Magigoo Manual Of Life

- Don't Panic -

Contents

1 Adhesive Range

- Magigoo Original
- Magigoo PP
- Magigoo PC
- Magigoo PA
- Magigoo PPGF
- Magigoo Flex
- Magigoo HT
- Magigoo Metal

2 Optimising settings for improved build-plate adhesion in FDM printing

2.1 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**

2.2 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.

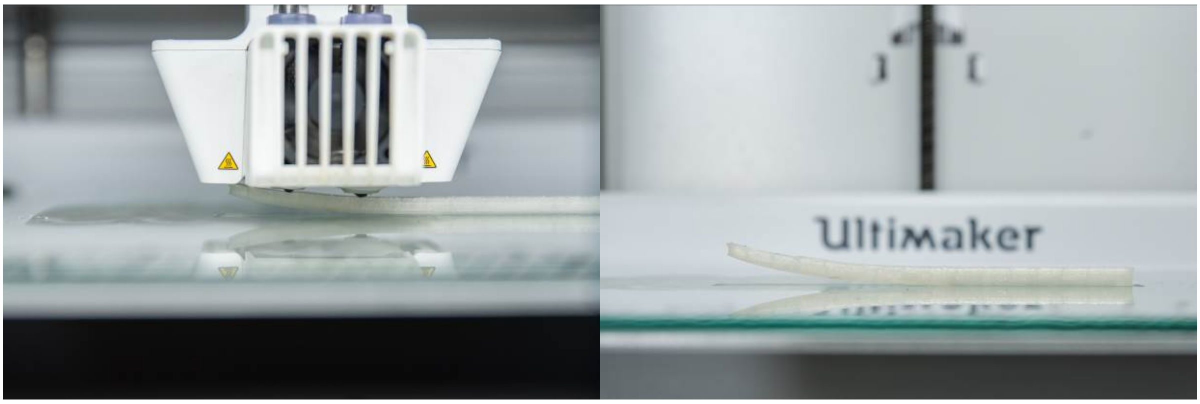


Figure 2: Nylon warping due to insufficient bed adhesion, one side is printed on regular glue stick, the other on Magigoo PA adhesive

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

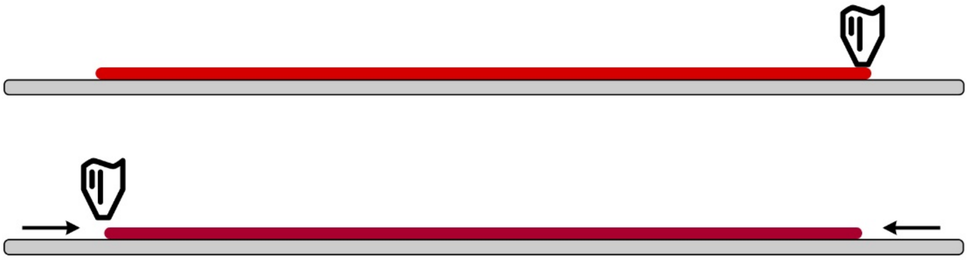


Figure 3: First layer

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).

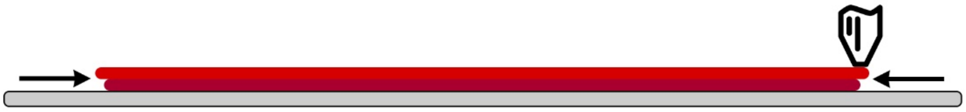


Figure 4: Second layer

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).

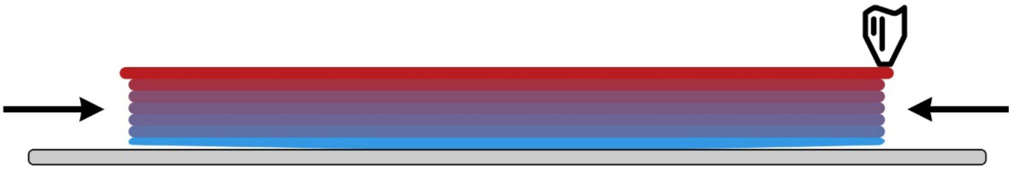


Figure 5: Print warp due to thermal gradient

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).

