MAGIGOO MANUAL

How to get the most out of the MAGIGOO® Adhesive range

A comprehensive user manual describing the main challenges in adhesion faced in the FDM 3D printing process as well as their solutions. This manual also contains useful tips on the 3D printing process and printing materials - v1.2

01/2021

Magigoo Manual

- Don't Panic -

Table of Contents

1. What is Magigoo?

We wouldn't blame anyone for thinking that Magigoo is a magical formula that solves all first layer adhesion mishaps. This is in fact only partly true, since as we all know, unfortunately Magic isn't real. Magigoo however is real and is also a bespoke solution scientifically formulated for solving first layer adhesion issues plaguing various FDM plastics.

The Magigoo adhesive range is not just about providing the best first layer adhesion for your printed parts. Each adhesive is developed in-house by our R&D team. Here we evaluate the chemistry of the formulation but also develop it in tandem with real word 3D printing tests in order to assess adhesive performance and usability. This is done whilst keeping the use of nasty chemicals to a minimum.

How it works

Magigoo adhesives are easy to use and only require that you apply them by spreading a thin coat on your build-plate, printing and then easily removing the part. As an interface layer, Magigoo will increase the adhesion of the printed first layer to your build-plate substrate.

Thermoplastic materials usually have a very different chemical make-up to the build-plate material. As a result, there is a big difference in surface energy often leading to poor adhesion especially for engineering thermoplastics.

Getting your 3D printed part to adhere well to the build-plate is only the first part of the job. Apart from bridging the gap in adhesion between the thermoplastic and the build-plate, the interface layer provides an additional feature. In order to be able to use the part you must also be able to remove it from the build-plate. This is actually the 'Magic' part of our adhesives as it lets the user to very easily remove 3D printed parts from the build-plate. The part removal process will depend on the type of adhesive, the printing surface and the printing material being used.

When using most Magigoo adhesives, after printing you can just remove the part as the build-plate cools down!

2. Product Range

Magigoo Original

Magigoo is an easy-to-use 3D printing adhesive designed to address 'the first layer not sticking problem'. A strong adhesion between the 3D print and the heated build plate is ensured when applying Magigoo. However, the actual magic starts when printing is ready: just wait for the plate to cool down and then remove your prints with just a finger flick! Magigoo original works with most common 3D printing materials including PLA, ABS, PET-G and HIPS. Regardless of you being a hobbyist or professional – it is extremely rewarding to have a 30 hour print complete without any warping and having it release easily once cooled.

Magigoo PRO PC

Magigoo Pro PC is specifically designed for 3D printing in polycarbonate plastics in FDM printers. It offers a very strong adhesion platform for polycarbonate based filaments. Such materials include pure Polycarbonate, filled Polycarbonates and PC alloys such as PC-ABS. Just like the original Magigoo adhesive, Magigoo PC is easy to use and releases on cooling.

Magigoo PRO PA

This is Magigoo PA (Nylon), the adhesive specifically designed for 3D printing in polyamide (nylon) plastics. It offers a strong adhesion platform for polyamide based filaments including PA-6, PA-6,66, PA12 and the various filled counterparts. Magigoo PA also shares the magical release on cooling properties of other Magigoo adhesive products.

Magigoo PRO PP

Magigoo PP is a first layer adhesive specifically designed for 3D printing in polypropylene plastics. It offers a strong adhesion platform for polypropylene based filaments and other polyolefin based materials such as DOW OBC. As all other Magigoo adhesives, Magigoo PP is a water-based adhesive intended to be as safe to use as possible. Magigoo PP offers a strong and convenient adhesion surface for PP materials without requiring the need to apply tapes or other surfaces to your 3D printer platform.

Magigoo PRO Flex

This is Magigoo Pro Flex, the adhesive specifically designed for 3D printing flexible plastics. It offers a strong adhesion platform for thermoplastic elastomer copolymer based filaments and other flexible filaments whilst ensuring easy removal. Magigoo Flex is primarily intended for use with TPEs such as DSM Arnitel ID 2054, which are often a challenge to stick to most 3D printing surfaces. However, Magigoo PRO flex also ensures easy removal of parts printed in flexible materials using water to help separate the parts from the build-plate.

Magigoo PRO HT

High temperature filaments prove to be challenging materials for FDM printing due to their higher tendency to warp. These materials are generally printed at elevated temperatures owing to their high temperature properties. Magigoo Pro HT is a specialized solution for FDM printing of PEEK and PEKK filaments which was developed to work at elevated printing temperatures. Magigoo PRO HT is tested for use at build-plate temperatures up to 150 °C.

Magigoo PRO PPGF

This is Magigoo PPGF, the adhesive specifically designed for 3D printing in glass filled polypropylene plastics. It offers a strong adhesion platform for glass filled polypropylene based filaments such as Owen's Corning GF-30PP and BASF Ultrafuse GF-PP. Magigoo PRO PPGF is Magigoo PP's stronger cousin, offering a stronger adhesion platform for the stiffer reinforced polypropylene blends. Magigoo PPGF requires that the bed is heated up for safe and easy part removal.

Magigoo PRO Metal

Magigoo Pro Metal is designed as a professional first layer adhesive with BASF Ultrafuse® 316L. Magigoo Pro Metal is non-hazardous, non-flammable and also approved for use in post processing stages of BASF Ultrafuse® 316L. Due to the delicate nature of the green part, care must be taken when removing parts printed in BASF Ultrafuse® 316L to avoid part deformation. Magigoo Metal provides sure and reliable adhesion during printing and enable the easy removal of the printed part through submersion in water, thus avoiding potential damage to the green part.

Magigoo Large Format (120 mL)

This extra size version with a bigger applicator area makes applying on larger print surfaces a breeze. The Magigoo 120 mL bottle was designed for use with FDM printers having a larger surface area on the build-plate. The extra capacity of the larger bottle also means that your Magigoo will last you for endless spools of filament!

Pro Kit

The Magigoo PRO kit contains the core PRO adhesive range for the most commonly printed engineering material types. These Magigoo Original, Magigoo PC, Magigoo PA, Magigoo PP and the kit also contains a cleaning cloth. The kit is designed for professional users printing with a variety of engineering materials.

Magigoo Coater

What is the coater?

The Magigoo Coater is designed and built for use in FDM/FFF 3D printing to cover the build plate of a 3D printer with a thin layer of Magigoo and Magigoo Pro bed adhesives, to ensure reliable 3D printing, reduce warping and provide easy release. The Magigoo Coater is a new Magigoo application product that is designed for build plates with larger build areas as well as printing farms operating with multiple printers. Its main component is a pump spray attached to an array of nozzles that evenly distribute a set amount of Magigoo onto the build plate. This pump spray is attached to a 250 ml Magigoo Bottle.

The Magigoo Coater allows for a precise coating of Magigoo on a large build plate, giving an optimal adhesion with an equally desirable release. With an increase of 3D printing farms, the Magigoo Coater would be beneficial since it is able to cover multiple build plates in a short time. It offers a degree of repeatability that would be useful in 3D printing farms. Another benefit of the Magigoo Coater is the ability to apply Magigoo to large build plate, in a matter of seconds. Unlike DIY adhesives and tools such as brushes or rollers, as such tools which might interfere with the precision and surface finish of the adhesive layer, the Magigoo coater is specifically designed for this purpose to provide an easy reliable and repeatable application experience.

Before using Magigoo Adhesives, read through the adhesive safety data sheet for safe use. Check that the Coater valves and the applicator are clean and dry. If necessary, wipe the applicator and valves with a damp cloth and/or wipe dry prior to using the Coater. If the valves are still clogged, clean the manifold channels.

Silicone Coater versus Thin film Applicator

There are two versions of the Magigoo Coater, one with a static metal rod as the spreader and one with a silicone rubber roller as the spreader. The thin film applicator is currently discontinued.

The main difference is that the metal rod is static with respect to the Coater - there is no motion involved. It does not rotate, but rather it slides on the build plate and carries the adhesive with it. This rod is specifically designed to provide a thin even layer of Magigoo adhesive of a precise thickness. Using the metal rod, one needs to press down onto the build plate firmly. This is done to ensure that there is contact all across the rod thus allowing for a finer, more precise application.

The Silicone roller on the other hand rolls and rotates on its axis. This allows the adhesive to rotate around the roller and spread across the build plate in an easier and more fluid application. Furthermore when using the rubber roller, there is less need to press down onto the build plate. It is simply like rolling a paint roller on a surface. The use of the roller also makes it easier to spread around the build-plate adhesive especially on larger surfaces. One additional advantage of the rubber roller is that it can provide more even coverage on surfaces which are not completely flat.

Other than the mentioned differences, the method of application is pretty much the same and a guide can be seen in this [First use video](). Similarly, the [Cleaning video]() gives you a guide on how to clean and maintain the rubber roller.

While the Silicone roller might be more familiar and intuitive to use, the application it creates is not as precise and thin as the one created by the metal rod. Either way, the Magigoo Coater is a useful product to have if you plan on applying Magigoo to build plates of large areas, or simply to a lot of build plates!

Adhesive Compatibility

The Magigoo Coater should be used only with adhesives produced by Thought3D under the brand Magigoo or other Thought3D approved adhesives. The Magigoo Coater has been designed to work with Magigoo and Magigoo Pro 3D printing build plate adhesives.

Surface	Original	PC	PA	PP
Regular Glass	✓	✓	✓	✓
Borosilicate Glass	✓	✓	✓	✓
PEI	✓	✓	✓	✓

Tips and Tricks

- When applying using the Silicone Coater, alternate between along the build plate and across it a couple of times, to ensure a better spread of the adhesive.
- If the applied coating of Magigoo adhesive is inconsistent, check if the valves need to be cleaned or replaced.

  N.B: When using the Silicone Coater ONLY, it should still function well with 1 or 2 clogged valves. On the other hand when using the Thin Film applicator one needs to make sure that all valves are working correctly.
- Make sure the bed is completely clear of any debris or dust before applying, as this may tamper with the layer of Magigoo Adhesive.
- It is important to clean the valves and the applicator after each use of the Coater. The manifold should be cleaned when the Coater is going to be stored away for a long duration or when the adhesive type is going to be changed.
  - Avoid drying the applicator with a paper towel, as this will most likely result in small paper fibres getting stuck to the applicator.
  - It is best to use the provided Magigoo microfiber cloth to wipe down the applicator after it is used.
- A couple of Coater Stand designs can be found over on https://magigoo.com/downloads/ under the 'Magigoo Coater Base' dropdown.

Magigoo is an Adhesive range!

Each Magigoo adhesive works with a specific family of thermoplastics, since each family is different in terms of chemical make-up, physical properties, and also thermal properties. As a result apart from showing different adhesive requirements in terms of surface energy, different materials also impose different thermomechanical requirements from the adhesive.

From our tests, a specifically formulated adhesive for your 3D printing material, not only gives optimum adhesive performance but also the best 3D printing experience in terms of ease of use. In fact, we believe that there is no such thing as a one-size fits all solution when it comes to first layer adhesion due to the different properties of thermoplastics.

For this reason, we always recommend using the appropriate Magigoo adhesive for your 3D printing materials. Firstly, adhesive performance will be affected if the wrong adhesive type is used. Furthermore, even though in some cases a material can show good adhesion with several Magigoo adhesives, there might still be issues with the part removal process. For example, ABS can probably be printed without warping on most Magigoo adhesive formulations. However, when using formulations other than Magigoo Original for ABS it can increase the chances of glass chipping or the part getting overly-stuck to the build-plate.

3. How to Use:

Application:

All Magigoo Products

- **Shake** the bottle.
- **Apply** to the desired area by pressing the nib on the bed (activating the inner valve) and spreading as needed.
- **Print** according to the filament/printer manufacturers recommendations

Part Removal:

Magigoo Original, Magigoo PRO PC, Magigoo PRO PA, Magigoo PRO PP and Magigoo PRO HT

The part should self-release or become easier to remove on cooling, in most cases the part can be directly lifted off from the plate. If the part cannot be easily removed by hand, a part removal tool can be used to gently lift the part off the build-plate. When using a brim, it is advisable to first peel off the brim around the part and then attempt to remove the print.

Magigoo PRO PPGF

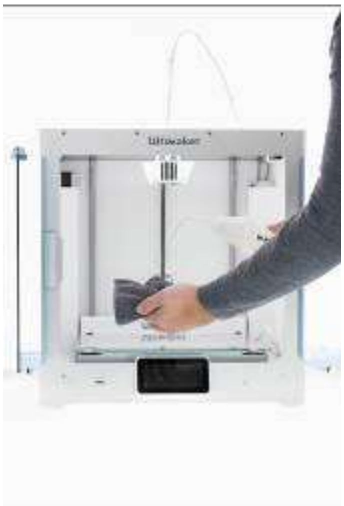
Heat the build-plate up to above 80 °C and gently pry off the part using a part removal tool. When using a brim it is advisable to first peel off the brim around the part and then attempt to remove the print.

Magigoo PRO Flex and Magigoo PRO Metal

Submerse the whole build plate in water or soak around the part with water. After short period of time – generally 30 minutes to 2 hours depending on the size of the part, the print should come off on its own or be easy to remove using a gentle twist or a part removal tool.

Cleaning

Magigoo Original, Magigoo PRO PC, Magigoo PRO PA, Magigoo PRO Flex, Magigoo PRO Metal and Magigoo PRO HT



Use a damp cloth to wipe off all the goo residues until a clean surface is visible. It's as easy as that! To avoid cross contamination between adhesives we recommend wiping the glass build-plate once more after rinsing the cleaning cloth with water. Only use water to clean off Magigoo residues, other solvents will not work as well as water or at all!

Magigoo PRO PP and Magigoo PRO PPGF

Magigoo PRO PP benefits from a few minutes soak before the adhesive layer is wiped off. It is recommended to place a damp cloth over the Magigoo PP/PPGF application after which the adhesive layer will turn white and can be then easily wiped off with a cloth.

Build-plate material compatibility

The adhesives work on heated build-plates, with glass surfaces giving the best results. However, Magigoo also works very well with a variety of build-plate surfaces. The build-plate material will have a direct effect on first layer adhesion. The adhesives are generally designed and optimized primarily on glass build-plate surfaces. This includes a variety of glass types such as normal glass, borosilicate glass and ceramic glass.

Other common surfaces such as PEI, Buildtak (PC), and Kapton tape show similar levels of adhesion to glass build-plates when Magigoo adhesives are used. This is true for the vast majority of 3D printing materials. Extensive material testing has however highlighted that a few engineering grade 3D printing materials might show a slight drop in adhesion when printed on a surface other than glass, with Magigoo adhesives. Nonetheless, the use of our adhesives will still improve the adhesion of the print to that particular surface when compared to printing on the same surface without any adhesive solution.

The use of Magigoo can also extend the life of you build-plate surface. The adhesive in fact acts as an intermediate layer between the build-plate material and the print. In some cases, such as when printing flexible materials, Magigoo Flex can act as a release agent making part removal much easier!

Magigoo is Safe and Easy to use

As we've already mentioned, when formulating our adhesives, we also ensure that our adhesives are as easy to use as possible and with the least amounts of hazards to the user. In fact, all Magigoo adhesives are water-based and solvent-free. Having an adhesive which is water based has several advantages including non-flammability, and the fact that the adhesive residues components are water soluble.

The dab type applicator bottle also ensures that Magigoo is easy to apply. No additional components such as brushes are required to spread the adhesive. Furthermore, when using the nib type applicator one can be more specific with his adhesive application. This means that the user can only apply adhesive to the area being printed on. Furthermore, unlike spray type application systems, this application method prevents overspray from landing on your 3D printer components or work area.

4. Optimising settings for improved build-plate adhesion in FDM printing

Introduction

Build-plate adhesion is one of the many challenges faced during FDM printing of thermoplastic materials, be it in a domestic, professional, or industrial environment. Some types of FDM materials tend to be more challenging to print with, especially in the case of engineering materials and high temperature materials. Insufficient build-plate adhesion will inevitably lead to warping or premature print failure, which in both cases results in the printed parts being rendered inadequate for use in their intended application. Insufficient build-plate adhesion thus needs to be effectively controlled in order to save time and printing material.

The various factors which cause a print to warp will be explored. These include the adhesion of the printed material to the build surface, the thermal differential during printing, the material type and the printing settings, amongst others. The various parameters which could be tweaked in order to mitigate the warp will be discussed with a focus on the build-plate temperature. In this part, a method on how to determine the optimum build plate temperature is presented. Further refinements and tips on how to optimise first layer adhesion and reduce warping will also be discussed. Finally, one can find a section about the use of Magigoo® products with different FDM materials.

Figure 1: **For domestic and professional applications, reliable build-plate adhesion is essential for reliable and repeatable results**

What causes warping?

The FDM printing process requires that a polymer is molten and extruded onto a build-plate or a previous layer of extruded material, layer by layer. Each layer will thus be cooling at different rates leading to a temperature differential when the object is being printed. This manufacturing method will thus result in a part which is cooling non-uniformly, leading to several issues including warping and print-failure due to insufficient adhesion.

Warping is when the print starts to lift up from the corners and deforming in a lateral direction (Figure 2), in extreme cases warping will cause the print to completely detach from the print bed. Nonetheless, even in mild cases it can be detrimental due to loss of dimensional accuracy which can lead to the part being unusable depending on the application. The severity of warp will depend on a number of factors with some materials being more prone to warp than others. For a successful print this detrimental effect needs to be avoided as much as possible.

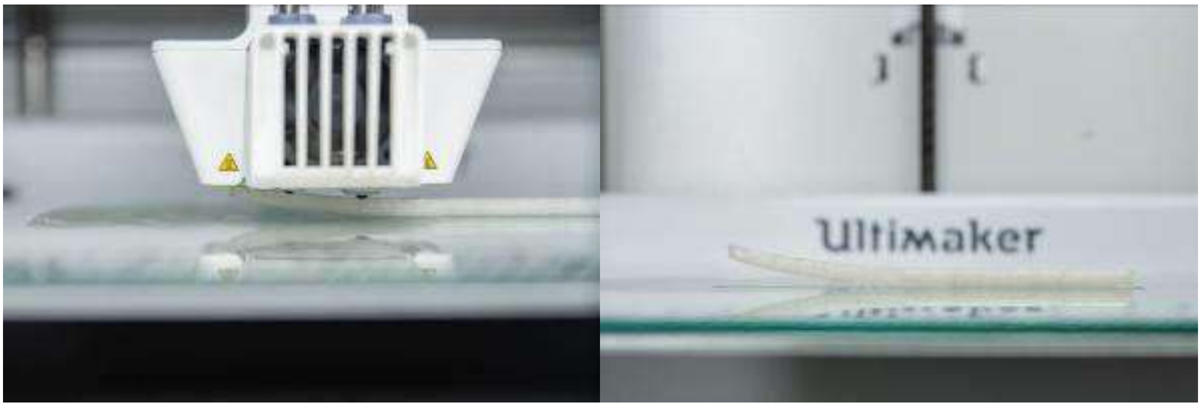


Figure 2: Nylon warping due to insufficient bed adhesion, one side is printed on regular glue stick, the other on Magigoo PA adhesive

The cause of warping can be attributed to the differential thermal contraction of each successive printed layer:

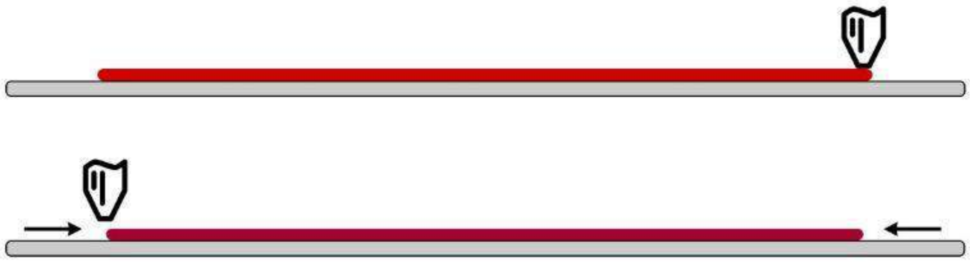


Figure 3: First layer

1. When the first layer is extruded onto the build-plate, **it starts immediately cooling down** to the build plate temperature, this will lead to the **first layer to contract slightly** (Figure 3).

