PARIS TEXAS PUBLIC LIBRARY

Introduction to 3D Printing

Introduction to 3D Printing

Thank you for choosing Paris Public Library for your print job! In this guide we will detail everything you need to know about the 3D printer, our guidelines for printing as well as general info about 3D printing along with tips to a successful print.

Guidelines:

1. The 3D printer is NOT to be operated without first consulting with staff.
2. We will keep in contact with you throughout the print process.
3. We DO NOT allow patrons to connect their own personal devices to the printer.
4. You are not allowed to print from the pre-programed print queue on the machine.
5. All files must be uploaded as .STL
6. We will not take the supports off of your completed print, nor will we assemble it.
7. You have 7 business days to collect your completed model.
8. All projects will be completed using PLA filament in order to maximize success rate. We will not order any filament type outside of PLA. However, you may request colors.
9. We charge $0.10/gram of what is printed
10. Only 1 object at a time can be submitted & printed

The library is unable to print the following:

1. Content or objects that are illegal or harmful to minors.
2. Content or objects that may be construed as having intent to harm.
3. Content or objects that may infringe upon the intellectual property rights of a third party.

By submitting content or objects, the patron agrees to assume all responsibility for, and shall hold Paris Public Library harmless in, all matters related to patented, trademarked, or copyrighted materials. Paris Public Library is not responsible for the functionality or quality of content produced on the 3D printer.

How Does a 3D Printer Work?

Much like traditional printers, 3D printers use a variety of technologies. The most commonly known is fused deposition modeling (FDM), also known as fused filament fabrication (FFF). In it, a filament—composed of acrylonitrile butadiene styrene (ABS), polylactic acid (PLA), or another thermoplastic—is melted and deposited through a heated extrusion nozzle in layers.

Our Current Model:

The model of 3D printer that we currently have is the Dremel DigilLab 3D45. Its specifications are as follows:

Dremel DigiLab 3D45 Specifications	
Technology	Fused Deposition Modeling (FDM)
Built-in Devices	4.5 in LCD touch screen
Connectivity Technology	Wireless
Interface	USB, LAN, Wi-Fi
Min Layer Thickness	1.27 mil
Input File Formats Supported	STL, OBJ
Flash Memory	8 GB
Build Materials Supported	Polylactide (PLA), acrylonitrile butadiene styrene (ABS), nylon
Max Build Size	10 in x 5.98 in x 6.69 in
Nozzle Diameter	0.02 in
Filament Diameter	0.1 in
Networking	Print server
Connections	1 x USB 1 x LAN-RJ-45
Power Device	Power supply-internal
Frequency Required	50/60 Hz
Software Included	Dremel DigiLab 3D Slicer, Dremel Print Cloud
Min Operating Temperature	60.8 F
Max Operating Temperature	84.2 F
Width	16 in
Depth	15.9 in
Height	20.2 in
Weight	42.77 lbs

What Software Do I Need for 3D Printing?

Nearly all 3D printers accept files in what's called STL format (named for stereolithography). These types of files can be produced by most any CAD software, from expensive commercial packages like AutoCAD to free or open-source products such as Google SketchUp and Blender. For those not inclined to make their own 3D files, 3D object databases such as MakerBot's Thingiverse offer numerous 3D object files that can be downloaded and printed out. Recommended websites to obtain models are:

Recommended Websites to Obtain Models	
Cults 3D	https://cults3d.com/en
Pinshape	https://pinshape.com/
Thingiverse	https://www.thingiverse.com/
GrabCAD	https://grabcad.com/library
3D Warehouse	https://3dwarehouse.sketchup.com/?hl=en
CG Trader	https://www.cgtrader.com/
TurboSquid	https://www.turbosquid.com/
3D Export	https://3dexport.com/
Printables	https://www.printables.com/
Yeggi	https://www.yeggi.com/
My Mini Factory	https://www.myminifactory.com/

Most 3D printers come with a software suite, either supplied on disk or available for download, that includes everything you need to get printing. The suites typically provide a program for controlling the printer and a slicer, which, in preparation for printing, formats the object file into layers based on the selected resolution and other factors. Some suites include a program to "heal" the object file by correcting problems that could interfere with smooth printing.

In our case, Dremel DigiLab 3D Slicer and Dremel Print Cloud are the recommended software/programs needed to complete your build successfully.

Dremel DigiLab 3D Slicer can be accessed by going to:

https://3pitech.com/pages/desktop-slicer-software

Dremel Print Cloud can be accessed by going to:

https://printcloud.dremel.com

Please note however that we DO NOT allow patrons to connect their own personal devices to the printer. You are also not allowed to print from the pre-programed print queue on the machine. In order to better streamline print jobs, all print jobs must be submitted through the application form listed in the Guidelines.

Types of 3D Printer Filaments

FDM (Fused deposition modeling) 3D printers make use of filaments. These materials are supplied on a 3D printer spool and are directed through a heated nozzle by means of an extruder. The extruded plastic is then used to build up a 3D printed part, layer by layer. There are different 3D printer filament types. However, 10 of the most commonly used will be described in this article including their mechanical properties, characteristics, advantages, and disadvantages.