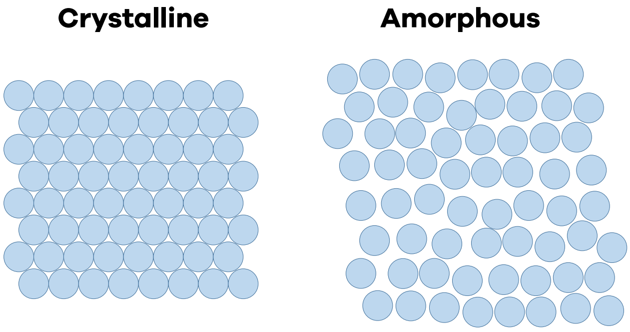


Figure 6: A visual representation of the building block arrangement in crystalline and amorphous solids

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.

2.3 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 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 virgin 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).

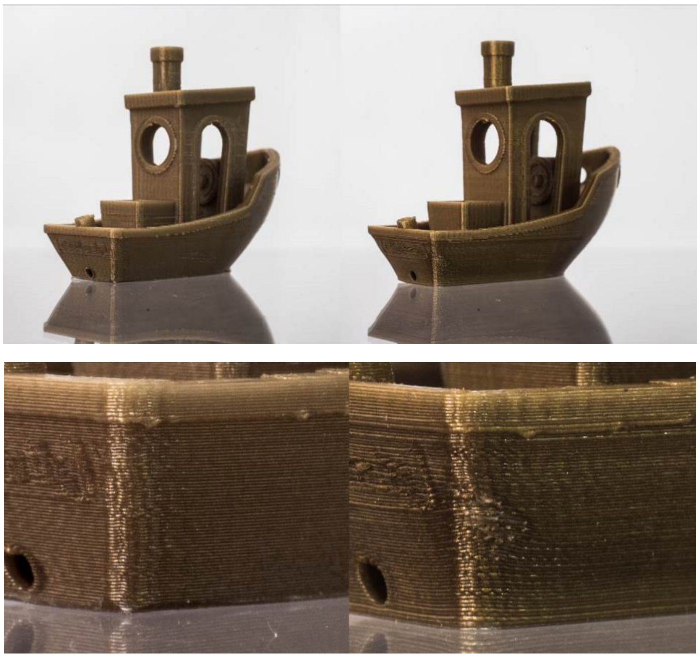


Figure 7: Benchy print using PLA, images on the left shows a benchy printed with the heated bed set to 60 °C, while the images on the right show a benchy printed at 80 °C, on closer it can be noted that the benchy on the right warped and also shows artefacts due to the layers curling up from excessive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

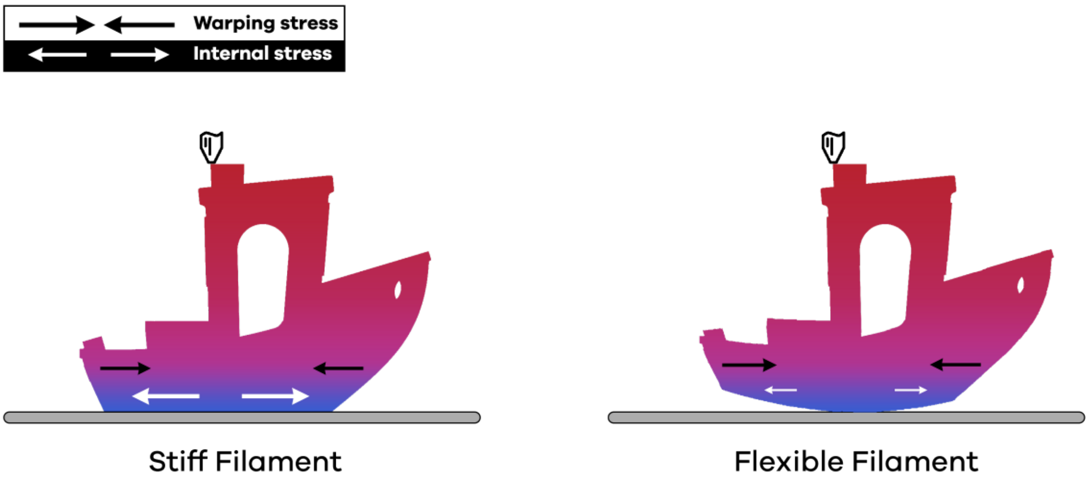


Figure 8: Printing with a stiff material versus a flexible material, the stiff material will be able to resist the warping stress, since the cooler material at the bottom of the print has enough strength (internal stress) to resist further warping. A flexible or soft material will not be strong enough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

2.4 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 a different temperatures. This print is preferably small in order to minimise use of filament and also to save time, however 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

2.4.1 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).