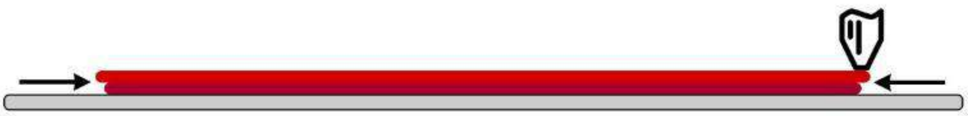


Figure 4: Second layer

2. The second layer will be deposited on the already contracted first layer while also cooling down, thus contracting on top of the first layer. Since the bottom layer is already slightly contracted when the upper layer is deposited, **the upper layer will cause the layer below it to compress** (Figure 4).

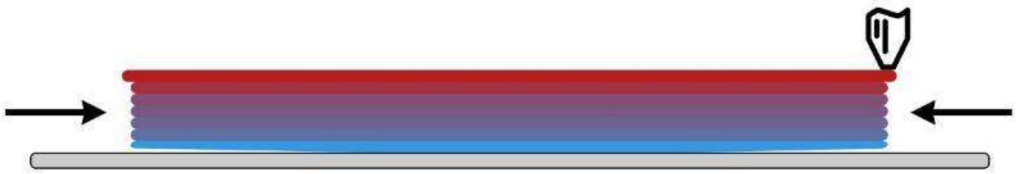


Figure 5: Print warp due to thermal gradient

3. This process will keep on repeating itself as new layers are added causing more **lateral compression of the lower layers**. This results in an **overall shear force between the printed layers** which we can call **warping stress**. If the **warping stress** is larger than the stiffness of

the part and the bed adhesion, the bottom of the print will inevitably start pulling away from the build plate. (Figure 5).

The extent of warping depends on several factors including the material properties and the printing conditions, which are not independent of each other. One of the most important material properties governing the amount of warp in a print is the thermal contraction of a material. Thermal contraction is when a material shrinks on cooling. This is a result of an inherent property of most materials which expand on heating and shrink on cooling. The thermal contraction of a material can be described by the Coefficient of Thermal Expansion - CTE. The CTE describes the tendency of a material to change its shape, area and volume as the temperature changes. A material with a high numerical value for linear CTE exhibits large changes in length as a response to temperature change. As a result materials which have a high CTE are more prone to warping than materials which do not exhibit large changes in dimensions at the thermal changes present during FDM printing.

Additionally, thermoplastic materials, especially engineering grade materials tend to show varying degrees of crystallinity, these materials are often termed semi-crystalline. In this case, the change in the crystallinity of the material during solidification and cooling can also influence the contraction of a material. Crystallisation can lead to potentially higher shrinkage rates since crystalline structures tend to be more tightly packed (Figure 6).

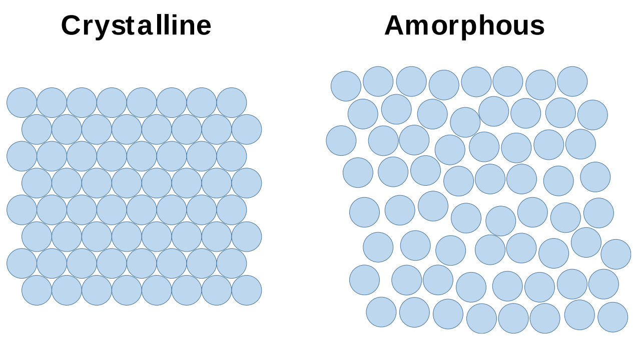


Figure 6: A visual representation of the building block arrangement in crystalline and amorphous solids

As a general rule the slower the cooling rate of a material, the larger the tendency for the material to become crystalline. The degree of crystallisation of a printed material is specific to each individual material and also depends on several factors including: ambient temperature, part cooling rate, additives, layer height and print speed, and merits a discussion of its own. At this point it is sufficient to assume that crystalline materials such as PP, some nylons and PEEK tend to warp more than amorphous plastics.

Controlling the warp

The solution to prevent warp is to ensure that the adhesion between the first layer of the printed object and the build plate is stronger than the thermally induces stresses on the first layer by the rest of the print. The warp of a print can be mitigated by controlling build-plate adhesion and the thermal gradient. The first layer adhesion depends on several factors, these include:

- Build-plate material
- Adhesive used
- Type of material being printed
- Build-plate temperature
- First layer nozzle temperature
- First layer print speed
- First layer flow
- Environmental/chamber temperature

The first layer adhesion is generally stronger when the bed temperature, nozzle temperature and first layer flow are high and the first layer print speed is low, however **these settings are highly dependent on the printer, material, and environment combination.**

A good control of the thermal gradient during FDM printing can also help in reducing warp by reducing thermal stresses. This is generally, but not necessarily achieved by keeping the internal build temperature slightly (10 ° - 20°C) lower than the glass transition point of the material being printed. The glass transition temperature is the temperature, or rather a temperature range above which a thermoplastic material starts acting like a rubber, whilst below it the material is in a hard 'glassy' state. In other words, below the glass temperature the material is hard and strong while above this temperature the material is softer and less stiff. This means that close to the glass temperatures the thermoplastic exhibits lower thermal stresses since it is softer. For a large portion of FDM filament materials, the temperature inside a printer without an actively heated and enclosed chamber is not enough to be close to the glass point of the material. Nonetheless keeping the build temperature constant is key to prevent printing issues and it is always advisable to prevent any drafts and sudden changes in temperature when printing.

A number of materials such as Nylon and ABS can be easily printed in enclosed printers with heated beds. The heated bed would be sufficient to keep the internal temperature of the printer high enough to mitigate warp if a strong enough build-plate adhesive is used. On the other hand, high temperature materials such as PC, PEEK, PEKK, Ultem and PPSU will probably require a heated chamber in order to reduce the thermally induced stresses during printing which often leads to warping and other printing defects. Other factors which may affect thermal stresses include, layer height, print speed, shell thickness and infill percentage, with higher values generally leading to a higher tendency to warp.

Looking at this information above one might think that just increasing the build plate temperature as much as possible will solve all the problems of warping, unfortunately this is not the case with most FDM materials. Increasing the build-plate temperature too much can cause three major issues:

1. Firstly, overheating can cause a loss in print quality since the printed polymer on the build-plate becomes too soft, this will usually cause curling at sharp edges and leads to deformation of the part (Figure 7).

Figure 7: Benchy print using PLA, images on the left shows a benchy printed with the heated bed set to 60 °C, while the images on the right show a benchy printed at 80 °C. On closer inspection it can be noted that the benchy on the right warped and also shows artefacts due to the layers curling up from excessive heat.

2. If the build-plate temperature is too high, the mass of material closest to the build-plate can become soft enough to allow newly deposited layers on top of the print to deform the base of the print. On the other hand, if the base of the print is stiff enough it would resist the stress generated from the uppermost layers of the print and thus minimising further warp (Figure 8). A practical example is when comparing the behaviour of rigid printing materials to that of flexible materials. Rigid materials such as ABS or PC, usually do not continue to warp, or warp significantly less as the print reaches a certain height. On the other hand, flexible materials such as PP or nylon will keep on warping significantly as the print progresses and reaches its full height, if the build plate adhesion is not strong enough.

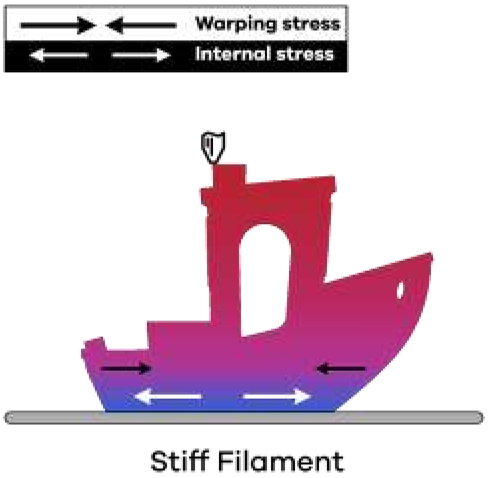


Figure 8: Printing with a stiff material versus a flexible material. The stiff material will be able to resist the warping stress, since the cooler material at the bottom of the print has enough strength (internal stress) to resist further warping. A flexible or soft material will not be strong enough to resist the warping stresses induced by the shrinkage of the uppermost layers of the print.

3. Another factor is related to the nature of the adhesive. While at higher temperatures the adhesion between the plastic and the adhesive is usually greater, the actual strength of the adhesive layer will start decreasing. Most adhesives, and even build-plate surfaces are made of thermoplastic polymers which soften as the temperature is increased. As a result, there exists a range of temperatures for each material in which the warp of the part being printed is at a minimum. In this range there will be a compromise in which the first layer adhesion is maximised, the adhesive layer strength is also maximised, and the thermal and warping stresses are minimised. For some materials this optimum range might be wide but for most challenging materials the optimum printing temperature range can be as small as 5-10 °C.

For these reasons determining the best printing temperature in your 3D printing system is important for best performance with the adhesive layer.

How to determine the optimal build plate temperature

This process of finding the optimum build-plate temperature is relatively simple but often overlooked. One needs to print a test print specifically designed to assess the warping behaviour at different temperatures. This print is preferably small in order to minimise use of filament and also to save time. However, it also needs to be a good indicator of adhesive performance. A print with a wide base and a low z-height is not an ideal indicator of adhesive performance, on the contrary prints with narrow bases such as wedges (Figure 9) and prints with sharp corners are usually more prone to warp and more ideal for assessment of adhesive performance. Such prints can be easily found online on sites such as *www.thingiverse.com*. (Figure 10, Figure 11)

Figure 9: Wedge print, an ideal geometry for testing the first layer adhesion of your build surface, the narrow base limits the contact area with the build plate. This shape is one of the geometries used in the Magigoo testing labs to assess the adhesive performance with different material and adhesive combinations.

Figure 10: The Warpinator 5000 by Maker's Muse, this is a similar concept to the wedge print but slightly more extreme.

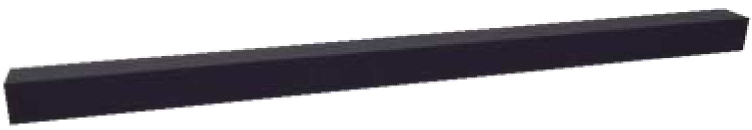


Figure 11: A simple warp test by Hagster, downloaded from Thingiverse

Tools and Materials:

Apart from a well-functioning and calibrated 3D printer and a 3D model for testing, one would also need some way to measure the amount of warp on a print. This can be done by visual comparison but can also be more accurately done using a feeler or slip gauge in order to determine the amount of warp (Figure 12).

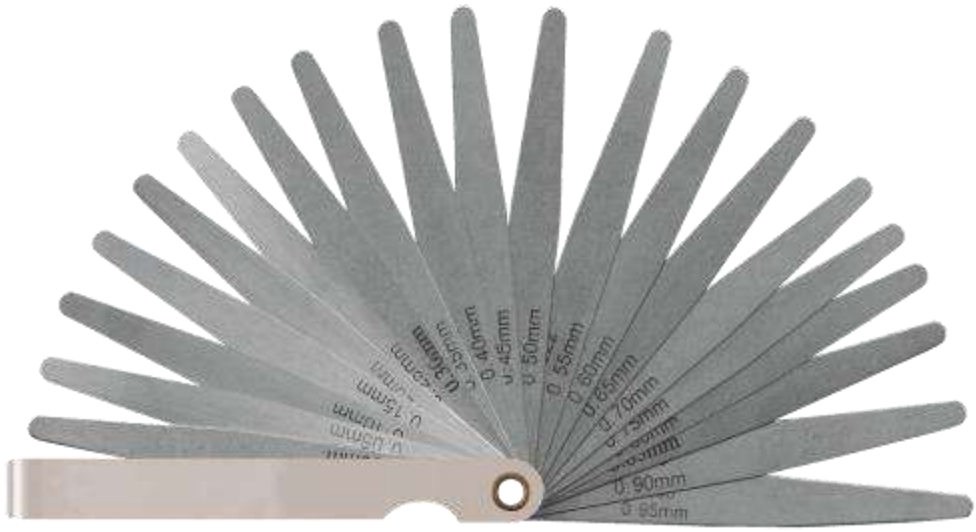


Figure 12: Feeler gauge, ideal for measuring small amounts of warp accurately

Method:

1. One can start the test from a recommended build plate temperature setting, either from the filament manufacturer or from other reliable sources or using Table 1 below as a rough guide for the optimum printing temperature for each material.

Table 1: General build plate temperature range for ideal adhesion with different FDM thermoplastic materials

Material	Build-plate temperature	Chamber Recommendation
PLA	50-70 °C	Open
PET-G	60-90 °C	Open
PET	60-90 °C	Enclosed
Carbon filled PET	60-90 °C	Enclosed
CPE	60-90 °C	Enclosed
ABS	80-110 °C	Enclosed
HIPS	80-110 °C	Enclosed
Nylon	60-100 °C	Enclosed
Glass filled Nylon	60-100 °C	Enclosed
Carbon filled Nylon	60-100 °C	Enclosed
PP	60-90 °C	Enclosed
Glass filled PP	70-100 °C	Enclosed
PC	90-120 °C	Enclosed/Actively Heated
TPU/TPE	50-100°C	Enclosed
PEEK	150 °C	Actively Heated
PEKK	150 °C	Actively Heated

2. Once the temperature at which testing will start is determined, let's call this T_0, one can print a test print at this temperature, wait for it to finish (if it finishes without detaching) and measure the warp at each end or corner of the print. If the print does not succeed, the point at which the print became completely detached from the build plate can be used as a data point (Figure 13).

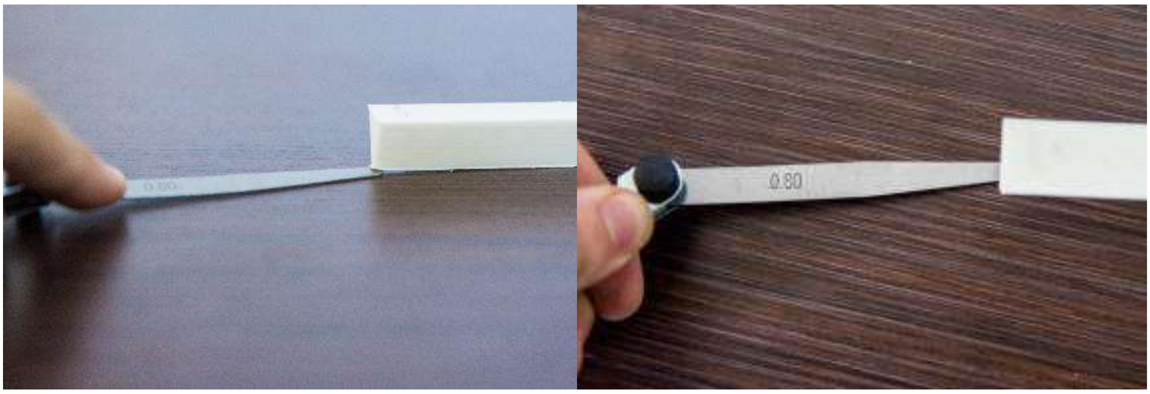


Figure 13: Measuring the warp on the edge of the wedge print using a feeler gauge

3. After this one can perform the same test using the same settings at different build plate temperatures **in steps of 10 °C above T_0.** As the temperature is increased for each successive print, the amount of warp will change, at a certain point the amount warp will start to increase.

4. If the warp continues increasing on the next increment for the build-plate temperature, one can safely assume that further increments in temperature will lead to an increase in the amount of warp.

5. The test is continued starting from **10 °C below T_0** and decreasing in steps of 10 °C. Again, if at a certain stage the warp will start increasing as the test print is performed at lower build-plate temperatures, one can stop testing.

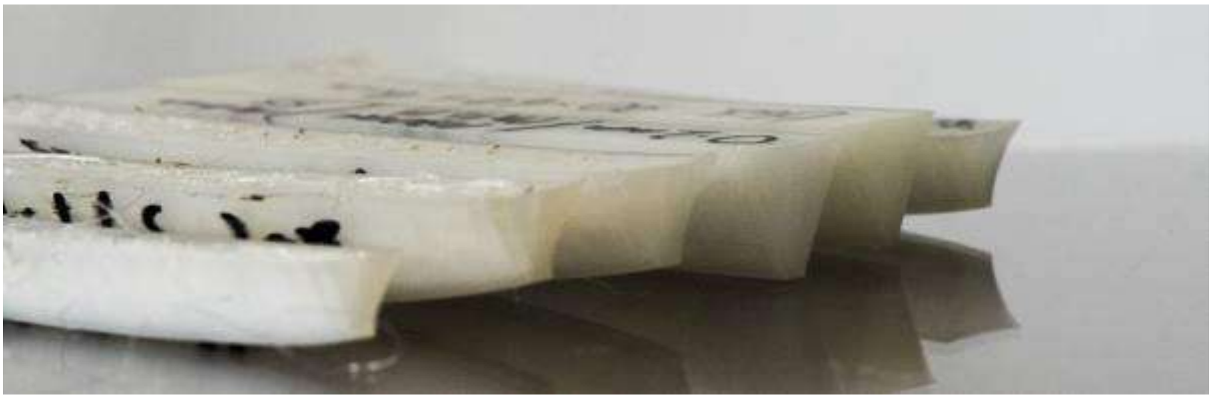
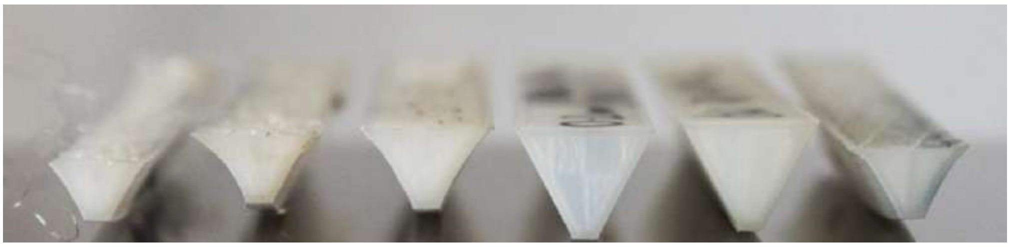


Figure 14: Wedge prints at different temperatures showing different amounts of warp and/or rates of completion

6. Once all the tests are performed the temperature at which the print with the least warp can be determined and that is your ideal build-plate temperature for the adhesive, material and printer combination (Figure 15).



Build-plate Temperature	40 °C	50 °C	60 °C	70 °C	80 °C	90 °C
Print progress (%)	50	69	100	100	100	76
Average warp (mm)	n/a	n/a	2	0.2	1	n/a

Figure 15: Results for bed temperature optimisation of a Nylon filament with Magigoo PA on the Ultimaker S5, the optimum bed temperature is around 70 °C

A blank template for noting and keeping record of the optimum build-plate temperature can be found in the next page.

Build-plate temperature optimisation test sheet

Filament:
Printer:
Adhesive:

	T_0 -40°C	T_0 -30°C	T_0 -20°C	T_0 -10°C	T_0	T_0 +10°C	T_0 +20°C	T_0 +30°C	T_0 +40°C
Build-plate Temperature (°C)									
Print progress (%)									
Average warp (mm)									

Additional Comments:

Filament:
Printer:
Adhesive:

	T_0 -40°C	T_0 -30°C	T_0 -20°C	T_0 -10°C	T_0	T_0 +10°C	T_0 +20°C	T_0 +30°C	T_0 +40°C
Build-plate Temperature (°C)									
Print progress (%)									
Average warp (mm)									

Additional Comments:

Improving adhesion further

With some engineering materials, printing at the optimum build-plate temperature and using a designated build-plate adhesive might still not be enough to prevent warping. Some materials will benefit from a heated build chamber which reduces the thermal stress however other method exists to mitigate warping and aid first layer adhesion. These include:

Application

Before application one must make sure that the build-plate is cleaned properly and free from oils and detergents which will negatively impact adhesion. Care must also be taken to apply the adhesive evenly. (Figure 16)

Figure 16: Apply Magigoo in an even layer on the area for printing

Using slightly higher temperatures for the first layer:

An additional 5 °C, 10 °C or even 15 °C on the build plate over the base printing temperature can often help with improving the adhesion on the first layer, the build-plate can then be turned down to the base temperature to avoid problems associated with the build-plate being too hot. The same principle can also be applied for the nozzle temperature, an additional 5 °C - 15 °C on for the first layer usually aids with improving the interaction between the first layer and the adhesive.