PLA Filament

Polylactic acid (PLA) is a thermoplastic monomer made from organic sources. This is in contrast to other 3D printer filament types that are made from petroleum products. PLA is easy to print and is environmentally friendly. However, it is brittle and has poor UV resistance. Additional key characteristics are:

Warping: PLA does not warp easily during printing

Solubility: PLA is not soluble in water. But, it can be dissolved in acetone, methyl ethyl ketone, or caustic soda.

Food Safety: PLA is food safe

ABS Filament

Acrylonitrile Butadiene Styrene (ABS) is a widely used engineering plastic and 3D printing filament type. ABS exhibits excellent toughness and can withstand relatively high temperatures. Printing with ABS requires high temperatures for both the hot end and the printer bed. Heated build volumes are also required for good results. Additionally, all types of ABS tend to warp during printing, which results in poor dimensional accuracy. Additional key characteristics are:

Durability: ABS has excellent resistance to overall wear and tear. It is both tough and impact resistant.

Solubility: ABS is not soluble in water. However organic solvents like acetone, methyl ethyl ketone, and esters will dissolve ABS.

Food Safety: ABS is a food-grade plastic

Carbon Fiber Filament

3D printer filaments can be made with specific additives to either improve their mechanical properties or aesthetic appearance. Typical 3D printing filament types used include PLA, PETG, or ABS. For the purpose of comparison, ABS-filled 3D printer plastic will be used.

Carbon-fiber-filled filaments have improved mechanical properties when compared to unfilled thermoplastics. They also have good dimensional stability. Carbon fiber filaments are brittle and clog easily. Listed below are additional key characteristics:

Durability: The addition of carbon fiber improves the durability of ABS

Warping: The addition of carbon fiber reduces the amount of warping that is common with unfilled ABS

Solubility: Carbon-fiber-filled ABS is soluble in organic solvents like acetone, methyl ethyl ketone, and esters.

Nylon Filament

Nylon or polyamide is a widely used engineering thermoplastic due to its excellent wear resistance and durability. The most commonly used grade of nylon for 3D printer filaments is PA 6. Nylon is both impact and wear-resistant. However, nylon tends to absorb moisture easily. It also requires relatively high print temperatures of up to 265 °C. Below is a list of nylon's other key characteristics:

Warping: Due to the high temperatures involved, nylon tends to warp during printing. As such, a heated enclosure is recommended.

Solubility: Nylon expands when exposed to water due to its hygroscopic nature. Acetic acid and formic acid will dissolve nylon.

Food Safety: There are grades of nylon that are food safe

FLEX Filament

FLEX filament is a proprietary blend of polymers that creates a flexible copolymer 3D printing filament type. Flex is part of the TPU (thermoplastic polyurethane) family of materials. Parts printed with this material can achieve a shore A hardness of 93 A. They are also flexible and impact resistant. Parts printed using flex are hygroscopic. This means that it should be dried before use and kept dry during long prints. Other key characteristics are:

Durability: Flexible materials are by their nature durable; this is no different with FLEX filaments

Warping: No warping

HIPS Filament

High impact polystyrene (HIPS) is a thermoplastic often used for pre-production machining prototypes. However, it is also one of two 3D printing filament types used as a soluble support material, alongside ABS. HIPS has similar properties to ABS, making it an ideal second extruder material. Despite being soluble, HIPS produces harmful fumes during printing. As such, it is recommended to print in a well-ventilated area or to direct fumes outside with a duct. Other key characteristics are:

Durability: HIPS has excellent durability due to its unique mix of flexibility and strength

Warping: HIPS can suffer from excessive warping if temperatures are not carefully controlled. Heated enclosures are recommended.

Solubility: HIPS is soluble in D-limonene

Food Safety: HIPS is a food-safe material

PVA Filament

Polyvinyl alcohol is a biodegradable 3D printer plastic that dissolves easily in water. It also has printing properties close to that of PLA. This makes PVA one of the more ideal 3D printing filament types for PLA support material. Although PVA is easy to use, it can be expensive due to it being used as a sacrificial support material. Some key characteristics of PVA are listed below:

Durability: Due to its water solubility, PVA is not useful in most applications, as moisture will degrade the plastic

Warping: PVA can warp to some degree

Food Safety: PVA will dissolve in the presence of water; as such it is not recommended for use with food

PETG Filament

Polyethylene terephthalate glycol-modified (PETG) is a modified variant of PET. The addition of glycol lowers the melting temperature sufficiently for PETG to be more user-friendly. Aside from being easy to print, PETG is also UV-resistant. Its key disadvantages are its poor adhesion and its tendency to create strings when the printhead crosses empty space between features. Other key characteristics of PETG are:

Durability: PETG has excellent mechanical properties, while also being resistant to a wide range of chemicals and high temperatures