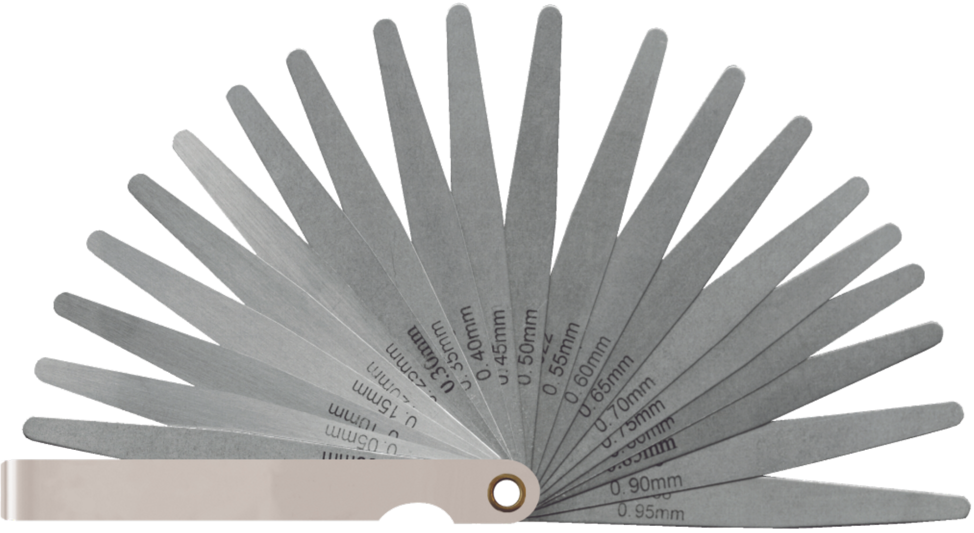


Figure 12: Feeler gauge, ideal for measuring small amounts of warp accurately

2.4.2 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).

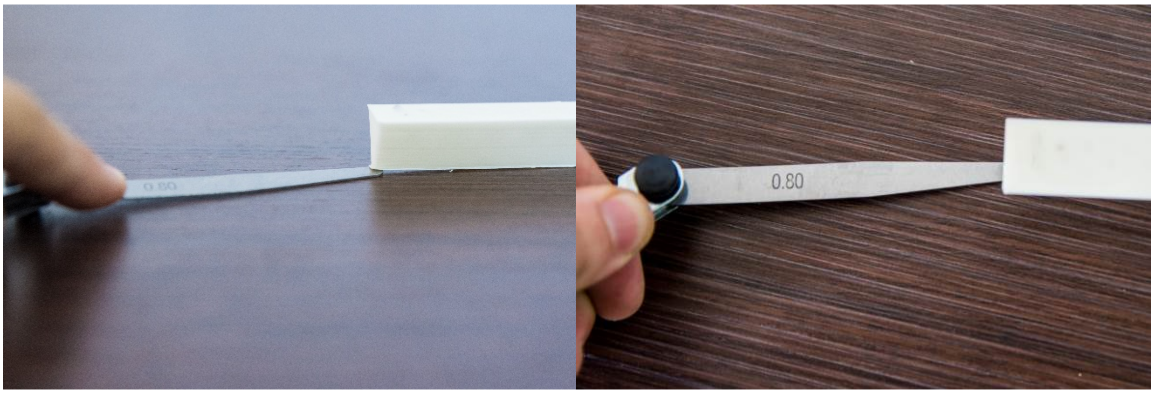


Figure 13: Measuring the warp on the edge of the wedge print using a feeler gauge

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.