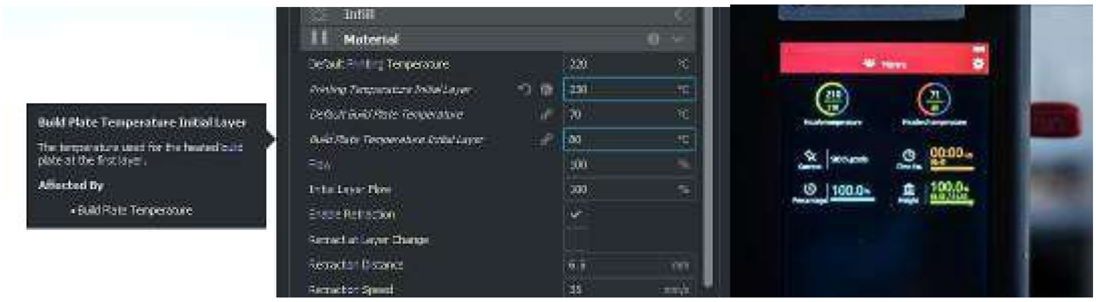


Figure 17: Adjust the first layer temperature settings from your printer or slicing software for better first layer adhesion

Using a brim

With certain materials such as PP, some Nylons, some Polycarbonate materials, and Glass reinforced PP, a brim is an absolute necessity with most parts (Figure 18). For example, polypropylene materials and some nylons show poor adhesion, are flexible, crystalline and have a high shrinkage rate, and thus are very susceptible to warping. This means that a wide brim will be needed to keep the part from peeling away from the build plate. On the other hand, materials such as polycarbonate are very stiff and pull on the build-plate with large amounts of force, a brim will help distribute this force on the build plate and thus reduce warp. When using a brim, it is also recommended to use the largest first layer height possible since a thicker brim is stronger and will thus be more effective against warping.

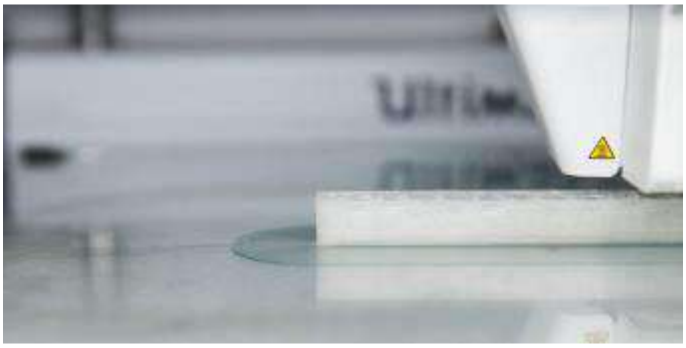


Figure 18: Using a brim helps mitigate warp with materials that are prone to warping.

Tweaking other first layer settings

Slowing down the speed for the first layer helps improve the interaction of the melted plastic with the adhesive layer and thus will aid in first layer adhesion. Similarly, **slightly** over-extruding the first layer also improves first layer adhesion (Figure 19).

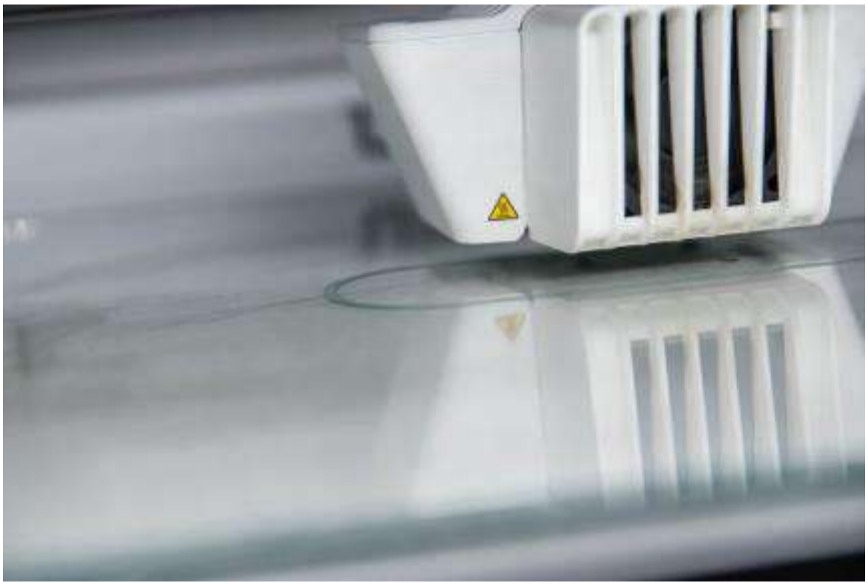


Figure 19: Tweaking the first layer extrusion settings can help with improving first layer adhesion

Turning off the build-plate after the first layer and using the cooling fan for certain materials

Some materials such as glass filled polypropylene and some flexible materials behave differently from other materials. These materials, counterintuitively benefit from a cooler environment while printing. This can be brought about by disabling the heated build-plate after the first layer is completed or using an active part cooling fan. By cooling the build plate, both the base of the print and the build plate adhesive become stiffer and thus restricting the uppermost layers from further deformation of the print. While this solution is highly effective for some filament types it can be disastrous for other filaments such as ABS and Polycarbonate, which tend to detach once the build plate cools.

5. What causes build-plate damage

Regular and seasoned FDM printer users have probably experienced the problem of chipped glass. After failed prints and warping, this is a problem that can disrupt your workflow and cost you dearly. Chipped glass be caused by a number of variables including material and adhesive choices, and printing conditions. However, there is one primary factor that causes glass build-plates to break and it all has to do with thermal contraction.

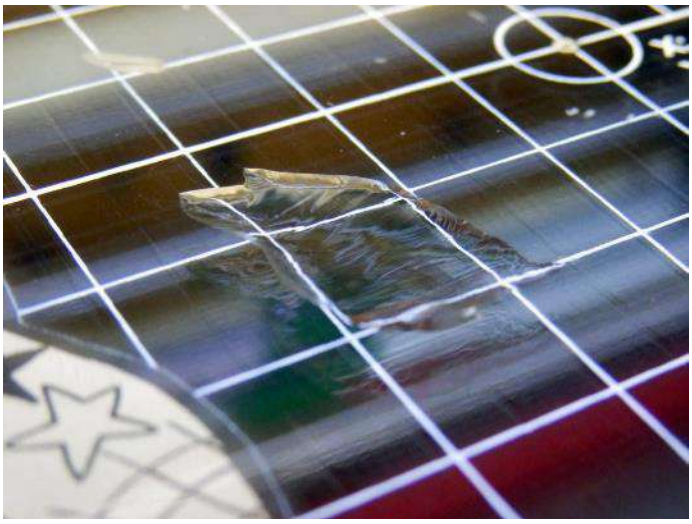


Figure 20: Chipped glass plate, image via richrap.blogspot.com

Thermal Shrinkage and Chipped Glass

Glass has a linear CTE of around 1-17 x 10^{-6} K while that of 3D printing thermoplastics lies in the range of 40 to 200 x 10^{-6} K. The linear CTE of glass is thus about an order of magnitude lower than that of the printed thermoplastic. As parts cool, a tensile force is exerted on the surface of the glass while the parts are still adhered to the glass. If the parts are adhered too strongly to the build-plate, they will continue to exert a tensile force as they cool down further (Figure 21, Figure 22).

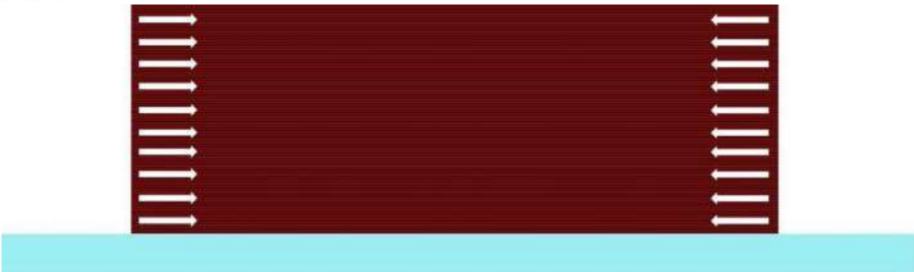


Figure 21: Printed part which has just finished printing, the part and the build-plate are roughly at the same temperature. The part is already exerting some force as a result of shrinkage during the solidification of the deposited FDM thermoplastic

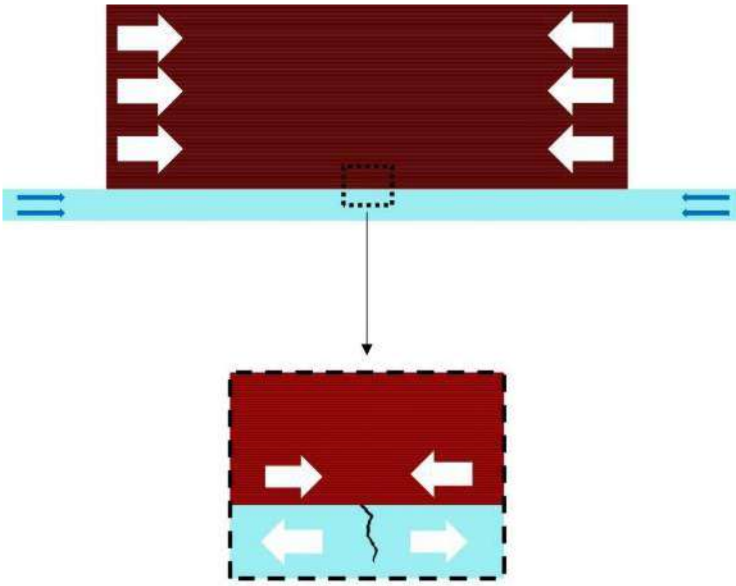


Figure 22: Part and glass build-plate cool down. Since the part experiences a larger rate of shrinkage than the glass build-plate the net result is a tensile force on the glass surface which leads to crack initiation and propagation.

To the naked eye a sheet of glass appears to have a perfect and smooth surface. However, at the microscopic level glass has defects. These defects can either result during the manufacturing process, or else can be introduced during use in the form of scratches. These can act as points of stress concentration and thus increase the likelihood of microscopic cracks to form. Since glass is a brittle material, tensile forces exerted on the surface will cause the cracks to propagate, eventually leading to failure. This can either lead to small piece of glass chipping off the build plate or to a catastrophic failure of the whole glass sheet in some cases.

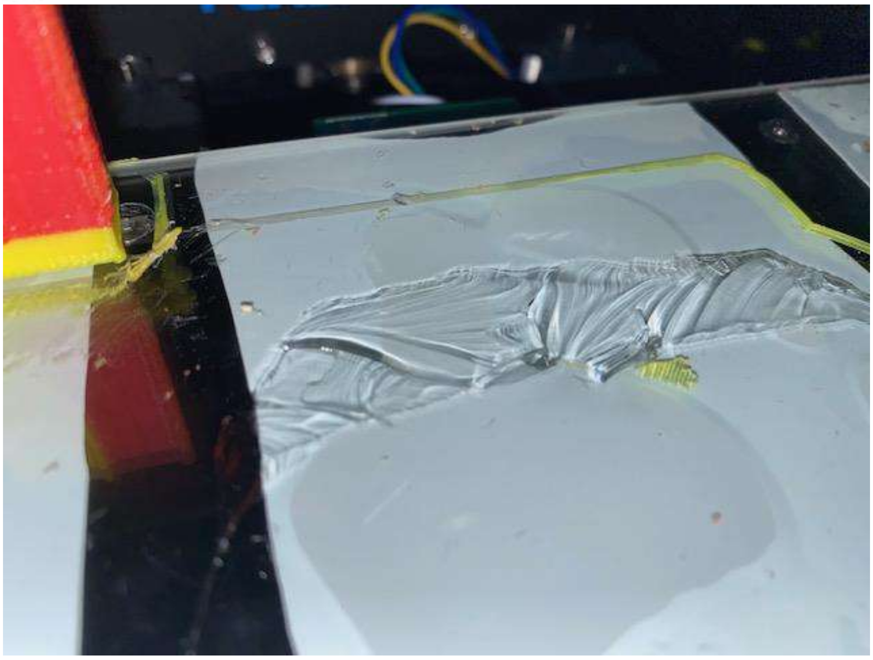


Figure 23: A more severe case of glass chipping resulting from the use of improper adhesives, in this case using Magigoo PA with ABS filament

Thus, the combined action of thermal shrinkage and the adhesion of the part to the glass can cause enough force to damage the glass surface. Generally this is not a problem, especially with consumer grade filaments and non-demanding printing conditions. Nonetheless several factors can increase the probability of chipped glass.

What makes glass prone to chipping?

Materials

Most users experience problems with chipped glass on switching to PET-G after working 'chip free' with PLA. Unlike PLA, PET-G sticks aggressively to glass, and also PEI, increasing the chance that the build-plate will be damaged. As with PET-G some materials can be more prone to damaging glass build-plates due to over adhesion.

Figure 24: A PET-G print causing a catastrophic failure on a glass build-plate, image via richrap.blogspot.com

These include:

- PET/PET-G/CPE, PET type materials are very stiff and tend to stick too well to glass and other surfaces in a similar fashion to PET-G.
- Polycarbonate, PC is a very strong and stiff material. While sometimes it has difficulties adhering to glass, the high strength of the material can pull on the glass with so much force to chip or completely crack it.
- ABS, our experience with ABS is generally positive, however some brands of (possibly modified) ABS materials tend to stick too well the build-plate. This makes the part removal process more difficult and can lead to glass chipping.
- High temperature materials, HT materials such as PEEK require very high build-plate temperatures. Additionally, these materials tend to be quite stiff. Thus, the high temperature drop combined with the high stiffness of these materials can spell disaster for glass build-plates.
- Modified filaments, some filaments such as PLA, Nylon and ABS are modified to improve printability and reduce warping. This can cause these types of filaments to over-adhere.

Printing conditions relating to chipped glass

Glass cracking can also be result from the printing conditions which can promote over adhesion or accelerate wear on the glass build plate. These include:

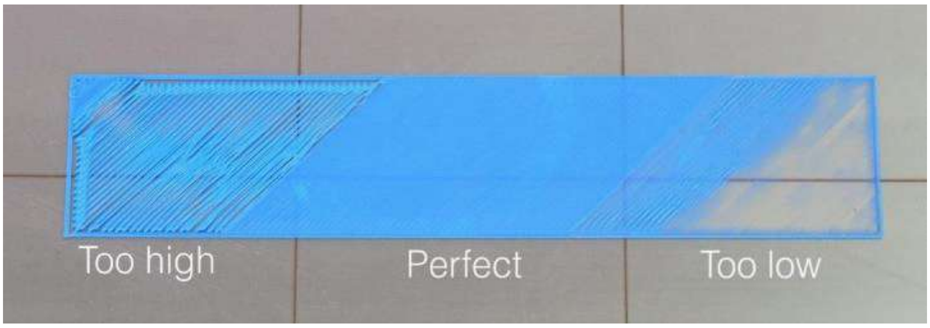


Figure 25: Printing with the nozzle far too close to the build-plate

- Printing too close to the build-plate: Printing too close to the build plate will increase the chance that the filament over-adheres to the build surface. Furthermore, it can also directly damage the build-surface or increase the rate of wear on the build-plate.
- Over-extruding the first layer: while slightly over extruding the first layer can help with initial layer adhesion, overdoing it can lead to similar consequences as printing with the nozzle too close to the build-plate.
- Removing the part early or cooling the build-plate quickly: Printed parts tend to self-release on cooling. Especially when using a specialty build-plate adhesive. With most materials it is advisable to wait for the part to cool completely before attempting removal. Attempting to remove the part while it is still hot can cause unnecessary strain on your build surface. This can lead to premature failure. Similarly, cooling the build plate too quickly by taking it out of the printer and putting it on a cold conductive surface can also cause thermal shock and damage the glass sheet.
- Using the incorrect adhesive: Nowadays there are a number of adhesion solutions you can use. Some of them are specific to certain types of materials. Using the incorrect adhesive for your materials can cause the part to over-adhere which can damage your build-plate. Our adhesives are carefully formulated to provide the right amount of adhesion according to the material. Furthermore, Magigoo adhesives ensure that part removal is as easy as possible.
- Using too much adhesive: Using too much adhesive can also lead the part to over-adhere to the build-plate.

Wear and tear

With every print, the surface of glass (and other materials) will deteriorate. Every heating and cooling cycle and part removed will introduce further defects and exacerbate existing ones. This is inevitable and some materials as mentioned above will wear the surface more quickly.

The wear on glass build surface can also be worsened when using sharp tipped tools to remove parts. These might introduce scratches to the surface of the glass which can act as points of stress concentration and eventually leading to cracks and chips.

Figure 26: Sharped tipped metal tools similar to the one pictured above can scratch the glass thus providing with points of stress concentration

6. How to prevent damaging glass build-plates

Ease off on that first layer

Some 3D printing filaments are modified to improve their adhesion and also reduce warping. This can cause the printed part to over-adhere and can lead to build-plate damage. If a specific brand of filament is giving you a tough time during part removal, it is probably a result of over-adhesion. You can slightly reduce the adhesion by tweaking your slicer or printer settings.

By either reducing the first layer flow or slightly increasing the Z-offset, your printer will not flatten the first layer on the build-plate as much as it usually does. This reduces the adhesion and makes part removal less of a chore. Through testing we have found that reducing the first layer flow to about 85% significantly improves the ease of part removal for materials which tend to over-adhere. This value also ensures that the adhesion still adequate to prevent warping.

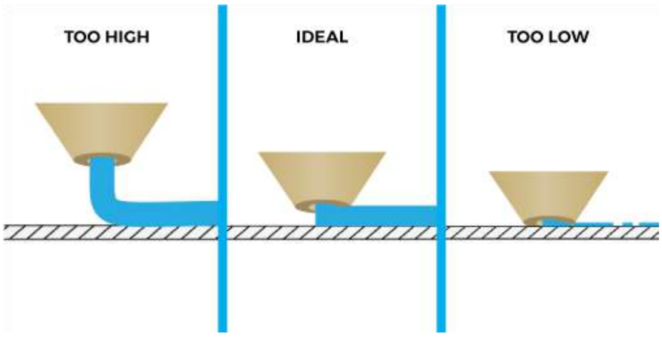


Figure 27: Ideal nozzle height, image via matterhackers.com

Nonetheless, since each printer and material varies we recommend performing some adhesion tests to find the optimum settings.

Be patient!

With most 3D printing materials, it is always advisable to wait for the part to cool before attempting to remove it. In most cases it becomes easier to take the part off once it cools. Furthermore, you run less risk of deforming the part on removal.

One should also always leave the part to slowly cool inside the printer rather than taking the build-plate out to cool it more quickly. Cooling the build-plate too quickly will increase the chances of damaging the build-plate. Some manufacturers also recommend that the end g-code is modified so as to cool the build plate more slowly.

If you follow these steps and you find that the part is still firmly attached to the build plate, do not despair. It is not a good idea to try and wedge the part up using a scraper, as this can damage the build-plate further and also promote glass chipping. If using Magigoo, you can wet around the perimeter of the part with water. After waiting for the water to seep in under the part, it should become much easier to remove. Alternatively, you can submerge the part and build-plate in warm water and wait for it to self-detach.

Incidentally this method is also good for removing fragile parts or parts made out of soft materials such as TPUs which tend to stick too well. In this case Magigoo is acting as a releasing agent!

Use your adhesive, as intended!

The Magigoo range comes in a variety of different flavours for different types of materials. This includes the original Magigoo for ABS, PLA, PET-G, HIPS and TPUs. Magigoo PA for Nylon, Magigoo PC for Polycarbonate and Magigoo PP and PPGF for polypropylene and glass filled PP materials respectively. More recently we introduced Magigoo HT for high temperature materials and Magigoo Flex for certain classes of TPEs. It is important to use the correct Magigoo stick according to the material you are printing. For more information you can visit the tested materials section on our website. **Visit:** **https://magigoo.com/tested-materials/**

Other useful tips:

- Only use a thin layer of adhesive unless instructed otherwise
- Always apply Magigoo on a clean build-pate, do not apply Magigoo on another coat of adhesive
- Avoid printing on the same application when using Magigoo adhesives, since this can increase the chances of damaging your build-plate

Take care of your build surface

As mentioned previously, after repeated use build surfaces wear down and will become more susceptible to damage. By following these precautions, you can extend the life of you build-plate and prevent premature failure:

Figure 28: The use of sharp-tipped tools can scratch your build-plate surface

- Avoid using sharp tipped tools that may scratch the build-plate
- Avoid printing with the nozzle too close to the build-plate
- Keep the build-plate clean and free from debris
- Cool the build-plate slowly
- Higher quality borosilicate glass build-plates tend to be more resistant to chipping

7. Other Magigoo FAQs

The print is still warping what can I do?