Warping: PETG is not particularly prone to warping

Solubility: PETG is soluble in toluene and methyl ethyl ketone (MEK)

Food Safety: PET is food safe and by extension so is PET

TPE Filament

Thermoplastic elastomers are flexible materials that can be melt-processed in most types of 3D printers. There are many 3D printing filament types of TPE, and it is easy to confuse TPE with TPU. TPU is generally on the harder shore A range whereas TPE is softer. The properties and characteristics described in this section are based on the FilaFlex TPE filament. FilaFlex has high elasticity and good bond-ability. It is expensive, however. Some key characteristics of TPE are:

Durability: TPE has good abrasion resistance and excellent flexibility

Warping: TPE does not warp

Food Safety: TPE is not food safe

PC Filament

Polycarbonate (PC) is an advanced engineering thermoplastic with excellent mechanical properties and is the strongest 3D printer filament. It has high strength and a glass transition temperature of 150 °C, making it ideal for high-temperature applications. However, PC needs to be printed at very high temperatures of up to 310 °C. It is very hygroscopic and will readily absorb moisture. This moisture can then cause defects in the printed part. Other key characteristics of PC are:

Durability: PC is one of the most durable 3D printing filament types

Warping: PC is very prone to warping

Solubility: PC can be dissolved in tetrachloromethane, pyridine, and chloroform

Food Safety: PC can be used for food containers

How to Choose the Best Type of Filaments?

Selecting the best 3D printer filament depends on the application. If a quick prototype is required then PLA will suffice. However, if more strength is required ABS might be a better choice. It is also ideal to choose a material that doesn't warp readily and does not absorb moisture. Eliminating these two common problem areas will make the print easier. For more information, see our guide on printing in 3D.

Which Filament Produces the Smoothest Prints?

PETG is one of the 3D printing filament types that produces very smooth prints provided the printer is properly calibrated. However, printing in ABS and then smoothing with acetone can also create very smooth parts.

What Is the Strongest 3D Printer Filament?

Polycarbonate is the strongest 3D printer filament, provided it is printed correctly.

What Is the Best Filament To use?

For general purpose use, PETG is an excellent option as it is cheap, easy to print, and has good mechanical properties.

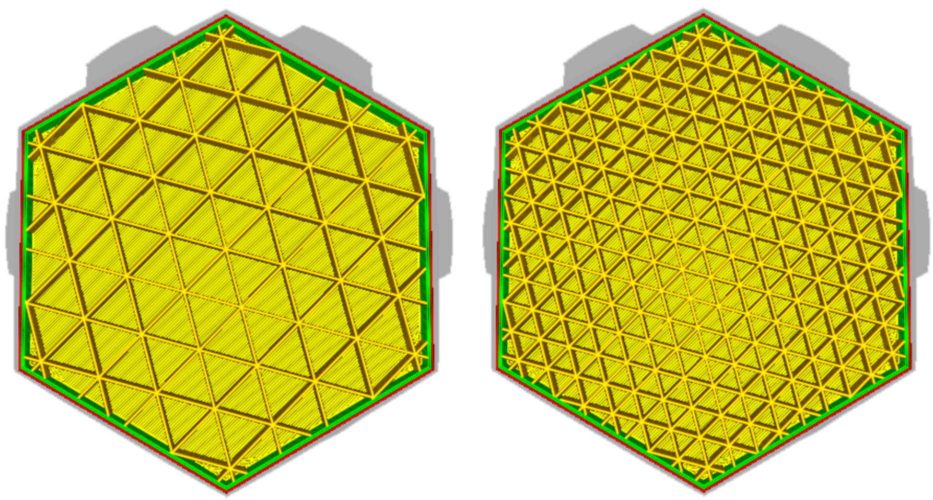
What Is the Best Filament for Beginners?

People have different opinions on what filament works best but it really all depends on what you are trying to print. In our case, we only supply PLA filament as it is beginner friendly and more likely to produce a successful build with our printer.

Infill Settings

Infill density

The infill density defines the amount of plastic used on the inside of the print. A higher infill density means that there is more plastic on the inside of your print, leading to a stronger object. An infill density around 20% is used for models with a visual purpose, higher densities can be used for end-use parts.



The model on the right has a higher infill density than the model on the left

Infill line distance

Instead of setting the infill density as a percentage, it’s also possible to set the line distance. This determines the distance between each infill line, which has the same effect as changing the infill density.