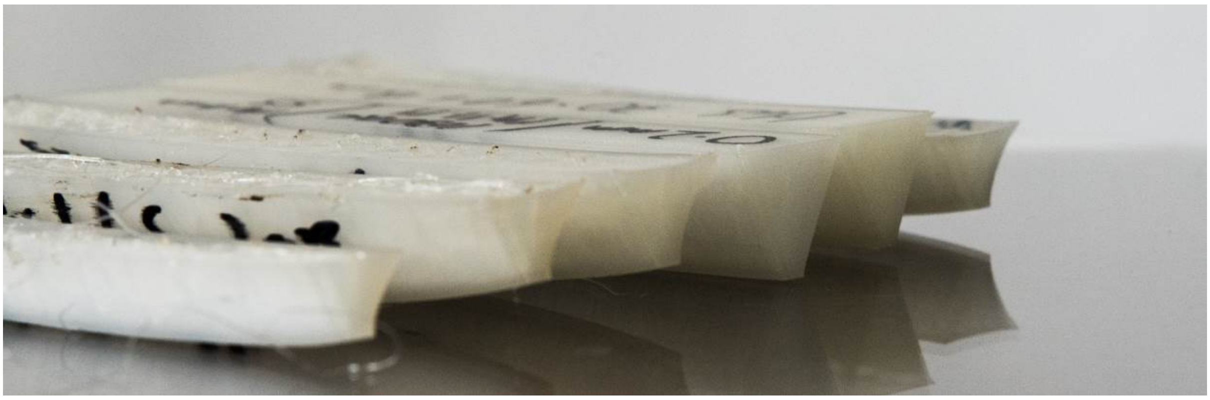


Figure 14: Wedge prints at different temperatures showing different amounts of warp and/or rates of completion

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:

2.5 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:

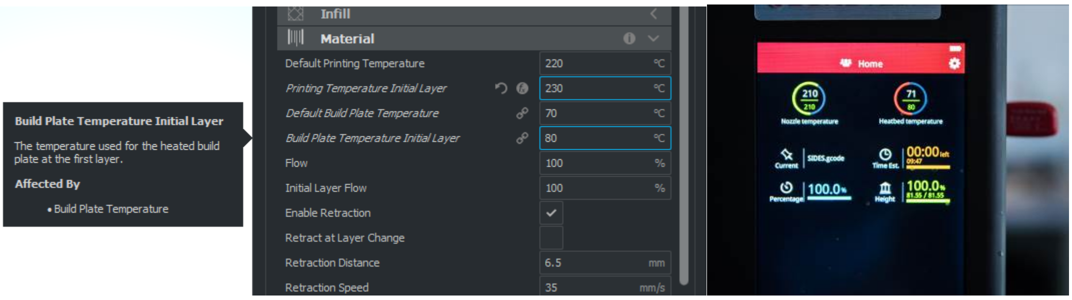
2.5.1 Application

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, for a more detailed application guide visit

2.5.2 Using a 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.



Build Plate Temperature Initial Layer

The temperature used for the heated build plate at the first layer.

Affected By

- Build Plate Temperature

Infill

Material

Default Printing Temperature	220	°C
Printing Temperature Initial Layer	230	°C
Default Build Plate Temperature	70	°C
Build Plate Temperature Initial Layer	80	°C
Flow	100	%
Initial Layer Flow	100	%
Enable Retraction	☑	
Retract at Layer Change	☐	
Retraction Distance	6.5	mm
Retraction Speed	35	mm/s

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

2.5.3 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.

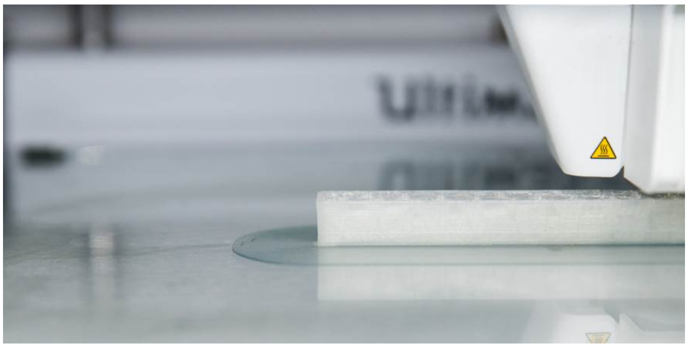


Figure 18: Using a brim helps mitigate warp with materials that are prone to warping

2.5.4 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).