There are several reasons why this can happen, these include:

- Using the incorrect settings – Magigoo® has an optimum temperature range where it offers the best adhesive performance – a guide on how to tweak your settings can be found in previous sections of this document.

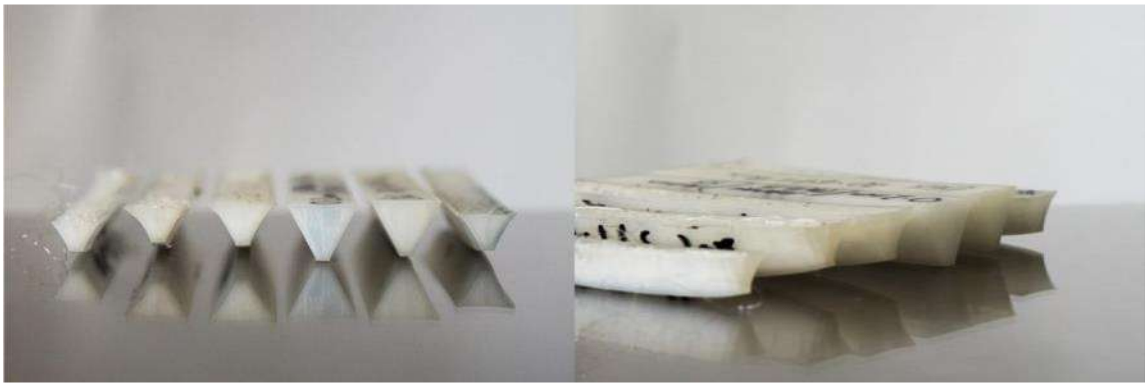


Figure 29: Prints using the same material warping to different extents when being printed at different bed temperatures.

- Not using a brim – Some materials still require that a brim is used in combination with the optimum settings for best performance. In some instances, for example with Cura, it is also possible to use a brim on the inside perimeter of the part as well (Figure 29).

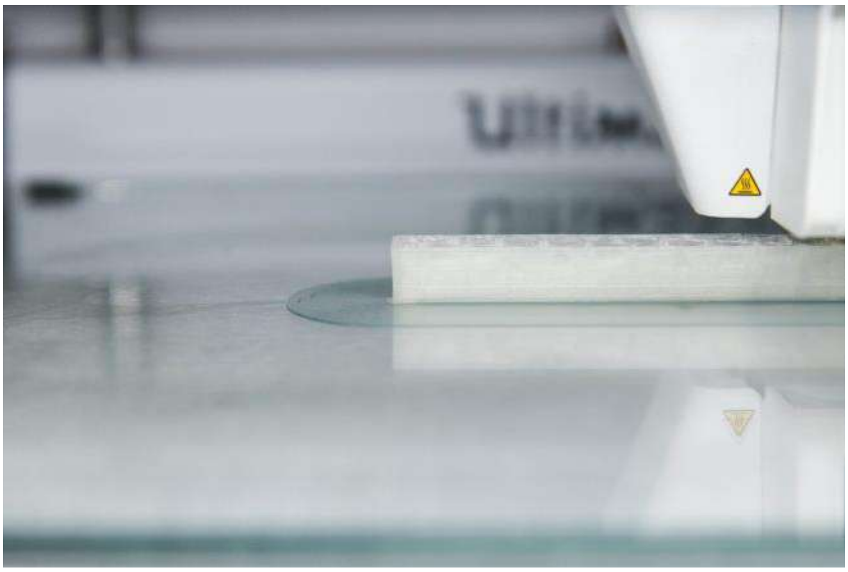


Figure 30: **Printing using a brim helps improve first layer adhesion.**

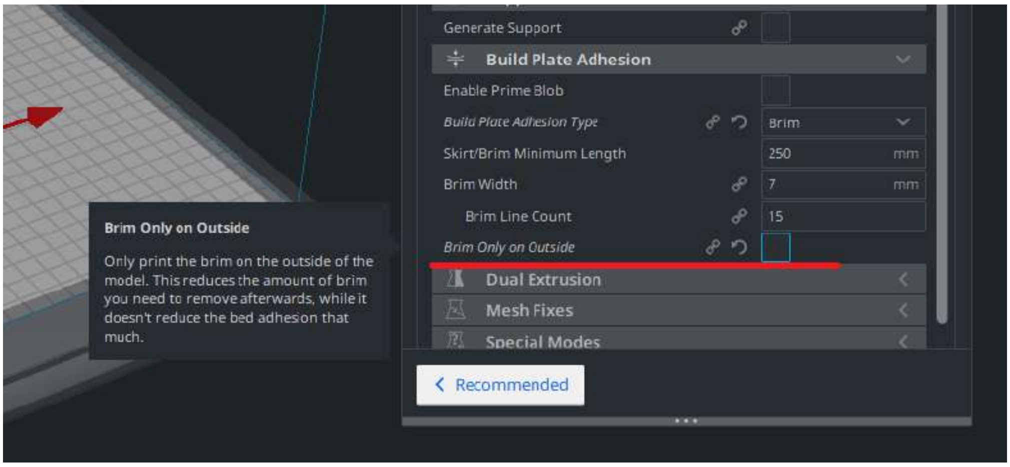


Figure 31: **Unticking the "Brim only on outside" in Ultimaker Cura will enable a brim in the inner perimeter of your part where possible, this helps improve adhesion further!**

- Incorrect first layer height – if your nozzle is too far away from your bed or else way too close, the adhesion of your 3D prints will not be ideal.

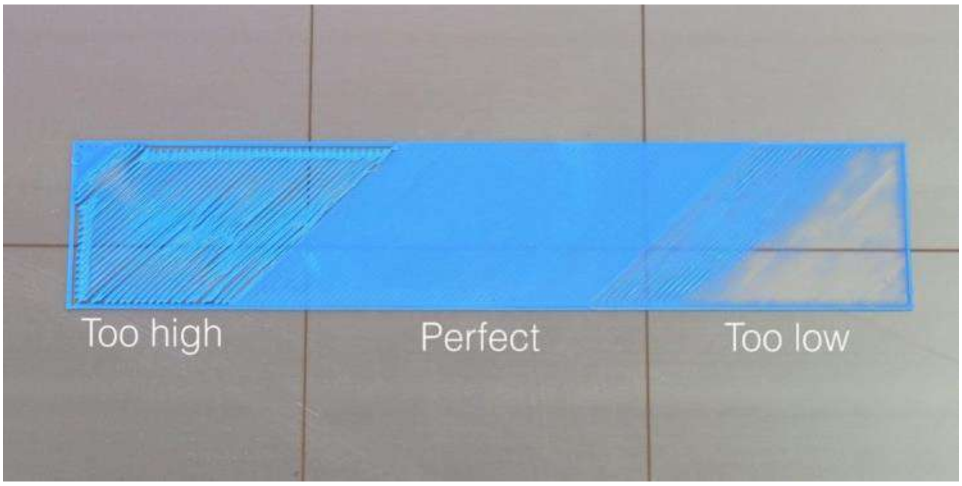


Figure 32: Image showing results of the nozzle being too far away, at an ideal level and too close from/to the build plate – image via Prusa Printers

- Using a cooling fan when you shouldn't – some materials shrink considerably on cooling. It is good practice to keep the cooling fan off for the first 5 to 10 layers of your print.
- Using a first layer speed which is too high – Some materials such as ASA, PC and PP need some time to interact with the build-plate adhesive, using speeds which are too high can cause poor first layer adhesion. As a general rule we recommend 20 mm/s or lower for the first layer.

The part is stuck to the build-plate and I can't remove it?

There are several reasons why this can happen depending on the material and printing parameters:

- Printing too close to the bed or with a very high material flow on the first layer, 'pushes' the material into the adhesive layer and/or the build-plate substrate. This tends to cause over adhesion which will make part removal very difficult.
- The material to adhesive combination does not 'release' on cooling. For example, TPUs, TPEs, some types of PLA, some types of Nylon and Glass-filled PP with Magigoo® PP-GF do not become easier to remove even upon cooling.

While it is tempting to grab a spatula and potentially convert your 3D printing space to an abattoir, there are several methods which can be used to safely remove a printed part which is too stuck to the build plate:

- Make sure that the build-plate is completely cooled down before attempting removal. Sometimes the temptation to remove a print is too high and one can easily forget that Magigoo® sticks when hot and releases when cold. A part removal tool can be used if the part is partially detached or it can be easily and safely slid under the part, gentle tapping on the sides of the part can also help with the release of the part.
- If even after the print has completely cooled down, it is still stuck, one method to help with part removal is to use water. One can either wet around the part with water or else submerge the whole build-plate under water. If the part has a brim it is suggested to first peel off the brim from around the part before wetting the build-plate. The part should be easy to remove after 30 minutes, if not a spatula can be carefully slid underneath the part to introduce water to the areas which are still stuck or by submerging the part for a longer time. This method is a useful way to remove delicate parts, made out of flexible or soft materials.
- Glass filled PP parts printed on Magigoo® PP-GF work differently from regular materials. As the heated bed is deactivated after the first layer of printing, the build-plate is already cool. In order to facilitate part removal one can, turn back on the build plate and heat it to a temperature of between 70 and 90 °C after the print is complete and remove the part when the build-plate is hot.

Magigoo® is not flowing out, the foam tip has dried up

The Magigoo® pen has a spring-loaded valve in the applicator, this means that when applying Magigoo® the applicator has to be pushed against the print bed for it to be activated. Please also note that the seal of the valve has to be broken when the Magigoo® pen is first used, which might require slightly more force the first time. (Figure 33)

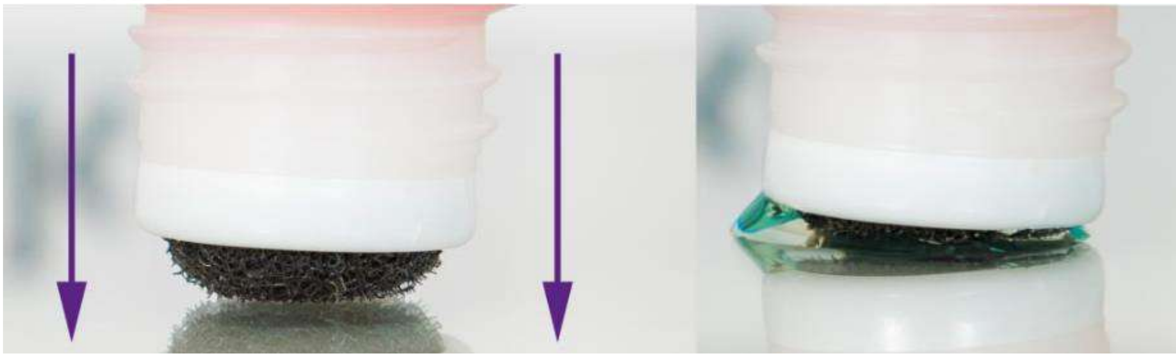


Figure 33: Depress nib on build plate to activate nozzle, Magigoo should then easily flow out!

- Also note that after long periods of not being used or if the screw on cap is not used, the residual Magigoo® in the foam tip can solidify and clog the nozzle. This can be easily unclogged by wetting the foam tip with some water.
- DO NOT under any circumstance press the sides of the bottle with any more force than a gentle squeeze – if Magigoo® still does not flow out, a very light squeeze can be used to help the Magigoo® flow out. If you squeeze the bottle too hard, the applicator can pop out, followed by the contents of the bottle.

Can I use the same layer of Magigoo®?

In order to achieve the best print results and the highest reliability when 3D printing, we always recommend to re-apply Magigoo before each print. This is especially so when using high performance materials such as PC, ASA, PEEK, and reinforced Nylons, given the higher cost of a failed print. Furthermore, re-using the same application increases the chance of a failed print or glass chipping. A Magigoo® bottle should last well over 100 full size prints on a medium size printer and even longer if you apply only on the area which is going to be used for printing. This means that on average the cost of reapplying Magigoo® per print is less than $0.20, well worth it, when considering the material, time and energy cost for a single failed print.

Having said this, various users have reported that they have used the same Magigoo® application multiple times. With regular printing materials such as PLA the part removal process does not damage the adhesive layer much which means that it can be re-used. With other materials such as GF-PP the adhesive layer will be damaged on part removal and cannot be re-used. With materials such as PP and most Nylons, the layer can be used until it is visibly damaged.

With the low cost per print for Magigoo® we recommend that it's better to be safe than sorry and re-apply before each print. This will ensure the best results and help to prevent failed 3D prints, those relating to first layer adhesion anyways.

How do I store Magigoo®?

Magigoo® is best stored upright, in a cool dark place. The screw cap should be tightly secured on the bottle when not in use and the applicator should be kept clean, free form any residual glue or dust. It is also recommended to store Magigoo adhesives away from direct sources of heat such as 3D printers, ovens and filament drying equipment.

Should I clean before applying Magigoo®?

Yes, make sure that the build-plate is free from dust, debris, previous adhesion products and oils. It is best to avoid detergents when cleaning your build-plate as these might interfere with first layer adhesion. Cleaning with water should be enough, however recommend using ethanol or isopropanol if grease or oil is present on the build-surface.

Why are larger prints warping?

In most cases you can be flawlessly printing smaller sized parts with no warping issues, however on attempting to print larger parts, the parts start warping. This normally happens due to a couple of reasons: firstly, since larger parts tend to warp more due to their size and additional printing materials. Large prints have a higher tendency to warp, by using Magigoo build-plate adhesives and fine-tuning your first layer settings the amount of lift at the corners can be practically eliminated.

Secondly some printers tend to have the build plate colder at the edges, especially in non-enclosed printers. This can lead to prints lifting towards the colder areas of the build-plate. In this case the warping can be controlled by using an enclosure or by increasing the build-plate temperature.

What is the best adhesive for TPU/TPE/TPC?

With regards to adhesion TPEs generally tend to show two types of behaviours. TPUs typically adhere too well to most printing surfaces, this means that while adhesion is not an issue during printing, removing a print can be quite difficult. As a result, there is a good chance that the soft and flexible part can be damaged during print removal. In the case of TPUs Magigoo Original can be used since it generally provides enough first layer adhesion.

On the other hand, other TPEs and TPCs usually do not stick too well to most build-plate surfaces which means that warping will be a significant issue. In this case Magigoo Flex is required to provide the additional adhesive force needed. As a first layer adhesive it provides a strong adhesive base for high performance TPEs such as DSM Arnitel, DuPont Hytrel and other TPCs and TPEs. On the other hand it ensures that TPUs and other flexible materials which tend to stick too well can be very easily removed from build surface just by the application of water.

My CPE/PET materials are warping, what can I do?

PET - Polyethylene Terephthalate or CPE - Co-Polyester type materials describe a number of polymers which can vary significantly in chemical composition and thus in physical and chemical properties. As a result, different CPE/PET type materials might require different printing settings.

Nonetheless in the majority of cases most CPE materials will benefit from using slightly higher build-plate temperatures when using Magigoo Original.

Why is the 120mL bottle not better value than the 50 mL bottle?

The 120 mL bottle is a specialty product designed to be used on larger printers due to its bigger applicator. Unlike the 50 mL bottle the 120 mL bottle is produced in much smaller quantities due to its niche application. As a result, the 120 mL bottle is much more expensive to produce per unit leading to its higher cost.

Magigoo now has a blue applicator tip, why?

The applicator tip of the Magigoo bottle has been recently changed from a black foam tip to a more durable blue fabric material. The new material is more resistant to wear and is also more temperature resistant. Nonetheless, these precautions below should be followed to extend the lifetime of the applicator tip:

- Always apply on a clean bed
- Never apply when the bed temperature is above 40 °C
- Do not apply excessive amounts of glue, a little bit goes a long way
- Keep the cap on and tightly secure when the bottle is not in use

- If you see any build-up of goo or gunk on the applicator tip or the inside of the screwcap you can clean it using a few drops of water to dissolve the goo and a damp cloth to wipe it off

Kindly contact us on feedback@magigoo.com should you experience any issues with the Magigoo applicators.

Which is the best way to apply Magigoo®?

Magigoo® is best applied on a clean, cold bed and re-applied to the cleaned printing area prior to each print.

- First shake the Magigoo® bottle vigorously for 5-10 seconds
- Carefully open the cap keeping the bottle in the upright position
- Inspect the foam applicator tip for any residue or damage (clean any residue from the tip if necessary)
- Invert the bottle and push the nib against the build-plate at one corner of the print area. Magigoo® should start flowing out when the valve inside the applicator is activated, if not a very gentle squeeze can be applied so that Magigoo® starts flowing out.
- Keeping the valve activated, apply the Magigoo® by moving the bottle in a to-and-fro motion to cover the whole printing area, horizontally (left to right).
- Then smooth out the adhesive layer by going over the printing area once more, without activating the valve. This time move vertically (from top to bottom). This also makes sure that you do not leave any missed spots.
- One can repeat the smoothing steps (alternating between horizontal and vertical to and fro motions) until the desired smoothness is achieved. A smooth Magigoo® application will give the bottom of the print a mirror finish (a rough application can show as a rougher finish at the bottom of the print).
- The Magigoo® layer will dry up before printing commences, in most cases the Magigoo® layer dries completely until the build-plate heats up.
- You can now print your part. Your 3D print will stick well during printing and should pop right off after printing when the bed cools back down.

8. Compatibility of Magigoo Adhesives with Soluble Materials

The compatibility of water-soluble materials with other FDM materials depends on the filament type. Soluble materials are designed to be printed at a specified temperature range depending on the filament type. In most cases when printing with soluble support material most users opt to print the both the part and support material directly onto the build-plate. This has the advantages of:

- Improving surface finish
- Minimising printing time
- Minimising post processing time
- Minimising adhesion issues between the part and the support material
- Minimising the amount of support material used

Having good compatibility between adhesives and the printing material lets users print a range of materials in combination with the soluble material whilst ensuring that both the printed part and the support material will adhere to the build-plate. The tables below show the compatibility of Magigoo® adhesives with different support materials.

Table 2: Compatibility of Kuraray MOWIFLEX support materials with Magigoo® adhesives

Magigoo Type	Temperature range	
	MOWIFLEX 3D 1000	**MOWIFLEX 3D 2000**
Original	25 °C - 75 °C	25 °C - 120 °C
PA	25 °C - 75 °C	25 °C - 120 °C
FLEX	25 °C - 75 °C	25 °C - 100 °C
HT	/	75 °C - 120 °C
PC	/	75 °C - 120 °C

Table 3: Compatibility of Aquasys 120 support material with Magigoo® adhesives

Magigoo Type	Temperature range
HT	70 °C – 120 °C
PC	80 °C – 120 °C
Original	70 °C – 120 °C
PA	50 °C – 120 °C
FLEX	50 °C – 100 °C
PP	Not recommended
PPGF	50 °C – 100+ °C

9. An introduction to thermoplastic materials in FDM printing

Polymers are made of large molecules with a repeating chemical structure, otherwise known as a monomer. The mechanical and chemical properties of a polymer are affected not only by the chemical make-up of the repeat unit but also the molecular arrangement of the repeat units in the polymer chains, and, the interactions between these polymer chains. In thermoplastic materials, the polymer chains are held together with weak intermolecular forces. As a result, thermoplastic materials can be repeatedly softened from a solid to a viscous fluid on application of heat.

Currently most of the thermoplastic material types available for injection moulding are also available for 3D printing, with more materials being adapted for FDM AM each day. Figure 34below shows the common material classes available for 3D printing. The materials are split into three different classes: ***commodity, engineering*** and ***high temperature***, with some materials overlapping the commodity and engineering class (Figure 34).