Infill line directions

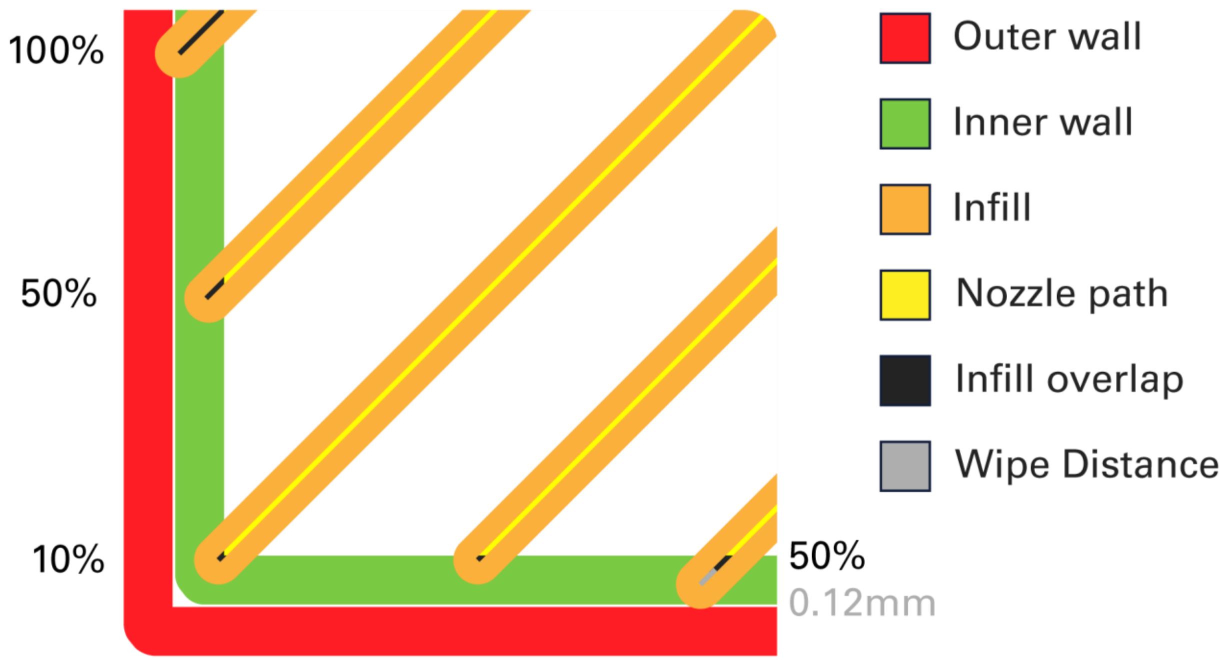
The infill lines usually print at a 45° angle. At this angle, both the X- and Y-motor work together to obtain maximum acceleration and jerk on the layer without losing quality. If the lines need to be printed in a different direction, you can set it here at 0° for vertical and 90° for horizontal. For example: [0,90] results in a horizontal-vertical top/bottom pattern.

Infill XY offset

Infill patterns are centered for each model loaded. To move the pattern to the left, right, top, or bottom a X or Y offset can be used. A positive value moves it UP and RIGHT, while a negative value moves it DOWN or LEFT. This does not work for the concentric infill types.

Infill overlap percentage

With this setting you can control the amount of overlap between the infill and walls. It can be set as a percentage or a true value. A higher value usually results in better bonding between the infill and walls. However, it might also reduce the visual quality of the print, as a value that is too high could lead to over extrusion. The default value in Ultimaker Cura will in be sufficient in most cases.



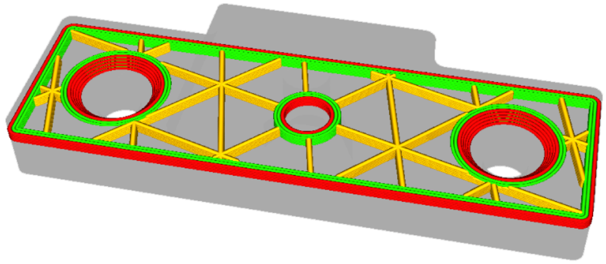
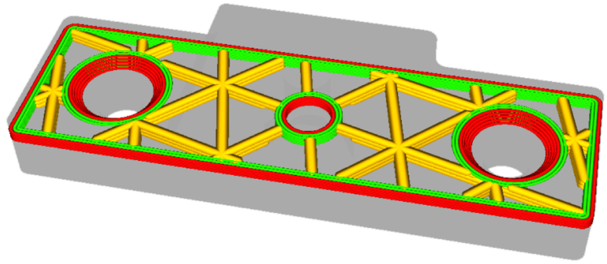
A visualization of the infill overlap and wipe distance

Skin overlap (percentage)

The skin overlap works in the same way as the infill overlap, which is described in detail above. It can be set as a percentage or a true value. The skin overlap influences all top and bottom layers in a print.