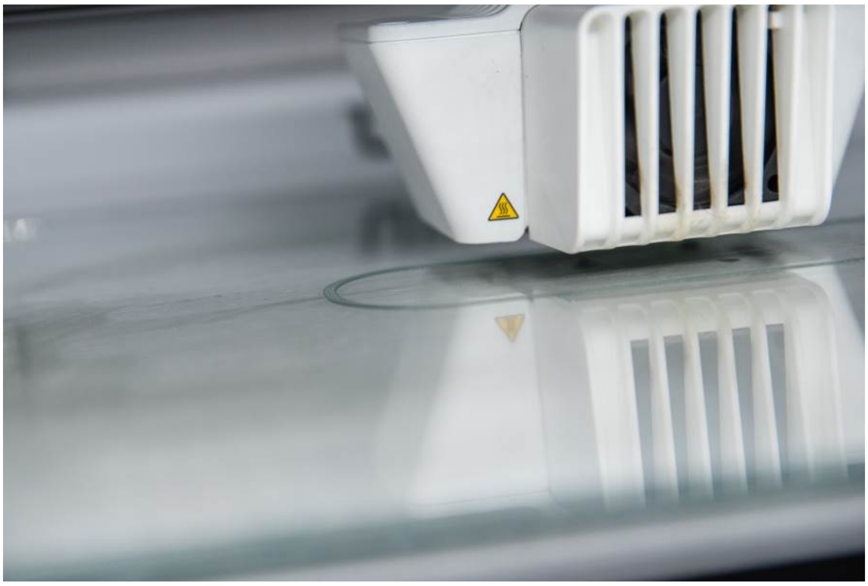


Figure 19: Tweaking the first layer extrusion settings can help with improving first layer adhesion

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

Some materials such as glass filled propylene 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, polycarbonate and which tend to detach once the build plate cools.

Link to temperature database: http://b.link/magigoolist

3 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.

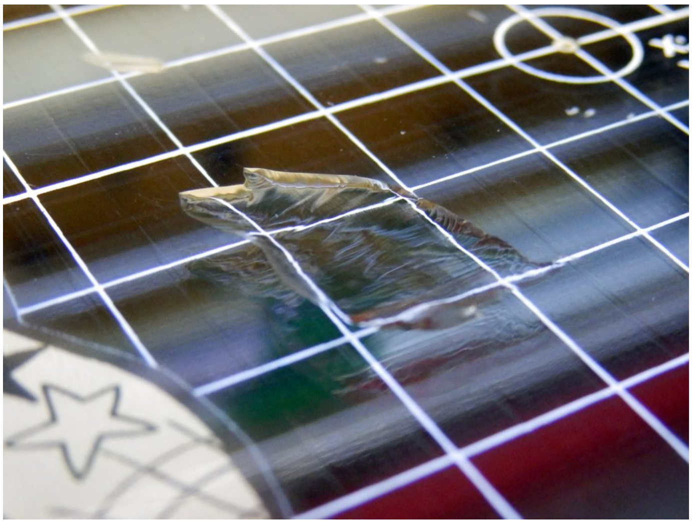


Figure 20: Chipped glass plate, image via richrap.blogspot.com

3.1 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).

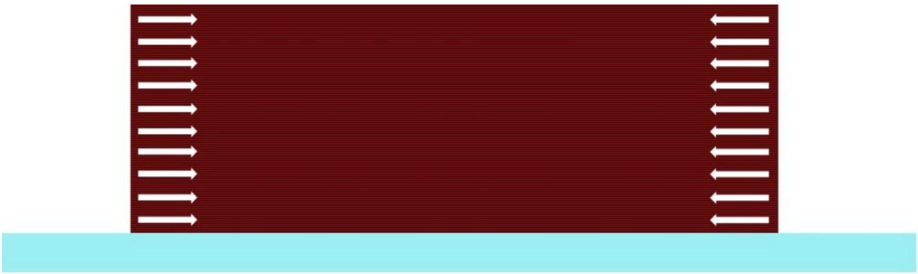


Figure 21: Printed part which has just finished printing, the part and the build-plate are roughly at the same temperature. The part is already exerting some force as a result of shrinkage during the solidification of the deposited FDM thermoplastic