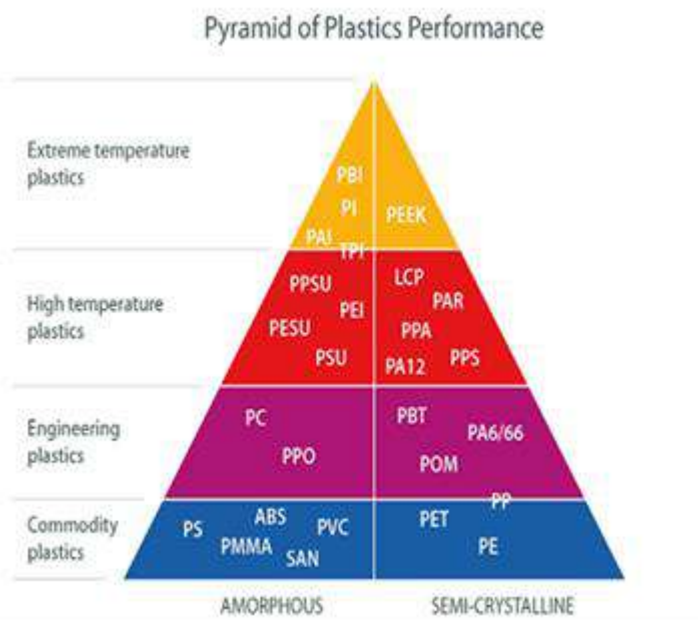
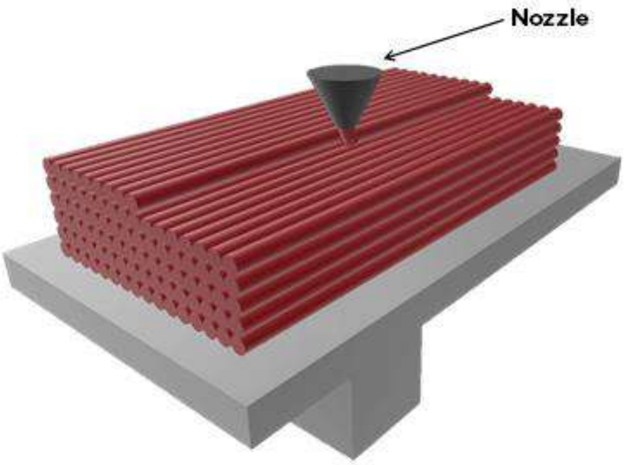


Figure 34: Classification of thermoplastic materials via: specialchem.com

Generally, engineering materials tend to require higher nozzle and chamber temperatures than commodity plastics during printing, since these materials tend to have higher melting temperatures. Printing at lower than recommended temperatures will potentially lead to print failure or parts with inferior mechanical properties. Similarly, high temperature thermoplastics require even higher nozzle and working temperatures for printing. These materials often require an actively heated build chamber during printing for reliable results.

FDM printing and its pitfalls



Due to the layer upon layer nature of FDM parts (figure 2) tend to have voids or empty spaces between the layers which are detrimental to the mechanical properties of the final part. As a result, the bond strength between layers of extruded material is often cited as being the weakest and most critical parameter affecting the mechanical properties of FDM parts. In other words, 3D printed parts are often much weaker in the Z-direction than in the X- and Y-direction.

The adhesion between printed layers is achieved through thermally driven diffusion welding. Thus, different fabrication parameters including nozzle temperature, environmental temperature, print speed, nozzle width, extrusion width, layer height and others will have an effect on the resulting mechanical properties of a printed object. Since different materials have different temperature requirements, the material type will also affect the z-strength of a printed part. In fact, higher temperature materials are more prone to poor z-strength and delamination if the printing parameters are not adequate.

Common 3D printing materials - are they good enough?

One of the misconceptions one may hold is that PLA is only suited for aesthetic prints and that functional prints using PLA were not feasible. This is due to the fact that often parts printed with PLA tend to break easily especially in areas with thin sections. This would lead one to believe that PLA is not a very strong material. However, when looking at the typical mechanical properties for this material in Table 4 below, one can easily see that it is stronger than most common 'engineering' materials in terms of Ultimate tensile strength and also in terms of stiffness.

	PLA	PET-G	ABS	Tough PLA
Nozzle (°C)	205	240	255	210
Bed (°C)	60	80	100	60
Tg (°C)	61	81	101	60
Strength (XY - MPa)	46.6	31.9	33.3	41
Strength (Z - MPa)	43.5	13.4	25.4	33
Stiffness (XY - MPa)	2636	1472	2174	2450
Stiffness (Z - MPa)	/	1087	1835	2200
Ductility (XY - %)	1.90	6.8	2.7	20
Ductility (Z - %)	/	1.3	2.4	2.6
Impact Strength (XY - kj/m²)	2.68	5.1	12.6	22.9
Impact Strength (Z - kj/m²)	/	/	10.5	2.2

Table 4: ***Typical mechanical properties of common 3D printing materials***

So why does PLA break so easily?

This is probably a result of a combination of factors including:

- PLA has very low ductility and is thus very brittle. In layman's terms, this means that PLA does not like to bend and tends to snap like glass
- The inherent inferior mechanical properties in the z-direction as discussed above - *especially when using low nozzle temperatures and active part cooling. This usually results in prints with a smoother finish and less artefacts at the cost of Z-strength*

- Design which does not take into consideration of these factors. While some designers take into account the material and manufacturing limitations, others don't. The relative ease by which even inexperienced users, (such as myself starting out) can design and print parts means that in most cases these design requirements are overlooked.

As a result, when all these factors are combined, the resulting printed parts will be fragile and unsuited for functional applications. Nonetheless with a good design approach and by slightly tweaking printing and slicing settings, one can get very strong parts out of PLA which will endure years of hard use, PLA also has the advantage of being easy to finish, paint and is also relatively easy to machine.

With that being said PLA still has its limitations especially in applications which require repeated loadings and also large extents of flexure and plastic deformation during use. The major drawback of PLA is however its relatively low temperature resistance. PLA has its glass transition at around 60 °C or lower. This means that at extreme ambient temperatures, such as in a hot car during summer, the material can easily become soft enough to deform, even under light or no stress. This limits the practical applications of PLA somewhat.

How about ABS?

ABS is generally regarded as being more suitable for general use plastics and everyday commodity materials. This is due to the fact that ABS is a copolymer, more accurately a terpolymer of Acrylonitile, Butdiene and Styrene. Without going into too much detail, the mixture of these three monomers gives ABS respectable toughness and impact resistance and an operating temperature of between -20 °C to 80 °C which suits the typical usage scenarios of most everyday items. Coupled with its relatively cheap price, and ease of production and post processing ABS quite frequently used as a thermoplastic.

More importantly ABS was probably the most common material that was widely available for FDM printing during its early years leading to its popularity in 3D printing. Nowadays while still widely used, and still a go-to material for 3D printing, it faces stiff competition from other materials such as ASA and PET-G in generic cases and other engineering materials in more specific applications.

Apart from its versatile material properties ABS has several advantages when it comes to 3D printing. These include the aforementioned ductility which means that parts are less much less prone to brittle, unpredictable failure and the material's higher temperature resistance when compared to PLA. ABS is also one of the cheapest and most readily available 3D printing filaments. Furthermore ABS is soluble in acetone, as a result parts can be 'glued' together using acetone and also vapor smoothed to remove the appearance of layer lines.

ABS also has several disadvantages when it comes to 3D printing. These include the material's tendency to warp; however, this issue has been long solved with the use of build-plate adhesives such as Magigoo Original. Since the material has a higher temperature requirement it is also recommended that an enclosed or actively heated printer is used with this material for printing larger parts. Otherwise there is a greater chance for the resulting prints to exhibit delamination and inferior mechanical properties in the Z-direction. As a result there is a greater chance that parts printed in this material break easily if the environmental temperatures are not accounted for. ABS is also known for its characteristic smell when printing. In fact ABS releases toxic fumes when heated and it is recommended to print this material in a well-ventilated area.

ASA

ASA (Acrylonitrile Styrene Acrylate) is a material which is quite similar to ABS in terms of mechanical and thermal properties. In fact, in most cases ASA can be printed using similar temperature settings and printing conditions to those used for ABS. The advantages of ASA is its vastly superior resistance to UV rays and water and chemicals resistance. This makes the material better suited for external parts.

PET-G

PET-G (Polyethylene Terephthalate - glycol modified) is another popular 3D printing material that can be considered as the middle ground between ABS and PLA. While still being relatively easy to print, with a glass transition temperature of around 80 °C, PET-G offers better temperature resistance than PLA. This material is also softer and less stiff than both PLA and ABS, this reduces the machinability of the material, but allows for greater ductility. In fact, PET-G is the one of the most ductile materials which can be easily printed. This makes it especially useful for parts which have to withstand large deformations and makes printed parts very hard to destroy.

Nevertheless, in terms of mechanical strength and stiffness PET-G is the generally the weakest of the bunch and is thus less suited for parts which need to bear higher static loads.

Tough PLA

Another alternative material which is still easy to print and quite popular is tough PLA. This material combines the stiffness of PLA with the toughness of ABS whilst retaining excellent layer adhesion. In fact, this material is ideal for application requiring high toughness and rigidity. It is still easy to print yet it is much less brittle than PLA and thus less prone to unpredictable failure. The downside is that this material has similar thermal resistance as PLA and thus is not suited for high temperature applications.

Engineering and High Temperature 3D printing materials

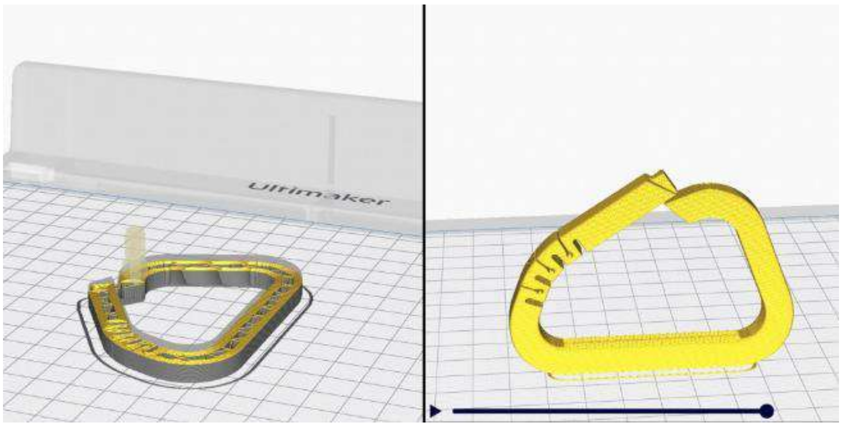
If however the aforementioned materials don't cut it in terms of mechanical, thermal and chemical properties there is a vast selection of other FDM materials available. Apart from offering improved mechanical properties Engineering materials often excel at particular applications. Several classes of engineering materials will be discussed in the next second of the manual!

It is not just the Material!

As previously mentioned, when discussing PLA, sometimes 3D printed parts can fail sooner than expected. Apart from the inherent limitations brought by the layer by layer build several factors affect the final strength of the printed part. However, with some design and process considerations premature part failure can be mitigated. Given that the correct material is chosen for the application depending on the intended use of the end part one also needs to look out for:

Part orientation and design

This is probably the most important consideration when it comes to using the FDM process to produce objects. On one side it is generally preferred to minimize the use of support structures when printing. The use of support structures increases printing time and material usage but more importantly adds an extra post-processing step for support removal. Nonetheless if part integrity is the main requirement, the part must be oriented in such a way as to avoid it being loaded perpendicularly to the build-direction during use.



Printing the carabiner by laying it flat on its side (left) will result in a much stronger print than printing it on its back (right) since in the first case the load will be mainly applied along the layers lines and not parallel to the build direction.

As previously mentioned, another factor which needs to be considered is the design of the part. This article from 3D hubs covers a few of the basics however the design consideration will be highly related to the function of the part. This would require more in-depth knowledge of design in relation to static and dynamic forces.

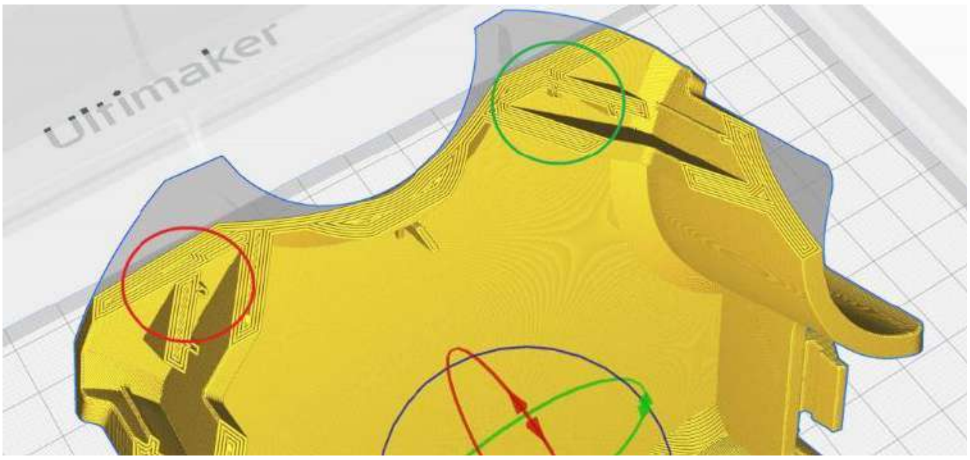
3D printing settings

Once you have chosen the correct material it is also important to find out the correct settings for your printing set-up. The nozzle, build-plate and chamber temperatures, part cooling and print speed will all affect the resulting mechanical properties of the final part, especially in the z-direction. It is thus important to optimize the settings for layer adhesion, there are several articles online on how this can be achieved by calibrating these temperatures. One must also consider if the printing set-up being used is capable of printing the required material at the optimal settings. While some open printers can reach the temperatures required to print polycarbonate for example, this does not mean that large polycarbonate parts can be printed on these types of printers without an appreciable loss in layer adhesion!

Other factors such as extrusion width, layer height, nozzle size, number of shells and infill density also affect the strength of the printed parts.

Model issues

Another point which is worth mentioning are issues with STL files leading to 3D printing artefacts. These include non-manifold edges, holes and gaps, unmerged bodies, and hollow structures. It is thus always a good idea to preview the g-code thoroughly before printing and to look for these errors. This happens more often than you'd think and can leave you wondering why a part is failing, only to find out that it was a problem with the virtual model on the machine.



Looking at the figure above it can be seen that in the case of the part circles in red, one of the clips on the print is being treated as a separate body, even though this was not immediately apparent in the .stl file. On the other hand, the same clip on the other side (encircled in green) is treated as a part of the whole body. It can be immediately observed that in the first case the printing paths for the clip do not incorporate it into the body which means that the clip will fail under much lighter loads than the clip marked in green.

10. FDM printing of Nylon

Why 3D print Nylon?

FDM printed nylon parts can show exceptional mechanical properties, especially when compared to other materials such as PLA and ABS. Nylon is frequently used to print gears and other high wear components and is also used to produce functional prototypes. The table below shows the mechanical properties of some unfilled Nylons compared to the mechanical properties of other common 3D printing materials.

Material	Nozzle (°C)	Bed (°C)	Modulus (MPa)	UTS (MPa)	Elongation (%)
DSM Novamid ID 1030	230	80-120	860	37	110
DSM Novamid ID 1070	260	80-100	2120	50	15
Polymaker PolyMide COPA	250-270	25-50	2223	66.2	9.9
DuPont Zytel	245-295	85-110	1600	40	15
Taulman 910	250	25-50	502.8	55.8	31
Ultimaker Nylon	230-260	60-70	579	34.4	210
Matterhackers Nylon Pro	240-260	60	/	36.2	34
FormFutura STYX-12	240-270	/	1400	60	150

**the values presented in this table were obtained from the manufacturers' websites and data sheets. Since manufacturers can use different testing methods, some values cannot be directly compared and only serve to give an indication of the mechanical properties. Furthermore, the mechanical properties of the printed part will depend greatly on the printing parameters and orientation of the printed part.*

Nonetheless while most Nylons generally offer excellent strength and toughness, Nylon 12 tends to be less sensitive to water absorption than Nylon 6. Furthermore Nylon 12 is said to have better dimensional stability, better thermal resistance, and chemical resistance. However, these properties are usually more dependent on the specific polymer or co-polymer.

How to Succeed when 3D printing with Nylon

While Nylon is a very versatile material and excels in a variety of mechanical, thermal, and chemical properties it does have its pitfalls. Nylon does need some extra care when it comes to printing, handling and storage.

Watch out for Moisture

Printing with wet filament can result in parts with poor surface finish, diminished mechanical properties and excessive stringing. Nylon filaments are among the most susceptible 3D printing materials due to the large amount of water they absorb in a short amount of time. Usually, filament spools need to be dried within a few hours of being left outside. A number of devices exist to keep the filament dry during printing and are especially useful for long prints (>24 hours). Nylon can be dried in an oven for about 8 hours at 80 ˚C.

Both prints were printed on the same printer with the same settings and same material (Taulman Alloy 910) The model on the left was printed using filament which had been previously dried while the one on the right was printed with filament left outside for a couple of days!

Make sure your printer is up for it

With some exceptions most nylons require a nozzle temperature above 250 ˚C. An all-metal, hot end is recommended when printing with this material. Some nylons can also be printed without the use of a heated bed. However other grades of Nylon can sometimes require the use of a heated bed to up to 100 ˚C.

Prevent warping

Since Nylon is a slippery material, this also means that it usually does not adhere well to other surfaces. While there are some exceptions, most Nylons require a specialized adhesion solution to prevent warping. It is also recommended to use a printer with an enclosed chamber and to disable active part cooling. While each filament will require different methods to improve adhesion, other useful suggestions include:

- Use a brim: For some Nylon materials medium sized and large prints can benefit from using a 10-20 mm brim.
- Start high then go low: Some Nylon materials benefit from using an additional 10 ˚C for the first layer on both the build-plate and the nozzle. This helps the first layer adhere better but prevents warping and other problems related to excessive printing temperatures.
- Print slowly on the first layer: Most materials tend to show better first layer adhesion when the first layer is printed slowly (10-20 mm/s)
- Use a heated chamber: Some Nylon materials can benefit greatly from the use of a heated chamber.

11. FDM printing of Polycarbonate

Nevertheless, several polycarbonate filaments which can be printed on consumer and prosumer printers more easily have been introduced to the market. These include easy to print polycarbonate materials such as Polymaker PolyLite or PolyMax PC and Ultimaker PC. Alternatively, PC can also be alloyed with other thermoplastics such as ABS to improve printability whilst retaining most of the excellent material properties of PC.

Why print PC?

The most obvious reason to use PC in your prints is its mechanical strength combined with its excellent toughness which greatly surpasses that of PLA, ABS and PET-G. This enables one to print functional parts and prototypes which can withstand a tough beating! The high temperature resistance, and electrically insulating properties also make this material an ideal candidate for electrical components and enclosures for electronics which tend to get hot during use.

The excellent optical clarity of PC also makes it useful for transparent prints used for both aesthetic and functional uses! Polycarbonate can also undergo large amounts of plastic deformation before failing making it useful for load bearing parts.

Material	Nozzle (˚C)	Bed (˚C)	Modulus (MPa)	UTS (MPa)	Elongation (%)	Impact Strength (kJ/m^2)	T_g (˚C)
Polymaker PolyMax	250-270	90-105	2048	59.7	12.2	25.1	113
Polymaker PolyLite	250-270	90-105	2307	62.7	3.2	3.4	113
Ultimaker PC	270	110	2134	76.4	6.4	/	112
Filament2print PC	250-285	110-130	2400	66	6	/	/
dddrop PC	270-290	130	2350	65	20	36	140

**The values presented in this table were obtained from the manufacturers' websites and data sheets. Since manufacturers can use different testing methods, some values cannot be directly compared and only serve to give an indication of the mechanical properties. Furthermore, the mechanical properties of the printed part will depend greatly on the printing parameters and orientation of the printed part.*

The table above shows the mechanical properties of some FDM Polycarbonate materials compared to the typical properties of other FDM filaments. It is immediately evident that the ultimate tensile strength of PC materials is about 1.5 times that of PLA and double that of ABS and PET-G. Furthermore, PC materials tend to have higher toughness (as indicated by impact strength) and tend to fail under higher rates of strain when compared to other materials. Moreover, Polycarbonate materials tend to have a higher glass transition temperature which indicates better high temperature performance.