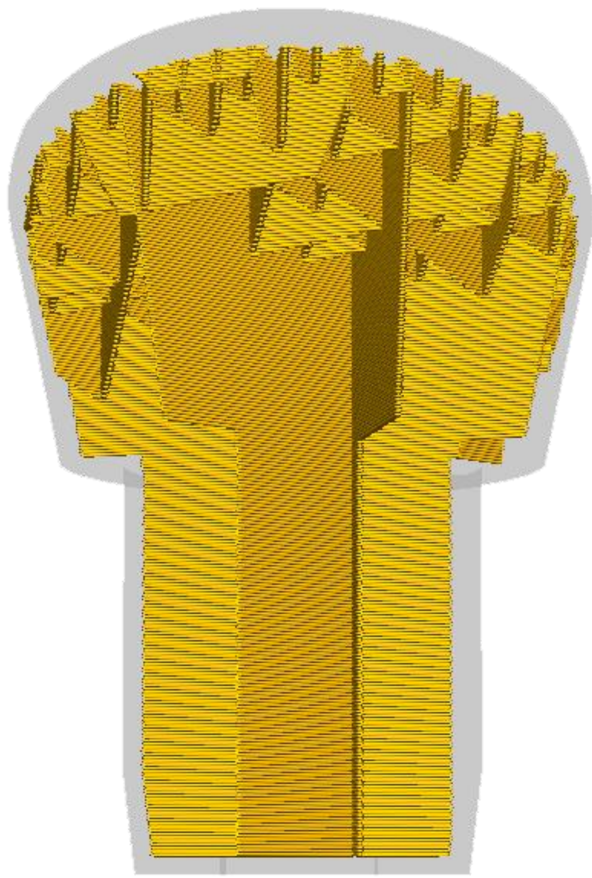
Infill layer thickness

Since the layer height of the infill is not important for visual quality, you can use thicker layers on the infill to reduce the print time. When adjusting this setting, always make sure that it is a multiple of the layer height, otherwise Dremel will round it up to a multiple of the layer height. This means that you can, for example, print with an infill thickness of 0.2 mm while the layer height is 0.1 mm. The printer will first print the walls for two layers, and then it will print one thicker infill layer.

Gradual infill steps

Gradual infill reduces the amount of infill used by decreasing the infill percentage in the lower layers. Every gradual infill step divides the infill percentage by a factor two. The result is a dense infill near the top layers, which is essential, and a reduced print time.

Example: Gradual infill steps = 2 and infill = 20% --> Infill = 20% for the top 5mm, infill = 10% for the rest of the print

Gradual infill step height

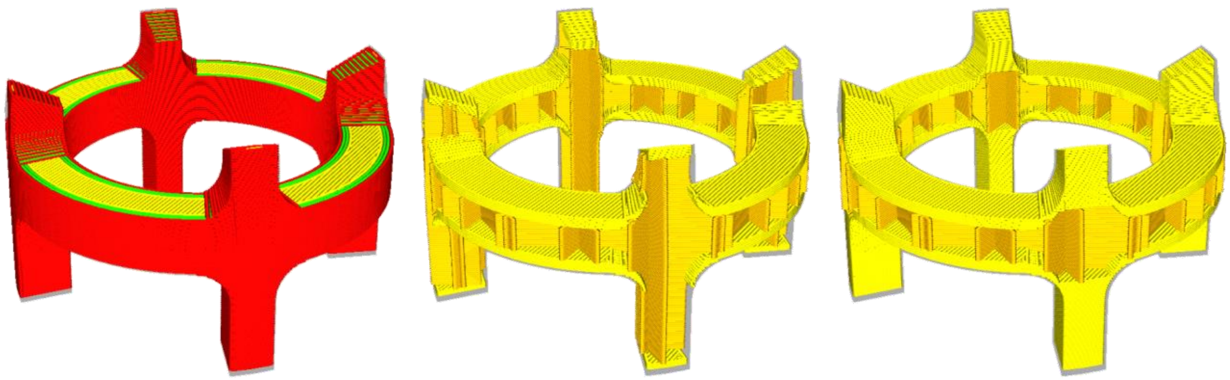
Gradual infill step height is the height at which the infill should be reduced, as calculated from the top layers. This way the top layers can easily be closed, without the use of extra infill in the print.

Infill before walls

With this setting enabled, infill will be printed before the walls. This results in better overhangs because the walls will stick to the already printed infill. Printing in this order can also have a disadvantage. If the infill is printed before the walls, there is a chance that the infill will be visible through the walls, resulting in a rougher surface finish.

Minimum infill area

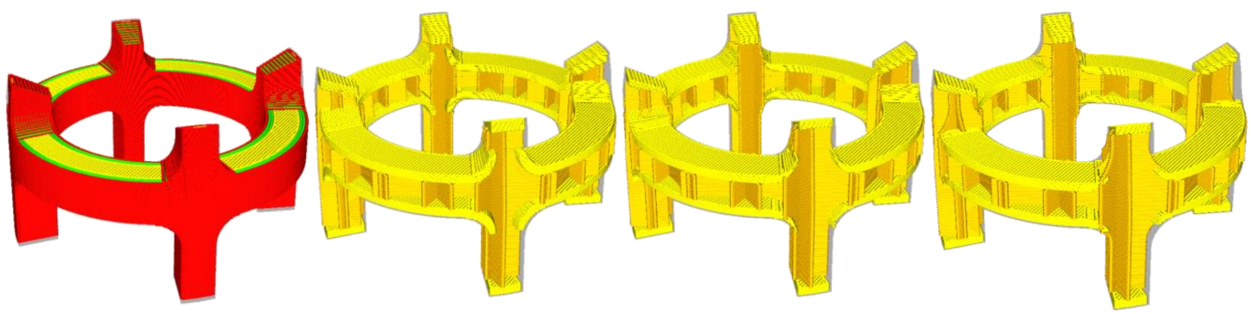
This setting allows small areas on a single layer to be printed with skin instead of infill. Take a flat roof with a chimney, for example, the chimney is thin and fragile and can be printed solidly with skin.



