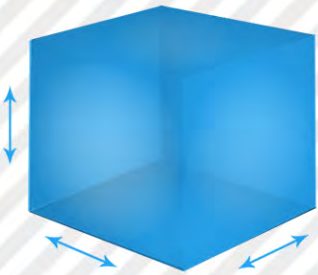
3D printing can often be presented as a plug and play solution to create limitless prototypes and models. Along with model errors and extreme overhangs ,there are still other limitations that should be understood. While the technology is revolutionary, there are distinct limitations in the market, some of which may always exist.

Speed

Each printer and extruder set up can each handle different accelerations and top speeds. If a printer does not have a large enough build area to move, the carriage may never reach top speeds due to acceleration and deceleration times.
The speed at which a printer can extrude material is not only limited by the printer, but by the material itself. Certain materials require a cooling time before laying the next layer. This just cannot be sped up without changing the properties of the material used.

Build Area

One distinct limtation on any 3D print is the build area. As of now, most FDM printers you can buy are limited to under 1 cubic foot. This means that large prints are required to be assembled and welded post print, or they are just not printable. At this time any current FDM printer that can build larger than this will require a large space and a high pricetag.



Nozzle Diameter

The diameter of the nozzle is directly correlated to the tolerance in the XY direction. The smaller the nozzle, the higher the quality and the lower layer heights that are possible That being said, small nozzles are far more likely to clog or experience issues. .6mm and .8mm are two of the easiest nozzles to work with, have stick to the bed, and complete a fast successful print; but anything with extremely thin walls or precise detail in the XY direction will be tough to complete. Remember that every wall must be at least the thickness of the diameter on the nozzle in order to even be recognized by the printer.

VENTILATION

While there has not been extensive studies done as of yet, it is intuitive that the gas and particles released by the heating of plastics for 3D printing cannot be healthy. This is why it is best to have proper ventillation. You do not want to be printing overnight, every night, with the machine on your nightstand.

DATA

At SD3D we have been working closely with a university in order to get real data on the health effects of 3D printing. We happen to print as much as possible in an enclosed environment with filters on the outside of the exaust fans, which drastically reduces any harm that may come. While it will take more time to get final data, it is clear that standing over a print of Carbon Fiber ABS while it extrudes is not beneficial to your health.

A paper published in the journal *Atmospheric Environment* by the Illinois Institute of Technology measured the amount of harmful ultrafine particles (UFP) that desktop printers emit into the air. Materials brought back a UFP of 20-200 billion particles per minute. The study says these particulates will stay in your lungs, but the longterm effects are still not perfectly understod. Other than this study, most data is taken from studies not directly related to FDM printing.

Enclosed Printer

We originally wanted to enclose our printer to keep ambient air hot to prevent warping and delamination. This transformed into a device that moderates the temperature and can blow air out anywhere that is hotter than the average. This air is then blown out of the specific location of the printer through a small carbon filter. This not only helps to produce a clean print, but can reduce any amount of particulates that exist in the air during the print.

Since there are extensive patents that currently exist for enclosed printing devices, we had to get creative to make sure we are not breaking any intellectual property laws. A patent such as this is not only preventing 3rd parties from expanding the industry, but can also cause health problems by limiting the amount of proper precautions a user can take.

WARPING

When a print's corners and sides curve upwards, it is refered to as warping. It is one of the most common problems when printing in materials with a high shrinkage rate. As polymers cool, they shrink, and if the ambient temperature is not consistant, the print will shrink unevenly. Even when parts are stuck to the bed entirely, there will often be delamination of layers as a result of warping.

Causes

As mentioned above, warping is generally caused by uneven cooling of the print. The larger the part and the higher the infill, the higher the chance that the print will cool unevenly and either delaminate or warp, since the center will be hotter than the surrounding areas. The mere fact that the build plate is heated to a temperature hoter than the ambient air will always make warping a possiblity, even in enclosed environments. This is why it is imporant to know your surroundings - if printing in very cold environments, warping will be a much higher probability.

The material being used is also a huge factor in the amount of warping that can occur. **PLA** has a very small shrinkage rate, so long as the temperature is not extremely cold around the area and a heated build plate is used, the print should not warp. **ABS** actually has a very high shrinkage rate, and high glass transition temperature, which both mean **ABS** is very suseptable to warping.

Some specialty filaments are even more likely to warp, making it very crucial to understand the material being used.

Prevention Tips

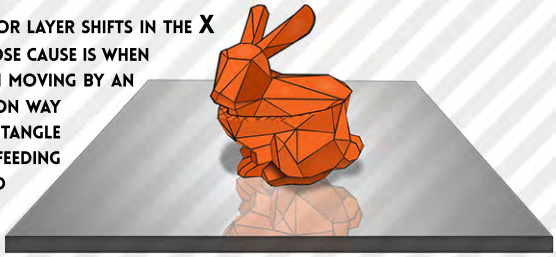
Warping may not be entirely identified until hours after the print has started. Unlike an unlevel bed, this may take your printer offline for half of a day and waste a lot of material.

Taking tips from the Bed Adhesion section will go a long way. If you do not have a enclosure for your printer, you will need to understand your alternatives to high probability warping materials. There are materials with similar strength properties as **ABS** such as **PETG** and **PET+**, that do not have such a high liklihood of warping. These can often be easier to print, though they have a lower glass transition temperatures and different elongation properties. Preparing your build plate and planning ahead with the right materials can go a long way in saving hours of wasted printer time and frustration.