How to succeed when 3D printing Polycarbonate

Polycarbonate filament absorbs moisture from its surroundings and thus needs to be dried before printing. Most PC materials will be completely dry if heated to 80-100 ˚C for 4-8 hours in an oven. If you have left your Polycarbonate filament outside of an airtight container for a few days or if you see small bubbles in the filament during extrusion or excessive stringing in printed parts, it is recommended to dry your filament. Printing with wet filament can cause printed parts to exhibit diminished mechanical performance. Furthermore, when printing with transparent filament, excessive moisture will cause the printed parts to become opaque.

As previously mentioned, another consideration when printing PC are the high temperatures required. It is recommended to use a printer which can heat the build-plate to at least 100 ˚C and the nozzle to at least 250 ˚C. For this reason, an all metal hot end is required when printing with PC. It is also recommended to print PC slowly (20-40 mm/s) especially if your printer cannot reach very high temperatures. In fact, printing slowly can prevent layer splitting and can also help reduce warping!

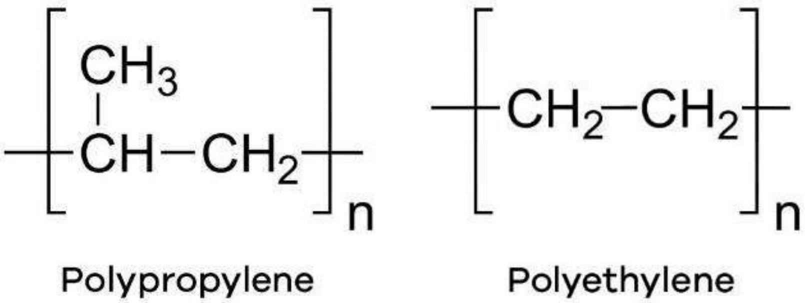
Another requirement for successful printing with PC is a printer enclosure, especially for medium and larger sized prints. Using an enclosure retains some of the heat generated by the heated build-plate. This will reduce the tendency for poor layer adhesion and for warping.

Professional and engineering grade materials tend to have a narrow range in which they show optimum printability with polycarbonate being one of these materials. It is thus generally recommended to find the optimum printing settings for your material printer combination. It is also recommended to avoid using active part cooling when printing with PC and to use a brim for medium and larger sized prints.

12. FDM printing of Polypropylene

What is PP?

PP is a very useful material and is one of the most commonly used thermoplastics in injection moulding due to its low cost and flexibility. PP has a relatively simple chemical structure and is very similar to polyethylene both in chemical structure and mechanical properties. PP can also be co-polymerised with PE to further optimise its mechanical properties.



The chemical structures of Polypropylene and Polyethylene are very similar

Some of the properties of PP are:

- High toughness
- Can be used in food safe applications, since it does not leach chemicals
- Resistant to attack by acids, water, detergents, oils and fats
- Soft and flexible
- Resistant to fatigue i.e. less prone to failure under repeated loads
- Some grades can show surprisingly good heat resistance

This means that PP finds its way in a variety of products including food packaging, food containers, utensils, athletic apparel and automotive parts such as car batteries. PP utensils can also be autoclaved and thus PP also finds use in laboratory and surgical equipment. Due to its fatigue resistance polypropylene is often used for the creation of living hinges.

A living hinge is a thin section of plastic that connects two plastic bodies together such as the one joining these two halves of a ketchup bottle cap

3D printing polypropylene can be challenging

Examples of FDM printed polypropylene parts - image via Ultimaker.com

While PP is cheap and widely used, PP is often overlooked as an FDM printing material. Due to its relatively simple non-polar structure PP shows very low adhesion to most surfaces. Furthermore, its simple structure also means that polymer molecules can easily pack near each other making it more prone to crystallisation. Thus, PP and PE materials can shrink considerably on solidification. As a result of both of these inherent properties, PP and PE parts can warp significantly on cooling. Furthermore, these materials have a glass transition temperature which is usually below room temperature. As a result, these materials will slowly solidify during printing and after printing leading to more pronounced levels of warp in prints.

Other drawbacks encountered when printing PP include:

Filament can be hard to print with some extruder set-ups: The inherent flexibility of the material means that most Bowden extruders and some direct drive extruders can have difficulties pushing this material to the nozzle. Furthermore, this material is prone to jamming when retraction is used. As a result, this material tends to ooze and string.

- Material needs to be printed slowly to avoid jams and ensure good filament flow
- Some filament types require very precise temperature control: Some filaments are very prone to warping and might require additional experimentation with cooling settings for optimal results.
- Support removal is challenging due to the excellent layer adhesion of this filament it is very hard to remove supports from printed parts. Furthermore, this material does not bridge very well.

Advantages of 3D printed PP parts

On the other hand, PP can have a number of advantages over other 3D printing materials such as:

- Low moisture absorption: the relative inertness of PP makes it practically impermeable to water
- Low density: PP has one of the lowest densities of any of the currently available FDM thermoplastics
- Excellent layer adhesion: Parts printed in PP tend to have very good layer adhesion and are rarely prone to layer splitting
- Excellent flexibility: PP can be used to make easily deform-able lightweight parts when printed with thin walls and light infill. On the other hand, tough parts with exceptional impact resistance can be produced when printing solid parts with thick walls.
- PP usually requires low printing temperatures: Most PP materials can be printed using a nozzle temperature of 200-220 ˚C and a build-plate temperature of 60-80 ˚C.
- Water tight parts can be easily 3D printed: Due to the very good layer adhesion of PP water tight parts can often be easily produced using a few slicer tweaks.

3D printed mini bottles made out of PP

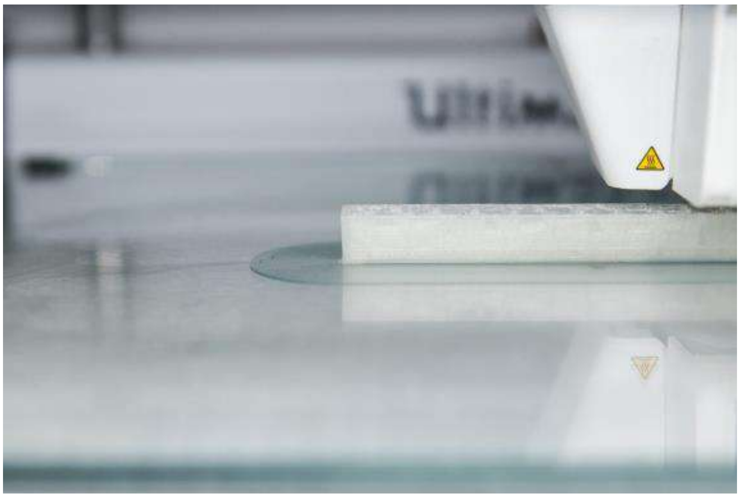
How to succeed when printing PP

Temperature control

With most filaments, temperature control is very important, with PP it is even more important due to the effects of crystallisation. Thus, apart from determining the optimum build-plate temperature and using an enclosed printer, careful tuning of print cooling might be required.

Crystalline or semi-crystalline materials tend to crystallise to a higher degree when they are cooled slowly. PP might benefit from quick cooling as opposed to slow cooling in regard to shrinkage. Of course, one also has to consider the temperature gradient as cooling too quickly can still lead to warping and also poor layer adhesion. In some cases, using 20-40% cooling can help reduce the amount of warp. As with most processes this will require some fine tuning and experimentation.

Tips for better bed adhesion



Using a brim helps prevent warping when printing with PP materials

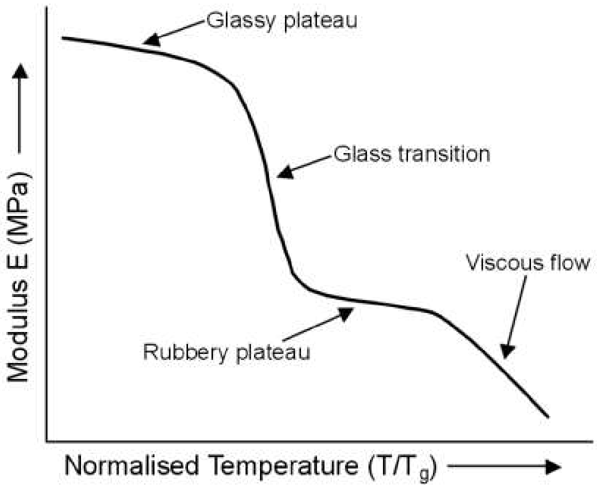
- Use a brim: For the majority of PP prints a 10-20mm brim is an absolute necessity for a successful print with little to no warp
- Start high then go low: All PP materials benefit from using an additional 10 ˚C for the first layer on both the build-plate and the nozzle. This helps the first layer adhere better but prevents warping and other problems related to excessive printing temperatures.
- Turn off the heated build-plate after the first layer: Some PP materials such as Owen's corning GF30-PP warp noticeably less when the heated build-plate is disable after the first layer. This helps anchor the material better to the build plate thus arresting warp!
- Print slowly on the first layer: Most materials but especially PP tend to show better first layer adhesion when the first layer is printed slowly (10-20 mm/s)

13. FDM printing of Flexible Materials

3D Printing flexible materials can be as challenging as it is exciting. However, choosing the right flexible material for your application and printing setup can be even less straightforward! Flexible materials come in a number of varieties. From soft materials which can be easily deformed before returning to their original shape, to firm materials which only flex slightly under load. The ease of deformation of material depends on the stiffness of the material and is often indicated by the hardness value.

What makes a material Flexible?

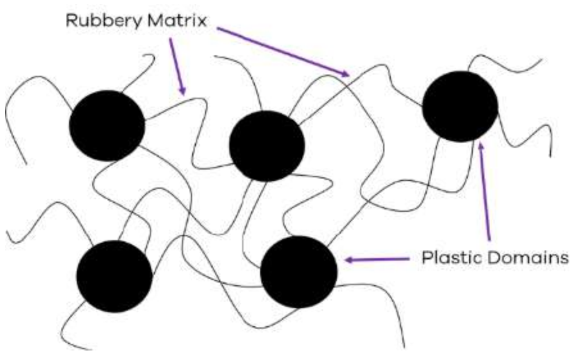
Thermoplastic materials get softer as the temperature increases since at higher temperatures polymer molecules can move around more freely. In fact, at particular temperatures (or over a temperature range) thermoplastic materials will get markedly softer and start behaving more like a soft rubbery material instead of a stiff hard plastic. This point is known as the glass to rubber transition temperature and its value depends on the material type. Some materials such as PLA have a low glass transition temperature (~60 °C) while other materials such as ULTEM1010 have a high glass transition temperature (~210°)!



Modulus-Temperature graph indicating Glass Transition Temperature

Unlike most other thermoplastics, TPEs are flexible and elastic at room temperatures and unlike natural thermoset rubbers, TPEs can still be re-moulded into shape by heating. This is due to the fact that TPEs are generally either copolymers (or a physical mixture of polymers) of plastic and rubber components. Thus, these materials can exhibit the advantages of both classes of materials due to the presence of soft and hard regions in the micro-structure of the material.

One of the first examples of commercially available TPEs was TPU (Thermoplastic Polyurethane) which became available in the 1950s. Following this styrene block copolymers (SBS) became available in the 1960s with more TPEs becoming available in the 1970s. By looking at the micro-structure of SBS one can easily understand how the structure gives rise to the unique properties of thermoplastic elastomers.



Schematic representation of the microstructure of SBS block co-polymer. The black circles represent the 'hard regions' made of polystyrene while the rubbery matrix is made of the elastomeric polybutadiene

Since SBS has hard sphere or rod like regions interconnected with soft rubbery regions, the rubbery regions can deform when stress is applied. This causes the chains to re-orient and "straighten" themselves out, going back to their original position when stress is relieved. The hard, plastic domains in fact act as physical cross-links helping the material regain its original shape. However once heated the plastic domains will soften and enable the material to be re-molded.

Flexible materials

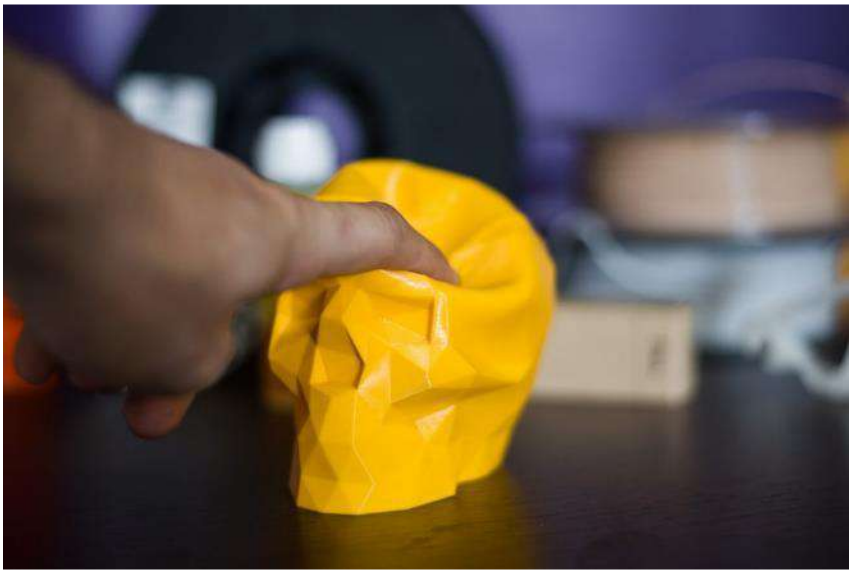
Nowadays there are myriad type of TPEs available for nearly every application. However, we shall focus on the materials available for FDM printing. These include:

- TPE: Thermoplastic Elastomer, this term is mostly used as a generic name for flexible filaments and can be used to describe most of the flexible materials available for FDM
- TPU: Thermoplastic Polyurethane describes a set of polymers which are usually on the rigid end of the spectrum. This makes them useful for parts which require some stiffness and are also easier to print.
- TPA: Thermoplastic Polyamides are TPE block copolymers of Nylon (Polyamide) and polyethers or polyesters. These materials are used in more demanding applications especially in lower temperatures. An example of this material is PCTPE ("Plasticized Copolyamide TPE") which is a produced by Taulmann.
- TPC: Thermoplastic Co-polymer, these filaments are generally derived from biological material and are easier to recycle. These materials can be made to be quite soft and thus might be harder to print. Furthermore, unlike TPUs these materials are more prone to warping during printing.
- Soft PLA: PLA can be chemically modified to behave more like flexible filament. Soft PLA tends to be slightly less rigid than the average TPU.

Choosing the right TPE

Choosing the right flexible material for your application will of course depend on the mechanical requirements. In fact one of the primary reasons TPEs are considered is their flexibility and elasticity. For this reason TPEs commonly have a clear indication of their Shore hardness, which indicates their softness.

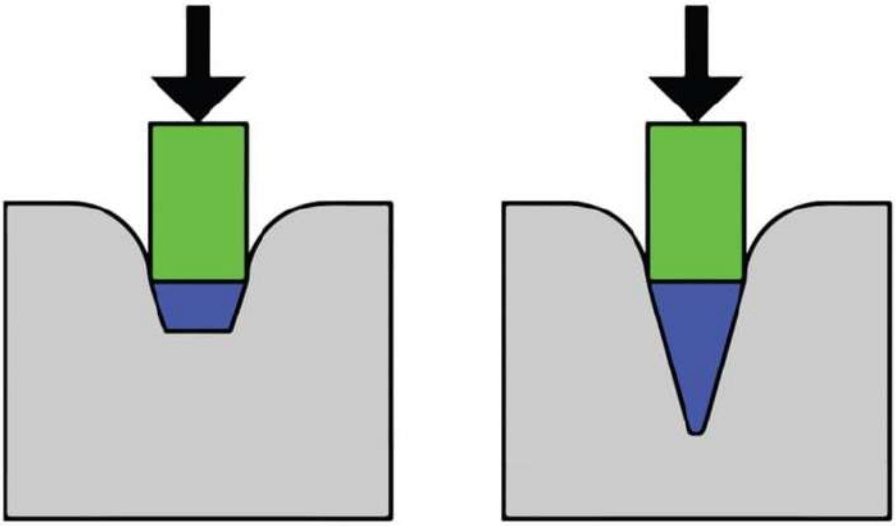
Hard materials tend to require a lot of force to deform while soft materials can be easily deformed with little force. Nonetheless in engineering terms hardness refers to the resistance of a material to localized plastic deformation either through indentation or abrasion. In the case of elastomers engineers usually care more about the materials resistance to deformation.



Soft materials deform easily on application of force such as this 3D print of a low poly skull. Model printed in Fillamentum Traffic Yellow TPU92A on a Raise3D N2 with a bondtech extruder upgrade which facilitates FDM printing of softer materials

Shore hardness?

The shore hardness refers to a scale defined by Albert Ferdinand Shore (1920s) to measure hardness using a durometer. A durometer is a device which measures the depth of indentation in a material as a result of a specific applied force. Several shore scales exist, relying on different indentation methods. However, the most frequently used scales are type A and type D, with the former scale being more suited for softer materials. Conversion between the two scales is possible albeit not recommended due to the weak correlation between the scales.



Shore A (left) and Shore D (right) Durometer tips. The shore D durometer has a pointed tip and is loaded with a higher force since it is intended for use with harder materials than those used during Shore A testing

Typically, FDM TPEs exhibit a shore A hardness between around 80A and 100A, this corresponds to a shore D hardness of between 40D to 60D (approximately). A lower number indicates a softer material. Typically, FDM materials having a Shore A hardness of around 90A or below are quite challenging to print due to the low stiffness of the filament. These materials are ideal for parts which need to be soft and elastic. On the other hands materials with a Shore hardness of 95A and above are easier to print since these materials are stiffer and more suited in scenarios where toughness and stiffness is required such as for printed tyres and bellows.

Nonetheless, when looking at a flexible material, the shore hardness does not necessarily tell the whole story. It is also recommended to consider the stiffness (Young's Modulus) and the ductility of the material according to the required application. The table below gives a recap of the properties of several TPE materials available for 3D printing.

Furthermore, it should also be considered that the geometry and density of the final printed part will have a marked influence on the deformability of a part.

Material	Type	Stiffness (MPa)	Elongation (%)	Shore Hardness	Other Properties
DSM Arnitel ID2045	TPC	29	350	34D	Good UV and chemical resistance, bio based
DSM Arnitel ID2060 HT	TPC	240	245	98A/61D	High temperature resistance, chemical resistance
DuPont Hytrel 3D4100FL	TPC-ET	130	250	60D	Chemical and thermal resistance
Filament PM TPE32	TPE	-	650	32D	
Polymaker PolyFlex TPU95	TPU	9.4	330	95A	
NinjaTek NinjaFlex	TPU	12	660	85A	Chemical resistance
NinjaTek Armadillo	TPU	396	18	75D	Chemical and abrasion resistance
Taulman PCTPE	TPA	6.6	500	>100A	Can be easily dyed
Fillamentum Flexfill TPE 90A	TPE	-	250	90A/30D	Chemical resistance, Certified for food and skin contact applications

The values presented in this table were obtained from the manufacturers' websites and data sheets. Since manufacturers can use different testing methods, some values cannot be directly compared and only serve to give an indication of the mechanical properties. Furthermore the mechanical properties of the printed part will depend greatly on the printing parameters and orientation of the printed part.

Applications of TPEs

- Vibration Dampening and Impact Resistance: Flexible materials tend to absorb energy quite well on impact. Furthermore, due to their lower stiffness and higher elasticity when compared to other materials, these materials have good vibration dampening properties. As a result, flexible materials are good candidates for use as shock absorbers, tyres, cases and couplings
- Grip and ergonomics: Some TPEs can be particularly soft and have a matte rubbery finish. This makes them ideal for use in ergonomic parts such as handles and touch points.
- Durability: Flexible materials can typically undergo large elastic deformations and tend to show high toughness and inter-layer adhesion. As a result, these materials are ideal for use in applications involving cyclic loading and a high toughness requirement such as bellows and flexible hosing.

3D printed Circular Flexbellow prototype. Printed in NinjaFlex NinjaTek on a Raise3D N2 with a bondtech extruder upgrade which facilitates FDM printing of softer materials.