The minimum infill area strengthens the legs of this coupler by filling it completely with skin

Skin removal and skin expansion

The skin (top and bottom layers) in a model is printed to reach the minimum thickness set in the print profile. However, some models need a stronger or lighter internal geometry. These settings expand or remove the skin horizontally, where infill would normally be printed. Expanding this slightly allows protruding model elements to have better adhesion to the rest of the model, making it stronger or lighter. Flat surfaces with protruding elements in the Z-direction will have a stronger base, making it sturdier.



From left to right; A preview of the model, skin expansion of 0.8mm, no skin expansion or removal, skin removal of 0.8mm

Skin removal width

This is the width of the skin to be removed when applying skin removal. It can be applied individually to the top and bottom layers.

Skin expand distance

This refers to the distance by which the skin will be expanded. A bigger value results in longer, but sturdier prints. A lower value will only marginally increase the strength.

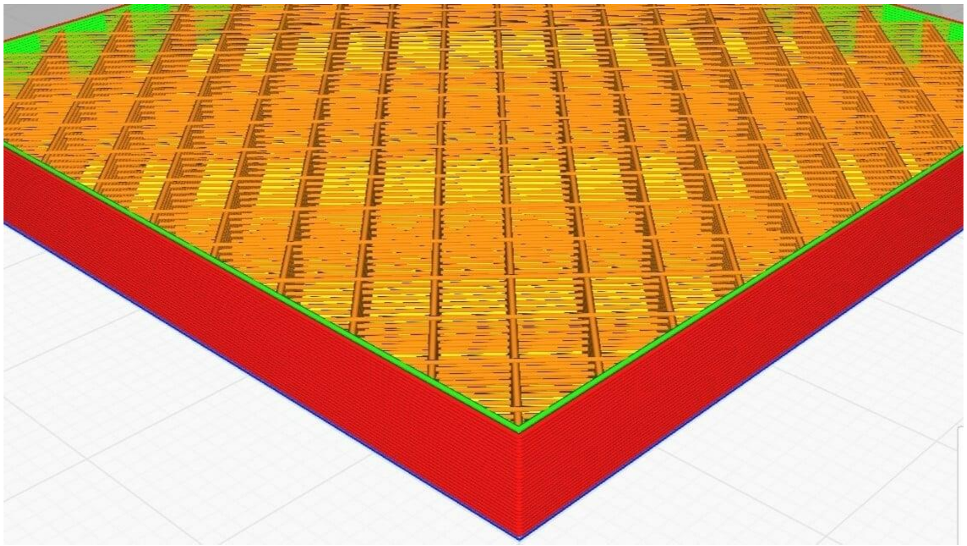
Maximum skin angle for expansion

Since skin is present throughout the model, it is unnecessary to expand all of the areas. Rather, only the areas below the angle specified here will be expanded. In that way, flat surfaces with protruding elements will be strengthened, without affecting the rest of the model. For this, 0° is horizontal (and does not expand anything), while 90° is vertical, and expands everything.

Minimum skin width for expansion

This parameter will prevent small skin (top and bottom) areas from expanding. It is specifically used when targeting only the strength of large, flat surfaces of the model.

Perimeters



Perimeters trace the outline of a part, creating a strong and accurate exterior. The infill is printed inside of these perimeters to make up the remainder of the layer. The infill typically uses a fast back-and-forth pattern to allow for quick printing speeds. It's important that the two sections merge together to form a solid bond.

The number of perimeters can affect the wall thickness. For example, a higher number indicates a thicker wall, which requires more material and takes longer to print.

Here are some tips for using perimeters:

Decorative: Use 1 or 2 walls and 0-20% infill

General use: Use 2 or 3 walls and 30-60% infill

Utility items: Use 3 walls and 40-80% infill

Get a smooth finish: Try printing the perimeters first to get the smoothest finish on the outside