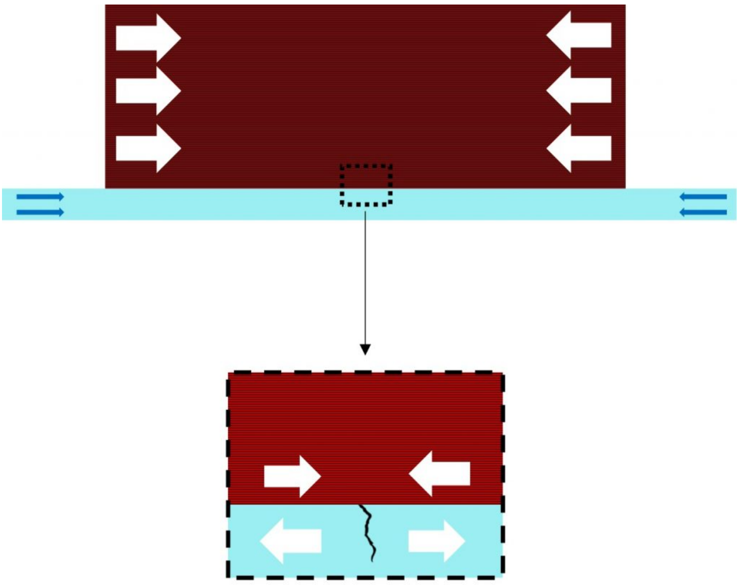


Figure 22: part and glass build-plate cool down, since the part experiences a larger rate of shrinkage than the glass build-plate the net result is a tensile force on the glass surface which leads to crack initiation and propagation.

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.

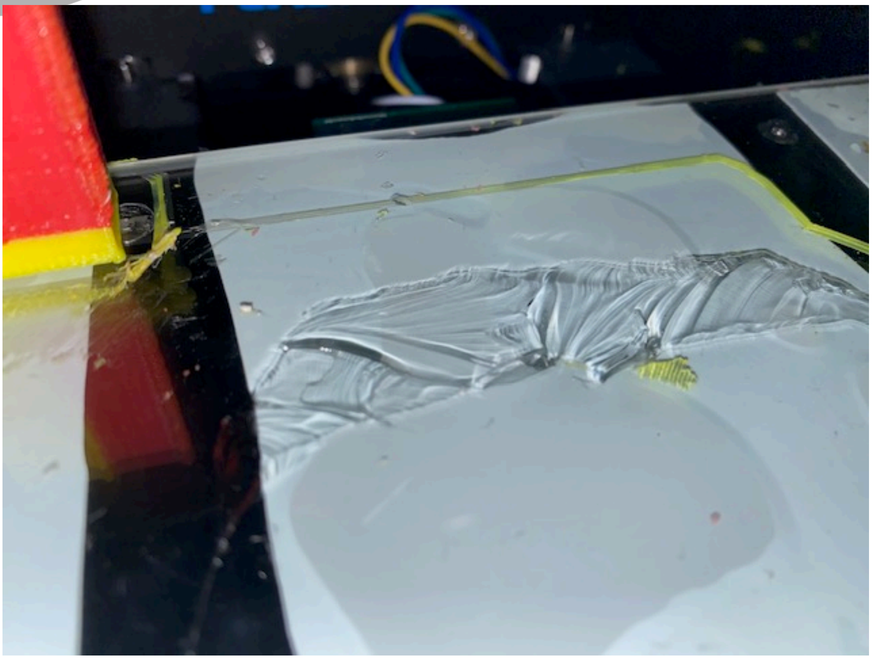


Figure 23: A more severe case of glass chipping resulting from the use of improper adhesives, in this case using Magigoo PA with ABS filament

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.

3.2 What makes glass prone to chipping?

3.2.1 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 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 be a recipe for making glass chips.
- **Modified filaments**, some filaments such as PLA, Nylon and ABS are modified to improve print-ability and reduce warping. This can cause these types of filaments to over-adhere.