- Chemical and Heat resistance: In some cases, TPEs used for bellows and flexible hoses will need to be used in environments in which the parts are exposed to elevated temperatures and petroleum-based products such as in automobiles. Specialised TPUs such as DSM Arnitel ID 2060HT are capable of withstanding such conditions making them ideal for the production of functional prototypes and parts.
- Seals: Due to the ability of TPEs (especially the softer ones) to easily deform, these materials are ideal for use as seals and gaskets.

Multimaterial 3D printed cap with printed in place o-ring, the cap is made from ABS material while the O-ring is printed in DSM Arnitel 2045.

- Rubber replacement: While the 3D printing of rubber materials is not possible on FDM machines, TPEs can be used as an alternative to produces rubber like parts and prototypes.
- Fun: Last but certainly not least, printing in TPE materials is ideal for the production of useful and not so useful flexible parts including phone cases, wallets, stress relievers and flexible toys.

A filament which likes to bend

TPUs and especially the softer TPEs are very flexible and easy to bend. As a result, these materials tend to be hard to push through the extrusion system as the filament deforms very easily. In fact, it is often recommended that a direct drive extruder is used for most TPEs with the exception of the stiffer materials which can be printer with a Bowden set-up as well. The softer materials might even require a specialised extruder for reliable printing of flexible and soft materials.

Due to its inherent softness and elasticity TPE filament often tends to jam and kink within the extrusion system especially when there are generous tolerances. Specialised extruders often have very tight tolerances and dual drive, in order to better push the filament to the nozzle.

Print it slow

Nonetheless in order to prevent jams and to ensure consistent extrusion it is recommended to print soft TPEs at slower speeds 20-40 mm/s and to keep the feed rate as consistent as possible. As a result, it is often recommended to minimize the number of retractions as much as possible and to keep the retraction speed and distance as low as possible. For inexperienced users, it is often recommended to disable retraction when printing with flexible filaments.

Lastly it is also a good idea to check for and eliminate any resistance on the filament spool since the filament can stretch and potentially lead to under-extrusion.

Ooze and stringing

Another drawback of printing TPE is that the resulting prints tend to have pronounced stringing due to the fact that the material is more prone to ooze from the nozzle. This is also a result of the filament's elasticity which often means that the material in the hot end can take more time to respond to changes in pressure.

Oozing and severe stringing in FDM printed parts

A careful application of retraction coupled with a capable extrusion system can help with eliminating ooze and stringing. Other setting such as coasting and avoiding outside travel movements will also help to reduce these artefacts. Nonetheless small amount of stringing can be easily removed by careful application of hot air from a heat gun.

Adhesion and part removal

With regards to adhesion TPEs generally tend to show two types of behaviours. TPUs typically adhere too well to most printing surfaces, this means that while adhesion is not an issue during printing, removing a print can be quite difficult. As a result, there is a good chance that the soft and flexible part can be damaged during print removal. On the other, hand other TPEs and TPCs do not stick too well to most build-plate surfaces which means that warping will be a significant issue. Magigoo Flex was designed with both of these problems in mind. As a first layer adhesive it provides a strong adhesive base for high performance TPEs such as DSM Arnitel, DuPont Hytrel and other TPCs and TPEs. On the other, hand it ensures that TPUs and other flexible materials* which tend to stick too well can be easily removed from build surface just by the application of water.

14. FDM printing of Composite Materials

The FDM 3D printing of composite material filaments opens up novel possibilities for 3D printed parts. A variety of different filler materials can drastically alter the properties of thermoplastic materials and the resulting printed parts. The appearance, mechanical properties, electrical properties or thermal properties of a material will thus also depend on the filler particles used and not just the base material.

Introduction - What are composite materials?

As their name suggests, composite materials are materials which have a composition of two or more materials. A carrier material or a matrix, which in the case of 3D printing filament is a thermoplastic material such as PLA, ABS or Nylon, and a filler material, which is insoluble in the matrix material. The filler material thus forms distinct and separate regions within the matrix material and will thus impart characteristics to the material which are different from that of the individual component. The properties of the resulting composite will depend on several factors, including the type of filler material present, its amount and also the shape and size of the filler particles and how well they interact with the matrix material. The matrix material will also influence the characteristics of the resulting material. In the case of FDM thermoplastic composites, the final material must retain thermoplastic behaviour to enable fabrication of the composite.

Why Composites?

As mentioned previously there are several reasons for 3D printing composite materials. In some cases, the choice might be due to aesthetic reasons however filler materials can also impart additional functionality and improve the mechanical properties of a thermoplastic material.

Appearance

The filler material in a composite filament will impart different visual characteristics to a filament. For example, wood particles will make the printed part look similar to a wooden part while copper filled filament will make the part look more like copper. Generally, unlike their neat counterparts, composite filaments tend to produce prints with a grainy texture. This imparts a very different finish to the printed part and helps hide the layer lines quite well. On the downside, filament filled purely for aesthetic reasons tend to show inferior durability than the unfilled material (usually PLA) since the filler material makes printed parts more brittle.

There are a variety of composite materials in this category including:

Wood Filament

Wood filament is probably one of the more popular 3D printing composite filament types. With this filament it is fairly easy to obtain good quality prints. Furthermore, the layer lines brought about by the FDM process will mimic the appearance of natural wood grains.

Bender from Futurama, printed in wood filament (ironically perhaps?), via: all3dp.com

Wood filaments can have different filler materials embedded in them including wood dust, bamboo or cork. Each type of filler material will give the printed part a different look, but the physical properties are more or less the same. Parts printed in wood filament can also be sanded easily and stained and varnished just like regular wood. This makes it very easy to produce parts which mimic wood very closely using 3D printing!

Einstein bust printed in wood filament and stained, via: pinshape.com

Metal Filament

In the FDM sphere Metal composite filament generally refers to PLA filaments impregnated with metal powders. Metal filament can be used to create printed parts with a similar look and feel to metal parts. Typically, metal filaments are filled with copper, brass, stainless steel or aluminium and usually have a higher amount of filler material, with values of around 80 % wt.

The high filler ratio means that the filament typically has a very high density and will also make the material more thermally conductive. As a result, printed parts will have a solid weighted feel when compared to parts printed in regular PLA. The increased thermal conductivity also means that lower printing temperatures are required to print these materials. The added heft however does mean that bridges and overhangs are harder to print than with regular PLA.

Once printed, the parts can be sanded and polished to a give a metallic shine to the finished part. Furthermore, some metal filaments can also be processed by 'oxidising' the parts to simulate the look of corroded or rusted metal.

ColorFabb copperFill Metal Filament before polishing and after polishing, via: matterhackers.com

Stone Filament

Another class of composite filament is stone filament which is composed of fine chalk particles in a PLA matrix. This gives the filament an appearance similar to that of stone. Furthermore, stone filled filaments typically come out printed with a gritty texture which dramatically reduces the appearance of layer lines. Similarly, to metal filaments stone filaments produce parts which are typically heavy and thus runs into similar printing issues as metal filaments.

Bust printed in stone filament, via: Airwolf3d.com

Filaments can also be filled with finely powdered marble particles. This gives the printed parts a speckled finish due to the presence of dark and light particles in the filament. As a result, the printed parts do not look like parts carved out of marble. Nonetheless the speckled appearance gives a pleasing finish which also hides the layer lines in prints very well.

Parts printed in Marble filament, via: 3dnatives.com

Glow in the dark Filament

As the name suggests glow in the dark filament produces parts which glow in the dark after exposure to light. This is a similar effect to glow in the dark toys which absorb light from the environment and emit it slowly when the environment is sufficiently dark.

via imaker3d.com

Glow in the dark filaments are typically made of ABS or PLA embedded with strontium aluminate, a phosphorescent compound. Typically, parts having thicker walls and a higher infill ratio will show a brighter glow due to the additional material in the print.

Functionality

Conductive Filament

While one might think that metal filament might be conductive, due to the insulating properties of the PLA matrix, current cannot pass through part printed in metal filament. Not sufficiently at least. In fact, conductive filament is filled with graphene which enables the movement of electrons in plastic parts.

Nonetheless conductive filament still does not conduct metal nearly as well as metals do. In fact, conductive filament is typically used to create custom electromagnetic and radiofrequency shields.

BASF Forward AM ESD safe filament

Another class of materials is labelled as ESD (Electrostatic Discharge) safe. These materials are conductive enough to prevent the build-up of electrostatic charges which might damage sensitive electric components. Conductive and ESD safe filaments are typically more expensive due to the high cost of graphene.

Magnetic Filament

Similar to metal filaments magnetic filaments are filled with metal particles. However, in this case the particles are ferromagnetic. Typically, iron powder is used which produces hefty parts which are attracted to magnets. In fact, magnetic filaments do not produce parts which are inherently magnetic but parts which can be attracted to a magnet. Since the filler material is iron, the printed parts can be sanded and exposed to saline water to produce a rusty finish.

Magnetic Iron PLA by Proto-Pasta, via filament2print.com

Metal Filament

Some types of metal filaments can also be functional in the sense that they can be used to produce fully metallic parts on a regular FDM printer. These filaments contain a high percentage of metal filler in a special polymer matrix that makes this process possible. An example of such filament is BASF Ultrafuse 316L which will be discussed further on in this manual. The printed part will then go through a bespoke process which first removes the polymer matrix to leave a solid part of metal particles which are then fused through a sintering process.

Parts printed in BASF Ultrafuse 316L after de-binding and sintering, via: crea3d.com

Other metal filaments which can be treated to form solid metal parts are available as well.

Mechanical Properties

A very interesting application of composite materials in FDM printing is the use of filler materials. Filler materials, generally in the form of fibres are added to enhance the mechanical properties of thermoplastic materials. Filled thermoplastics can exhibit exceptional improvement in the mechanical strength and stiffness of a material when compared to a similar unfilled material.

Typically, glass, carbon or Kevlar fibres are added to engineering thermoplastics such as Nylon, Polycarbonate, Polypropylene, ABS and PEEK giving rise to a multitude of possibilities when it comes to the production of functional parts produced by FDM printing. These fibres typically have a very high strength under tension, thus the incorporation of these materials within the polymer will reinforce the material under tensile load. When aligned to the direction to which force is applied, the polymer matrix will transfer the applied load to these fibres which will resist deformation. As a result, the shape, type, orientation, and relative amount of the reinforcing fibres will influence the reinforcing effects on the material.

The addition of filler materials also improves the dimensional and thermal stability of a polymer. As a result, there is a growing demand for use of these materials for the production of FDM parts to be used in the automotive and aerospace industry.

Parts printed in PA-CF low warp, via: colorfabb.com

The table below shows two materials from fiberthree. One is an unfilled Nylon (F3 PA Pure Pro) and the other is filled with 15% carbon fibres (F3 PA-CF Pro). It is immediately obvious that the filled material is much stiffer and stronger than the unfilled nylon, while the density of both materials is quite similar. The filled material also shows superior temperature resistance albeit with decreased ductility as suggested by the lower elongation values.

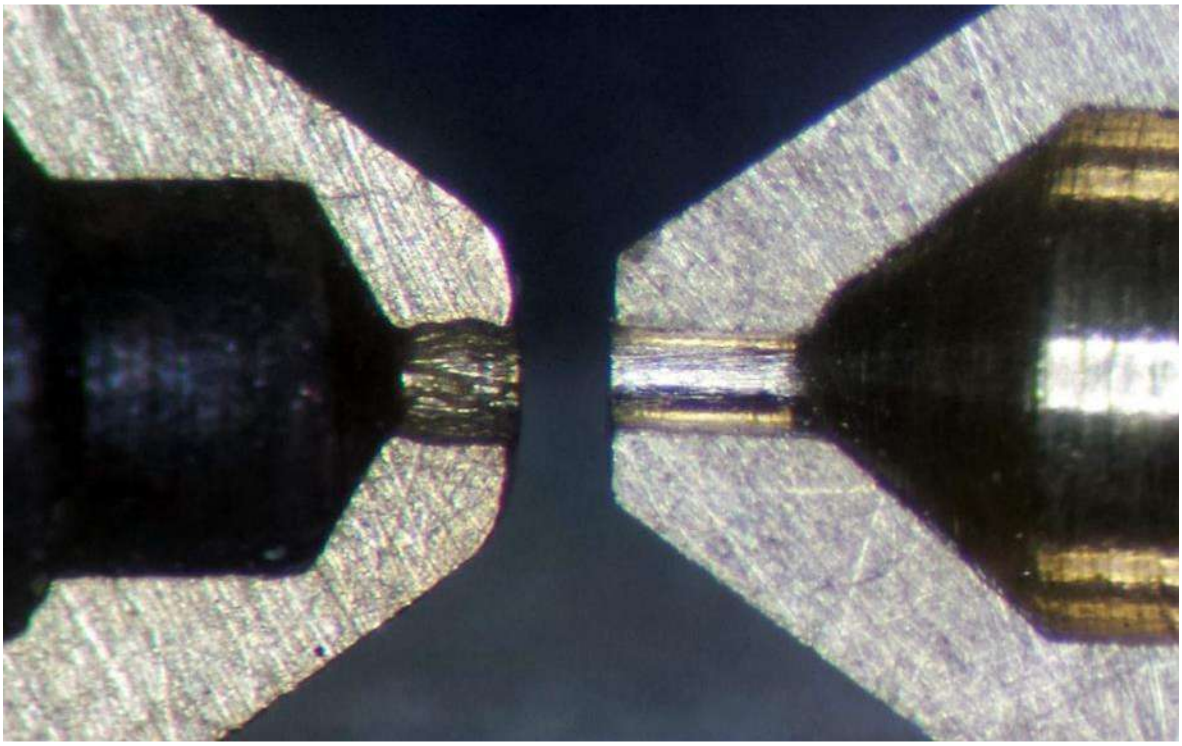
	F3 PA Pure Pro	F3 PA-CF Pro
Density (g/cm³)	1.2	1.25
Tensile Modulus (MPa)	3400	14400
Tensile strength (x-y) (0°) (MPa)	80	120
Elongation (0°) (%)	3.65	0.98
Heat Distortion Temp (°C)	90	120

Another way to improve the mechanical strength of printed parts is to embed continuous fibres during the printing process. The technology is used by a number of printer manufacturers including Markforged, Desktop Metal and Anisoprint. This results in parts with exceptional strength and stiffness rivalling those of metal along the direction of the laid carbon fibres.

3D printing Composite Materials

Composite materials usually do not require more expertise to print than the base material. In fact, in some cases, composite materials can be easier to print since the addition of filler material increases dimensional stability and reduces warping. The addition of hard filler material particles however does also mean that there will be more wear on printer components moving and extruding the filament. This includes the extruder gears which should preferably be swapped out for hardened materials if you are printing a lot of composite filaments.

More importantly the nozzle will also experience much higher levels of wear when printing composite materials. It is recommended to switch to a hardened steel nozzle when printing any type of composite filament, since brass nozzles will wear out very quickly. Additionally, one can also switch to a ruby nozzle which should last much longer than hardened steel nozzles.



Nozzle wear caused by the use of carbon fiber filament, via: 3Dprinting-blog.com

Another consideration when printing with composite materials is the nozzle diameter. Since composite materials are filed with particles these tend to clog smaller diameter sized nozzles which will lead to under-extrusion and also failed prints. In most cases it is recommended to use a nozzle diameter of at least 0.5 mm to avoid nozzle clogs with composite filament.

Build-plate adhesion

As previously stated in most cases composite filaments tend to warp less due to the extra dimensional stability brought about filler particles. As a result, for most materials, the build-plate adhesive which is usually used with the neat material will also work well for the filled material. Nonetheless in some cases adhesion might still be challenging with some types of composite materials such as GF-30PP and BASF Ultrafuse 316L.

GF-30PP

Polypropylene (PP) is a thermoplastic that is widely used in 3D printing in various industries, most especially in sectors such as the automotive industry. This being said, parts printed out of PP lack stiffness, and reinforcement. This is where the demand for a strong, reinforced material begins. Glass Fiber Reinforced Polypropylene (GF30 PP) is a material that contains specially inserted glass fibers that offer additional properties to the base PP material. GF30 PP can be found in a filament format, specifically for 3D printing.

GF30 PP is mostly used in the automotive industry, for parts that are subjected to higher temperatures such as engine bay components. This is due to the high heat resistance that GF30 PP offers, as well as the low moisture absorption. PPGF also has a low density, which makes creating lightweight parts easy, despite larger sizes. These properties allow for this material to also be used to print operational prototypes and also jigs and fixtures.

For FDM printing GF30 PP is available as BASF forward AM and was previously available as Owen Corning's X-strand GF30-PP, which happens to be the filament we are currently using at Magigoo. These filaments offer excellent chemical resistance, as well as improved UV resistance. Using this filament allows the printing of stiffer parts, with elevated heat resistance.

	PLA	PET-G	ABS	PP	GF30-PP
Strength (XY - MPa)	46.6	31.9	33.3	15.5	41.7
Strength (Z - MPa)	43.5	13.4	25.4	9.0	15.9
Stiffness (XY - MPa)	2636	1472	2174	541	2628
Stiffness (Z - MPa)	/	1087	1835	435	2242
Ductility (XY - %)	1.90	6.8	2.7	118.6	4.4
Ductility (Z - %)	/	1.3	2.4	5.4	0.8
Impact Strength (XY - kj/m2)	2.68	5.1	12.6	37.7	20.5
Impact Strength (Z - kj/m2)	/	/	10.5	11.6	2.6
HDT at 0.45 MPa	60	78	100	54	127

Table 5: ***Typical mechanical properties of common 3D printing materials*** **compared with Owen's corning GF30 PP - The values provided are supplied by the manufacturer and only serve as an indication of the mechanical properties of printed parts**

The table above compares the mechanical and thermal properties of GF30 PP with those of common 3D printing materials and that of an unfilled PP filament. It can be clearly observed that the addition of glass fibers greatly increases the strength and stiffness of the PP material at the expense of ductility and toughness (impact strength). The addition of glass fibers also more than doubles the heat deflection temperature of the material. This gives the material a much higher heat deflection temperature than typical 3D printing materials.

Printing with GF30 PP

Printing with GF30 PP poses a couple of challenges especially for the uninitiated. The addition of hard filler material particles means that there will be more wear on printer components moving and extruding the filament. This includes the extruder gears which should preferably be swapped out for hardened materials if you are printing a lot of composite filaments.

More importantly the nozzle will also experience much higher levels of wear when printing composite materials such as GF30-PP. It is recommended to switch to a hardened steel nozzle when printing any type of composite filament, since brass nozzles will wear out very quickly. Additionally, one can also switch to a ruby nozzle which should last much longer than hardened steel nozzles.

Another consideration when printing with composite materials is the nozzle diameter. Since composite materials are filed with particles these tend to clog smaller diameter sized nozzles which will lead to under-extrusion and also failed prints. In most cases it is recommended to use a nozzle diameter of at least 0.5 mm to avoid nozzle clogs with composite filament.

Another challenging aspect concerning FDM printing with GF30 PP is its high tendency to warp. Even though filled materials tend to warp less than their unfilled counterparts when printed, GF30 PP is an exception. Due to its higher tendency to warp **Magigoo PPGF** was formulated in collaboration with Owens corning as a bespoke first layer adhesive for this material.

In order to minimize warping with this filament type it is also important to dial in the printing settings. For printers without an actively heated chamber such as the Ultimaker S5 it is recommended to first print with the heated bed set to around 80-100 °C for the first layer and then cooling the build-plate to around 50-70 °C. This helps the adhesive to hold on to the printed part more strongly. Furthermore, it is recommended to use a slow first layer printing speed (< 20 mm/s) and also a wide brim to help mitigate warp further, especially for larger sized parts.

In the case of larger sized parts, part removal might require more effort, in this case it is recommended to heat up the build-plate to around 80 °C to help with part removal.

BASF Ultrafuse 316L – A very special FDM material

BASF Ultrafuse 316L enables the production of solid 316L-grade stainless steel parts through the use of desktop FDM 3D printers and a sintering process. Ultrafuse® 316L filament consists of highly refined metal particles embedded in thermoplastic binders. Thus, as a filament, the metal-polymer composite comes with none of the occupational and safety hazards associated with the handling of fine metallic powders. This enables affordable, straightforward, and safe 3D printing of stainless-steel parts.