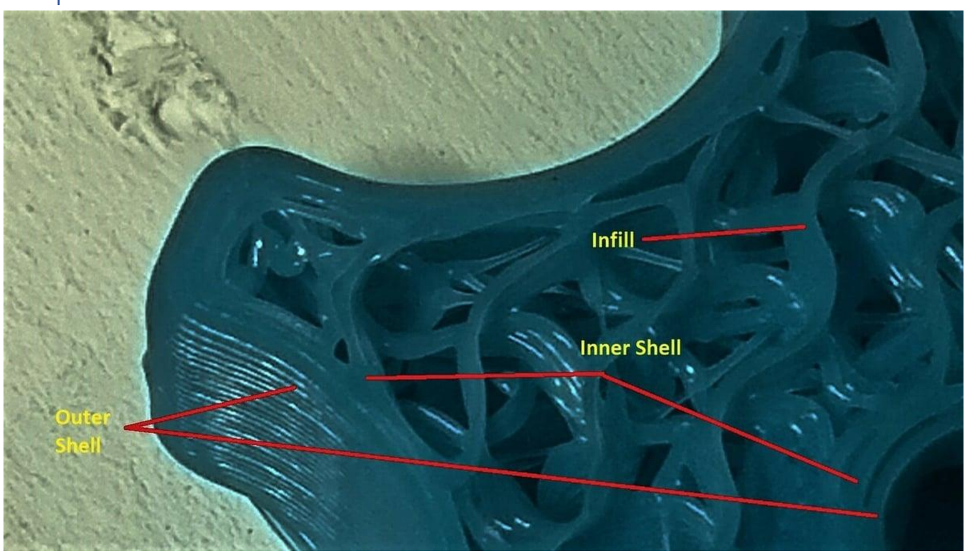
What you may not realize is that the shell, the bit you see, has a greater effect on part strength than you may have anticipated. While it’s true that infill plays a role in part strength, this also depends on the infill orientation and pattern, and it may provide little or no benefit in some directions. So, if you’re looking for a sturdier print, cranking up the infill to 100% might just be a massive waste of filament if you’re not taking a close look at other settings.

When we refer to the shell as “the bit you see”, there are technically a few layers to this (literally). A 3D print isn’t only built in layers in the Z-axis, but in all three axes. The shell is made of lines of filament usually laid down from inner to outer lines. The very outermost of those lines is the one you see (or that you sand smooth). The ones behind, known as inner shells, are backup and give a stronger structure to the outermost one.

However, it’s not just about the shell as a standalone aspect of the print. You see, to give a better structure to the print, the infill needs to be strongly connected to the shells. That bond is achieved by letting the infill lines slightly protrude into the inner shells, creating a good bond between the lines. This is part of the reason you can sometimes see the infill pattern on the outside of the print, particularly if there are very few shells or if you’re using a transparent filament.

3D PRINTING SHELLS: THE BASICS

How Shells Impact Your Print



A close-up showing inner and outer walls as well as infill (Source: Che Simons via All3DP)

The parts coming off your FDM 3D printer are anisotropic, meaning that they don’t provide the same strength in all directions (if they were, they’d be isotropic). This affected by a variety of things, as mentioned before. For now, we’re just going to look at the shells, or perimeters, and how these affect the prints.

Opting for more shells will lead to a part that’ll be stronger, heavier, and will take more time to print, and fewer shells will mean the opposite. There are a few other considerations, though.

For example, if you’re going to print a large, flat model with particularly big flat walls in the Z-axis, you’ll most likely want several shells. As the part cools, the plastic will contract a little, which can cause your walls to warp (sometimes badly). Sometimes infill can compensate for this, but generally, at least two or three lines will be of greater help.

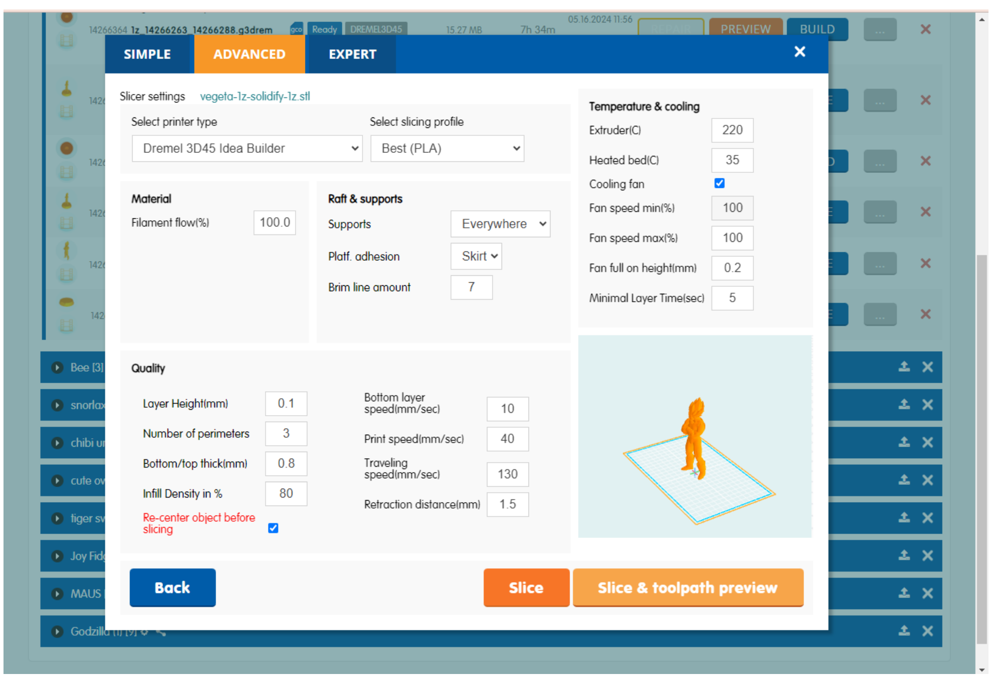
That said, if you print a large object with many walls, not only will your print take a long time, but you'll also risk bed adhesion issues. In a nutshell, thicker walls generate more force as they contract while cooling, which can cause a significant amount of uplift at the ends of the wall. Luckily, there are several ways to combat this problem.