XY AXIS SHIFTS

XY axis shifts, more commonly referred to as layer shifts, are when an extruder or bed shifts during a print and continues in a different area of the build plate. Parts may look perfectly fine other than everything being shifted on one particular layer.

Causes



There are quite a few causes for layer shifts in the X or Y axis. One easy to diagnose cause is when the extruder is prevented from moving by an obstruction. The most common way this occurs is when there is a tangle in the filament or it is not feeding correctly. We have experienced brand new filament spools that are tangled straight from the factory, which cause a layer shift hours into a print. It is important to store your filament properly and keep the spool tight with either tape or putting the end through a hole on the side to prevent this.

Other problems can arise from unlubricated rods and stepper motor malfunctions. If the bed or carriage has a stepper motor that is stiff or is has a pullie that is harnesed loosely, skipped steps will occur. Skipped steps in the motor wil almost always cause a layer shift. When a print has multiple layer shifts but continues the entire print, it is likely not due to a filament tangle, but rather stepper malfunction or roughness on the rods.

Prevention

As mentioned in the Jogging and Homing section, lubricating the rods is extremely helpful in preventing layer shifts. The likely best prevention method that can be implimented is having proper storage of your filament. Once you are done using a spool you should always keep everything as tight as possible and make sure it cannot unravel or become tangled.

As mentioned in the Filament Options section, 1.75mm filament is more flexible while on the spool, making feeding much easier. While we prefer 3.00mm filament for it's lower tolerances in diameter, we have experienced layer shifts when a spool is near the end and it just doesn't feed properly.

Since stepper motors are mechanical parts, they will eventually wear down. Periodically check your belts to make sure they are tight and that the stepper pullies are moving smoothly

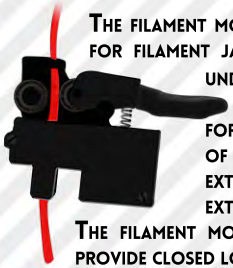
YEARLY ADVANCEMENTS

One amazing fact about 3D printing, as with all technologies, is there are giant advancements nearly every year. We at SD3D are doing our best to speed up these advancements and have some technological innovations that are going to change the entire market.

Dual Geared Extruder

This extruder is a drop-in replacement for any existing extruder assembly mounted to the standardized Greg Wade's adapter plate. Most importantly, the dual gear configuration provides twice the surface area distribution leading to a drastic reduction in filament stripping and material slippage. The integrated encoder on the extruder drive, along with the filament monitor, provides closed loop feedback for the extrusion axis throughout the print. This also allows printer manufacturers to configure their builds with slave assist drives on the extrusion axis.

Filament Monitor



The filament monitor provides a solution for filament jams, material shearing and under/over extrusion. Precision optical diameter monitoring is provided with enhanced signal filtering onboard. This allows for continuous filament diameter measurements with a resolution of 100 microns. These measurements are used to automate extrusion flow rate changes throughout the print to provide extremely tight print tolerances and enhanced surface quality.

The filament monitor also senses the physical position of the filament to provide closed loop motion control for the extrusion axis when used with our encoded stepper motors. This monitor also provides feedback for automated inventory control systems, including our RFID material tracking module.

Auto Ejection

SD3D's heated build platform is modular and can be configured to most standardized configurations. It includes a proprietary gradient free heated build plate which also contributes to lower surface level warping forces. Combining the precision bed leveling and advanced HBP stack make it possible to non-destructively auto-eject printed parts reliably. Printers can now be configured to run continuously without needing to be monitored or serviced by technicians between prints.

Z - HEIGHT

The Z-height is exactly what it sounds like, the motion of the extruder or build plate in the Z direction. Getting the Z-height correct on a print can actually be more time consuming than thought and will almost always cause more hiccups than issues in the XY axis.

Fine Tuning

Just as with leveling the build plate, the Z-height needs to be correct in order for a print to start, not warp, not get clogged, and to overall finish properly.

When using a printer that does not have precise auto bed leveling and a way to automatically find the proper Z-height, which most do not, you will need to tweak the distance between the extruder and the Z-endstop. This can be done by homing your device and then tweaking with the screw that can adjust the distance between the extruder and the endstop.

Build Plate Expansion

When materials are heated they will expand from their normal size. This can be seen in both heat creep in the Barrell Cooling section as well as the effects of shrinking when cooling in the Warping section. The same is true for the glass and build plate you are printing on.

When heating a bed to 110° celsius you are are bound to make the glass expand a bit. This expansion will cause the build plate to be slightly closer to the nozzle than when cold. This is why it is best to determine the Z-height while the bed is heated to the temperature you expect to print at. Once this expansion is recognized you can add G-Code to your slicer profile for that particular material with a Z-offset. This offset will cause the extruder to rise, or bed to lower, based on the amount of expansion.

Quality and Nozzle Size

As mentioned in the Quality Options section, when we say quality we mean the height of each layer as well as the nozzle size. When printing in fine quality of around 100 microns and with a nozzle smaller than .4mm in diameter, geting the proper Z-height can become quite a headache. The smaller the nozzle and the lower the layer heights, the more precise that height has to be. Printing at 400 micron layer heights on a .6mm nozzle is far easier to hone the Z-height than fine quality prints. So not only do high quality prints require more print time, but they require more labor as well.