Ultrafuse 316L by BASF Forward AM is a metal polymer composite filament designed for use with FDM (Fused deposition modelling) printers. The filament contains a high percentage (90%) of metal powder encased in a polymer binder enabling it to be formed in filament form. This avoids the potential hazards of handling fine metal powders which are present when using other AM techniques. The printed green part can then be processed using a catalytic de-binding and sintering process developed by BASF for the processing of powder injection moulded parts.

BASF Forward AM Ultrafuse 316L Filament

Through the de-binding and sintering process a full metal stainless steel parts which can be used in multiple applications including:

- Functional Prototyping
- Medical Equipment
- Automotive Parts
- Pipes, pumps and valves for chemicals, gas, and oils
- Parts for tooling and mold inlays with near-surface cooling
- Non-magnetizable parts with high corrosion resistance and toughness

Owing to its high flexibility and surface grip BASF Ultrafuse 316L can be printed on a multitude of different 3D printers either with a Bowden or direct drive setting. Nonetheless due to the presence of metal powder within the filament it is recommended to use a hardened nozzle to limit wear.

Printing with BASF Ultrafuse 316L

The following settings are recommended by BASF Forward AM for printing with Ultrafuse 316L filament.

Printing Parameter	Value
Nozzle Temperature	230-350 °C
Layer Height	0.15 mm
Infill	Minimum 60%, recommended 100%
Build-plate temperature	90-120 °C
Build Plate Adhesion	Magigoo® Pro Metal
Cooling fan	Off
Print speed	35 mm/s
XY Scaling Factor	120%
Z Scaling Factor	126%

Parts printed in BASF Forward AM Ultrafuse 316L

While BASF 316L allows easy production of metal parts on a wide range of desktop and professional FDM machines some precautions and considerations must be considered when printing with this material.

Since the material is made of a binder and metal powder, once the binder is removed and the part is sintered, the part will shrink considerably but consistently at 16% in the X and Y directions and 20% in the Z direction. This means that prior to printing the parts should be scaled to 120% in the X and Y directions and 126% in the Z-direction. This ensures that after processing the part has the correct dimensions.

Additionally, precautions must be taken to minimise the warpage of the material. It is recommended that an enclosed printer with a low chamber air flow is used when printing this material. This prevents sudden changes in temperature which often increases the propensity of a part to warp. Furthermore, it is suggested to optimise the build-plate temperature for the material to ensure that the optimal temperature is being used.

What is de-binding and sintering?

The de-binding and sintering steps can be done in a specialised facility using methods developed by BASF. Once the printed part is ready it is known as the green part and is now ready for catalytic de-binding. Catalytic de-binding is a thermochemical process in which the green part is heated and exposed to nitric acid fumes in a nitrogen environment. This removes the polymer-based binder to produce a brown part which is ready to be solidified into a fully metallic part by sintering.

Sintering furnace, source: DirectIndustry

The brown parts produced by debinding are quite porous and fragile, and thus require sintering to produce a solid densified part, known as the white part. Sintering is done in a pure hydrogen atmosphere at elevated temperatures but below the melting point of the material. The high heat will cause the aggregated metal powder to coalesce into a single solid metallic part whilst the hydrogen atmosphere protects the metal from oxidation.

Best practices for printing BASF Ultrafuse 316L with Magigoo Metal

While 3D printing with BASF Ultrafuse 316L might require some design and slicing considerations, Magigoo Metal makes the 3D printing process as straightforward as possible. A build-plate temperature of around 90-110 °C is required depending on the 3D printer and printing environment. This ensures that the 3D printed parts do not warp during printing!

On the other hand, due to the sensitive nature of the green part, part removal requires special attention. As mentioned previously the green part is easy to deform, thus it is not recommended to use a part removal tool or excessive force to remove the printed part. Instead it is recommended to submerge the printed part and the build-plate in water and wait for 30 minutes to 2 hours depending on the size of the part. If submerging the build-plate is not possible, one can also wet around the perimeter of the part with water.

Get the first layer right

It is also recommended to fine tune the first layer height to ensure easy part removal. In most cases ideal build-plate levelling will results in parts that are easy to detach. But in some cases, especially if the nozzle is too close to the build-plate, part removal might require more time since the water cannot penetrate in between the part and the build-plate. In these cases it is suggested to either increase z-offset or decrease first layer flow.

15. Moisture and 3D Printing

What is moisture?

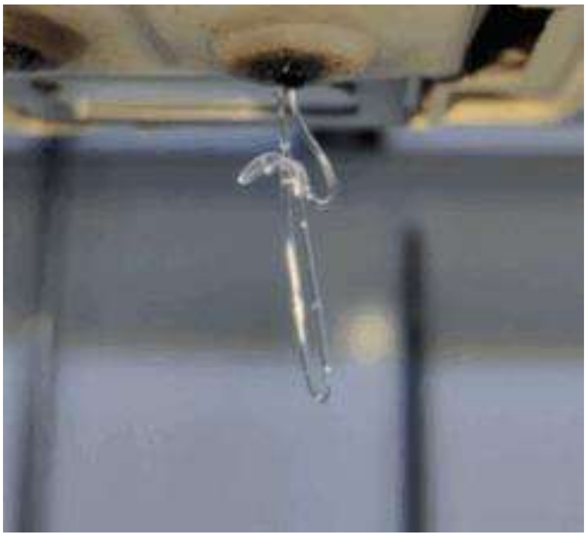
The word moisture describes the presence of slight-to-moderate amount of liquid, most commonly water, as a vapour, within a solid or condensed on a surface. When it comes to defining the moisture in the air, we talk about humidity. The moisture or humidity of an environment describes the amount of water vapour in the environment's air. In FDM printing, we usually speak about the moisture present the filament material. The moisture present in the filament plays a big role in whether a part is printed successfully or not.

Humidity may sometimes present itself as condensation on a cold window

Moisture in 3D Printing

There are various polymers that are used for 3D printing, and they all react to moisture differently. Some are very hygroscopic, and will readily absorb moisture from the air such as nylons, PET-G, polycarbonates and PEI. Some are less hygroscopic and will not absorb as much moisture, some examples include PP and PLA.

The easiest way to know if filament is wet, is to load it into the printer, and observe the extrusion. If the filament is wet, you will be able to see bubbles forming in the extruded material. Alongside this, one should be able to hear a hissing sound. If the filament has a lot of moisture, one might be able to see steam exit the nozzle.

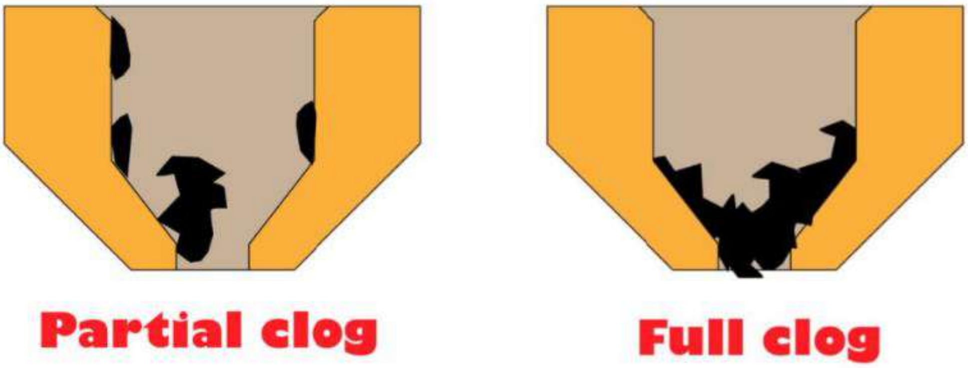


Most of the time, even a brand new spool of filament right out of the box, might have enough moisture in it to affect print quality. Furthermore, from our testing and past experience, we have found that hygroscopic filaments especially nylons absorb moisture quickly from a humid environment. In fact in a few days some filaments absorb enough moisture to result in severe printing defects in FDM printed parts.

Issues with Moisture and 3D Printing

The bubbles seen when extruding a filament with moisture present is one of the main problems that moisture creates in 3D printing. These bubbles occur due to the water (moisture) trapped within the plastic. When the nozzle heats up to temperatures over 100 °C, the water vaporizes and expands, creating a number of bubbles in the filament. The bubbles create voids in the heated filament strand, which causes a variety of problems when it comes to Printing.

One of the most inconvenient issues that such moisture in filaments creates is the blocking of nozzles. If, the filament has swollen slightly, it may cause a blockage in the printer's nozzle. A nozzle may be partially clogged or fully clogged.



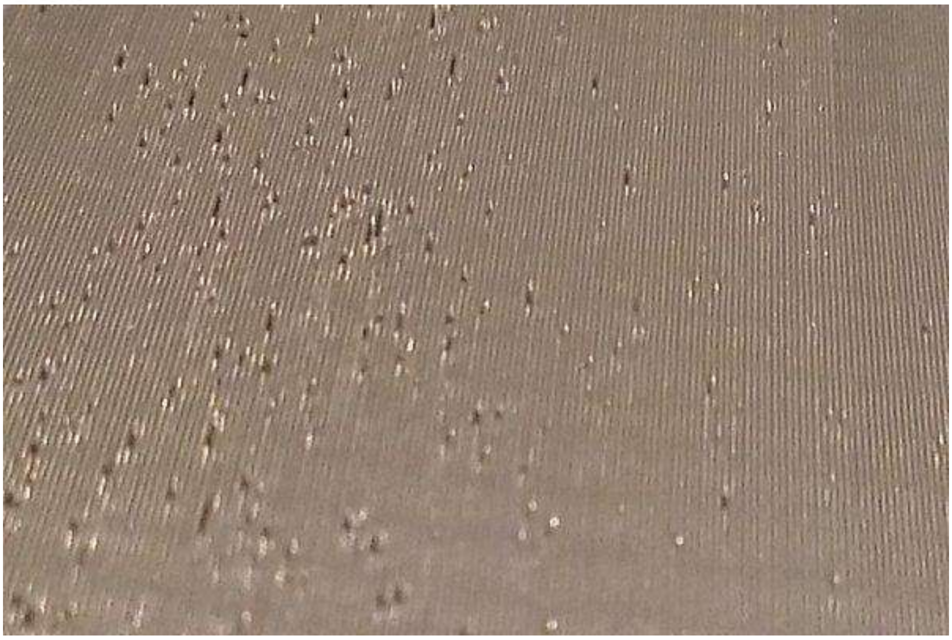
A Partial Clog and a Full Clog. Source : [all3dp.com](all3dp.com)

A partially clogged nozzle will still allow some material to flow through, but inconsistently, and will result in an under-extrusion. A fully clogged nozzle will not allow any material to pass through and will more than likely cause worse issues to the printer. If the filament cannot pass through the nozzle, it will get stuck in the hot end, and eventually burn if the printing process is not stopped. This might also cause some damage to the extruder mechanism if it tries to continue extruding the material. Depending on the severity of the blockage, this may be fixed by simply cleaning the nozzle internally, or in extreme cases, by dismantling the nozzle and cleaning it thoroughly.

Printing with wet filament may also present visual differences in the extruded filament and printed parts. The filament will be more prone to stringing, and may have a slightly bleached colour. The final print often also has a softer texture than a print with dry filament.

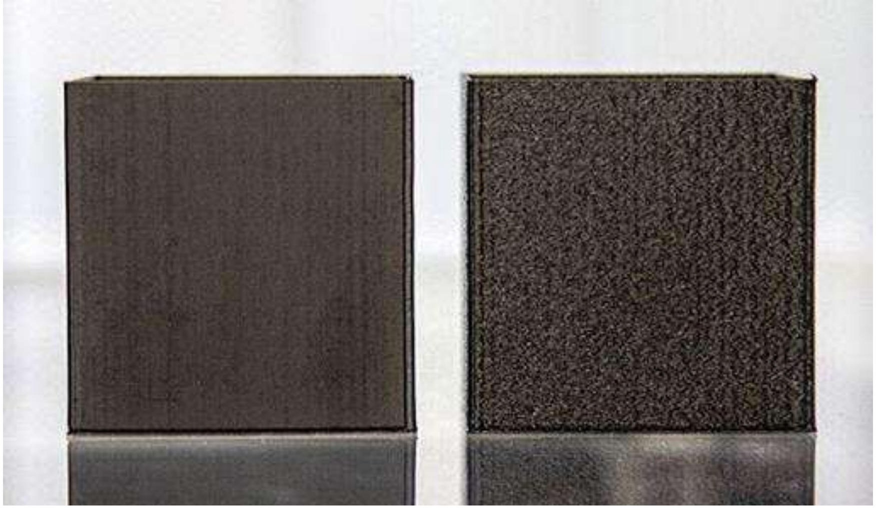
Properties of Parts 3D Printed with Wet Filament

Moisture absorption has a major effect on the mechanical properties of 3D printed parts that use these hygroscopic filaments. A common property change is that the Glass Transition Temperature (Tg) of a polymer decreases. The glass transition temperature of a material is the temperature at which an amorphous system changes from a glassy to a rubbery state. For most polymers that have absorbed moisture, this temperature will lower significantly. This makes the glass transition temperature easier to reach. The degree to which the moisture absorption affects the Tg varies from one polymer to another. As previously mentioned, printed parts also lose a lot of strength and structural integrity due to the air bubbles in them.



Voids in 3D Printed parts. Source: idig3dpritning.com

These bubbles will result in voids in the print. This becomes an issue when the print needs to support a load, since voids in the structure will reduce its integrity and strength. Parts with voids become brittle, and easy to break, especially under great loads. This is because the voids affect the interlayer bonds of the print, making them weaker.



A block printed using Dry filament (left) and Wet filament (right) Source: all3dp.com

How to Counter Moisture Absorption in filaments

There are two main ways to counter moisture absorption in filaments; a Preventive method, and a Recovery method. The preventive method to counter moisture is to keep the filament stored in a dry space such a sealed bag with desiccant or a specialised filament storage solution. The recovery method is to actively dry a filament that has absorbed moisture, and recover its ability to be printed.

What is drying?

Drying is the process of removing water molecules from the polymer chains. There are a number of ways that water can be removed from filaments, and these will be explained in the next section. Drying can be quite simple and cheap to do, if you have the right environment and parameters. Most drying techniques involve heating the filament to a temperature below the Tg. The heat excites the water molecules that have bound to the polymer and allows them to move freely and diffuse to the surface. It is important that the Tg is not exceeded, as this could irreparably damage the filament. The temperatures that each material is best dried at can be seen in the table below.

Material	Temperature (°C)	Time (h)
PLA	40-50	4
ABS	50-70	4
Nylon	60-80	8
PC	60-80	8

Methods of Drying Filaments

There are various methods and tools that one can use to dry their filaments, and they vary in effectiveness, reliability and cost. Some of the most used drying techniques and mention their pros and cons in regards to cost, reliability and how well they dry filament can be found below.

Using a Printer's Build-Plate

One way to dry filaments easily is by using a heated build plate of an enclosed 3D printer. By letting the spooled filament rest on the hot build plate for as long as the material requires, this is a cheap way of drying a moisture imbued filament. If the 3D printer allows for a heated chamber, this should be switched on and set to the same temperature that the build plate would be. While this does dry the filament in some cases, this method has a couple of disadvantages. Firstly, when using this technique you cannot use your printer it for printing. Furthermore since most printers can only heat the build-plate, the filament will probably not be heated and dried evenly.

A filament being dried inside a heated 3D printer chamber.

Using an Oven

An oven is a simple, easy way to dry your filaments. Using an oven requires you to preheat it to the specific temperature that you want to dry your filament in. It is important that you monitor the oven with a temperature probe inside the oven. This is because ovens have a tendency to overshoot the temperature the first time that it is heating up. The fan assist feature on ovens aids the drying and improves heat distribution. Once the oven has heated to the right temperature, the filament spool can be placed in the oven. When the time is up and you are ready to remove the filament from the oven, the best practice is to first switch it off and leave the filament to cool down slowly in the oven.

One disadvantage that ovens present when drying filaments is that their internal temperature is typically inconsistent and can overshoot. When the temperature gets too high, it might cause the filament spool to deform.

Warped filament spool due to excessive heating in an oven

Food Dehydrators

Food Dehydrators are tools used for drying fruits and food. They can be used within limits to dry certain filaments. The premise is similar to that of using an oven to dry filaments. While they are slightly better suited for drying some filament since they work at lower temperatures than typical ovens, they might not be capable of getting hot enough to dry certain polymers such as Nylons. This paired with the fact that they are not specifically designed for drying filaments, means that they might not actually be very effective filament dryers. Most food dehydrators will also need modifications be able to accommodate filaments inside.

A food dehydrator turned into a filament dryer, Source : bluepapertech.com

Filament Dryers

Filament dryers are machines that are meant for filament drying. Most of the times, they are specifically designed for that purpose. Sometimes, they are adapted from other machines, such as the food dehydrators mentioned earlier. They come in all shapes and sizes but typically all involve some dial or touch screen, to set the temperature and a space for the filament spool to fit in. There are various types of dryers which use different methods to dry the filament. Some dryers work alongside the printer, allowing the filament to enter and be used in the printer as it is drying. Filament dryers can offer a more reliable and tailored drying experience.

3D Printing Filament Dryer, source: esun3d.net

So what's the best solution?

As can be seen above, there are various machines and techniques to get rid of moisture and in turn, dry filaments. While all these machines can be used for drying, they each have their limitations and drawbacks. Firstly the dryers mentioned above do not only dry the amount of filament that you need for any given print, but also the rest of the spool. The multiple heat and cool cycles that the filament will go through throughout the lifecycle of the spool is detrimental to the material. Another issue with these machines is the fact that you need to have these running and drying for at least 4-8 hours before you load the filament into the printer.

The Drywise Dryer

Drywise is a stand-alone machine to be used in tandem with a 3D printer to dry hygroscopic filaments in real time while printing. After a pre-dry cycle on the first portion of filament, a user is free to print the freshly dried filament with ease, and consistency.

Why dry filament in-line?

Some materials, due to high moisture levels already present in the material, would need to be dried even before the first use. The presence of moisture in filaments can cause visible artefacts during printing, including poor surface finish, stringing and oozing. Without special storage or pre-treatment the print quality can still degrade noticeably during the print, especially during the course of a longer print

To combat this, most 3D printer users are resorting to drying filament in ovens and using material stations that reduce the uptake of additional moisture. In other cases, either bulky or costly commercial products are used or for money-saving, make-shift boxes with desiccant are crafted. This all is cumbersome and can take tens of hours of drying time. Many users have to follow filament manufacturer's instructions for drying the filament and having a multitude of unstandardized ways of treatment leave a lot of room for error and lead to unreliable results. Moreover, pre-treating filament spools multiple times may degrade the quality of the material.

Drywise takes out the guesswork when 3D printing hygroscopic materials. Drywise needs just 40-90 minutes to pre-treat a short portion of filament and bring it to the best printing conditions before 3D printing can start. An in-line drying design assures that long 3D prints will always be printed with consistent material dried to best printing conditions.

Drywise near Ultimaker S5

Key features of Drywise in-line dryer

- **Fast**: An in-line design dries only the section of filament that is 3D printed, reducing the amount of work needed.
- **Intelligent**: Smart sensors help to dry filament to the best 3D printing condition, avoiding overheating and over drying.
- **User friendly**: A user friendly UI guides the user all the way. Exchangeable and rechargeable desiccant cartridges will eliminate downtime.
- **Safe**: The smart sensor systems detect idle moments and switch off heaters for the safety
- **Upgradable**: The list of certified filaments with calibrated print settings is constantly growing.

Material: Novamid ID 1030 (2.85) at 2.8% w/w humidity. Filament was hydrated using an enclosed chamber circulating air at a constant RH of 75% at 23°C. Printed on Ultimaker S3. Print duration, ca. 10 hours

Test showing Drywise on and off

About Thought3D

Thought3D is an R&D company based in Malta. Thought3D combines knowledge of 3D printing, materials and chemistry and provides products and services for reliable additive manufacturing. It is a leader in creating solutions for material pre-and post-processing. In 2014 the company experienced the first layer adhesion problem first hand and came up with a novel smart adhesive that sticks when hot and releases when cold. In 2021 Thought3D released its new filament drying solution under brand Drywise. Today, Thought3D produces and sells Magigoo, Magigoo PRO adhesives, Magigoo Coater and Drywise in-line dyer worldwide through a network of resellers and industry partners. Thought3D aims to provide a superior user experience in 3D printing.