As we've mentioned, more shells will use more material and add weight to your object, and they will also take longer to print, which might increase the chance of print failure. The choice is a trade-off between ease of printing, part weight, and strength. The intended use case also needs to be considered, but a good rule of thumb is two to four walls with a 0.4-mm nozzle or one or two walls with a 0.6- or 0.8-mm nozzle. Keep in mind that two walls at 0.6 mm are equivalent to three walls at 0.4 mm.

You may have to reprint to tweak the strength of the part. Just be sure to dispose of your failed prints responsibly or recycle them!

3D PRINTING SHELLS: THE BASICS

Setting Shells



What is the difference between shells, perimeters, walls, and top and bottom layers? The terms are often interchangeable in the slicing software we use. Cura, PrusaSlicer, and SuperSlicer, and Dremel DigiLab 3D Slicer all have slightly different ways of accessing the same settings, and then you also have proprietary software like FlashPrint or IdeaMaker, as well as paid slicers like Simplify3D. Be sure to take the time to learn your specific tools if you want to see good results.

In a nutshell, “shell” means everything that’s not infill (although some slicers refer to shells and perimeters synonymously). Perimeters are, strictly speaking, the number of lines used in printing each wall. Wall thickness is the number of perimeters multiplied by the line width, so two perimeters at 0.4 mm would be a 0.8 mm wall thickness.

We should also mention that the top and bottom layers form part of the overall shell. However, they’re often referred to and controlled separately.

In Cura, for example, they can be found under “Top/Bottom > Top Thickness > Top Layers” and “Top/Bottom > Bottom Thickness > Bottom Layers”. These are the number of layers that form your top surfaces and also the number of layers directly on the build plate (or supported by support material). Decreasing your bottom layers can sometimes improve print bed adhesion by reducing contraction pressure across the bottom of the print. This is a bit hit or miss and reduces the finished part’s strength though, so we recommend perfecting your first layer instead.

Final Considerations

1. You can’t increase the number of shells in vase mode (or “Spiralize Outer Contour”), which is where the walls are printed as a single line spiraling up from the bottom to the top of the print. This means that vase mode always has just one perimeter.

2. If your model has a cavity in the middle, or a thin wall, setting more perimeters, shells, or a higher wall thickness won’t fill this in. The slicer will simply draw the number of walls that will fit within the model at the designated line thickness (with some adjustment by Arachne as discussed above).

3. Different filaments have varied properties depending on the specific material, but they’re usually quite strong. You might be surprised about how much force a single wall can withstand. Do some experiments on your machine and processes to save plastic in the future!

4. If you’re going to sand your print, we recommend that you use at least two – preferably four – walls. While you should be able to stop at less than one layer of removed material, the complex geometry of most prints often makes this difficult.

Tips to Improve Print Quality

Since we are printing for you, you don't necessarily have to worry about some of these tips but in the future you may want a 3D printer of your own or you are curious as to what we do to maintain the machine so we can keep it at optimal efficiency.

1. Level the bed & set the nozzle height
2. Keep an optimum nozzle temperature
3. Use different building plates to create different effects
4. Don't over tighten the belts
5. Replace your nozzle with a fresh one

How Do I Take the Supports Off?

Most 3D print jobs will require supports for a successful print. Some 3D print models do not require supports because the model is designed in a way so that it may maintain its stability on its own. You can technically print without the supports, but this is not recommended as it can greatly diminish the quality of your print.

Since we will not take the supports off for you, you will have to do this on your own. Here are some recommended tools you will need:

1. Precision Cutting Pliers
2. Electric Polishing Machine
3. Different types of sanding heads
4. Nail File
5. Wire Cutters
6. Removal Tool Scrapper
7. Craft Knife
8. Craft blade
9. Tweezers
10. Long nose pliers
11. Hand Drill
12. Drill Bits
13. Carving tool knife
14. Sanding Sticks
15. Screwdriver
16. Art knife
17. Prying Tool
18. Ruler

This set on Amazon is highly recommended:

https://www.amazon.com/Printing-Cleaning-Polishing-Including-Multi-Purpose/dp/B087Z1136V?th=1

References

3D printing shells: All you need to know. All3DP. (2023, June 17). https://all3dp.com/2/3d-printing-shells-all-you-need-to-know/

Ghisays, J. (2023, August 25). *3D printing tips to improve print quality*. www.allprintheads.com. https://www.allprintheads.com/blogs/news/3d-printing-tips-to-improve-print-quality

Infill Settings. Support community. (2022a, November 14). https://support.ultimaker.com/s/article/1667411002588