3.2.2 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:

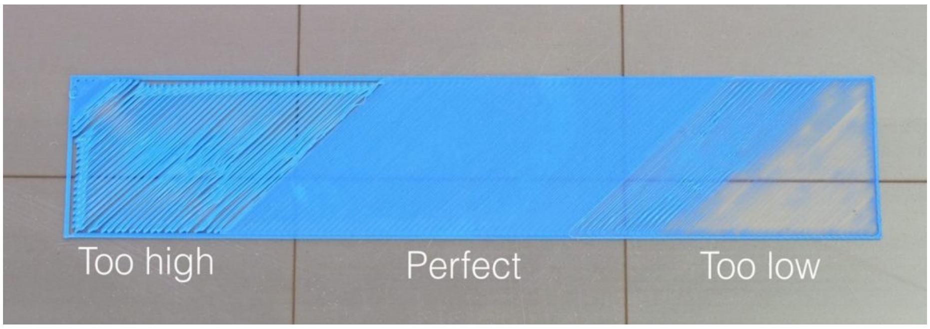


Figure 25: Printing with the nozzle too close to the build-plate

- **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.

3.2.3 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

4 How to prevent damaging glass build-plates

4.1 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 by about 85% significantly improves the ease of part removal. This value also ensures that the adhesion still adequate to prevent warping.

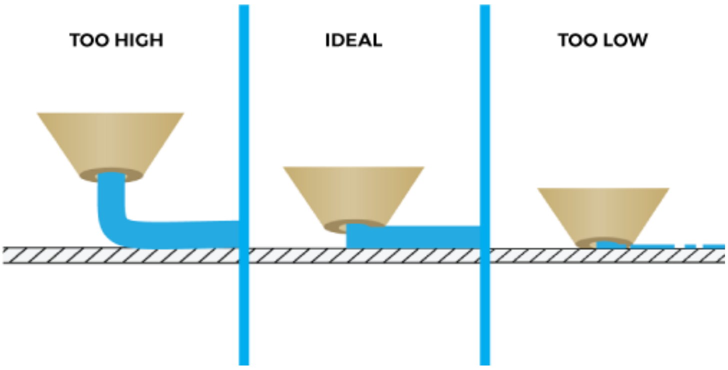


Figure 27: Ideal nozzle height, image via matterhackers.com

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

4.2 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!

4.3 Use your adhesive, as intended!

The Magigoo range comes in a variety of different flavors 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.

4.3.1 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 PC, since this can increase the chances of damaging you build-plate

4.4 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

5 Other Magigoo FAQs

5.1 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).

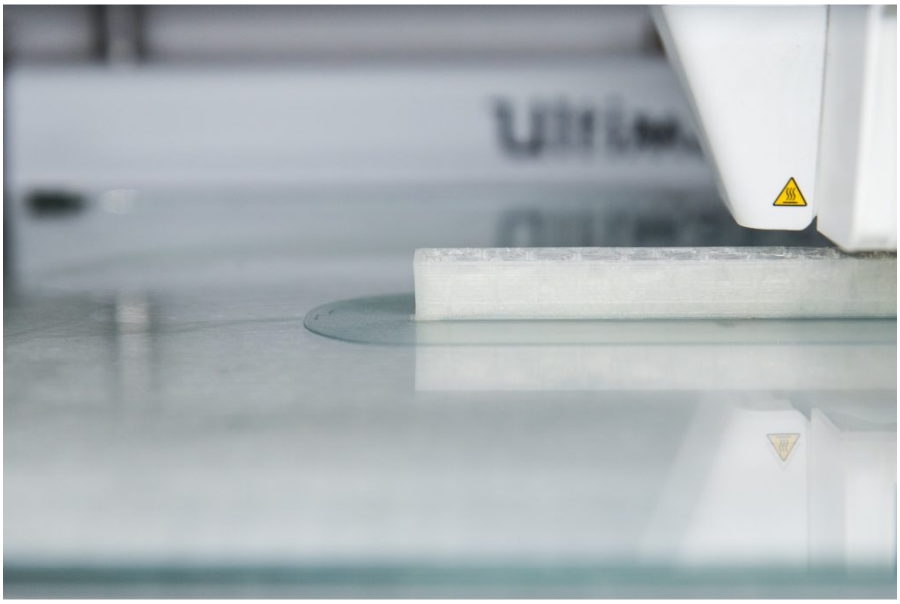
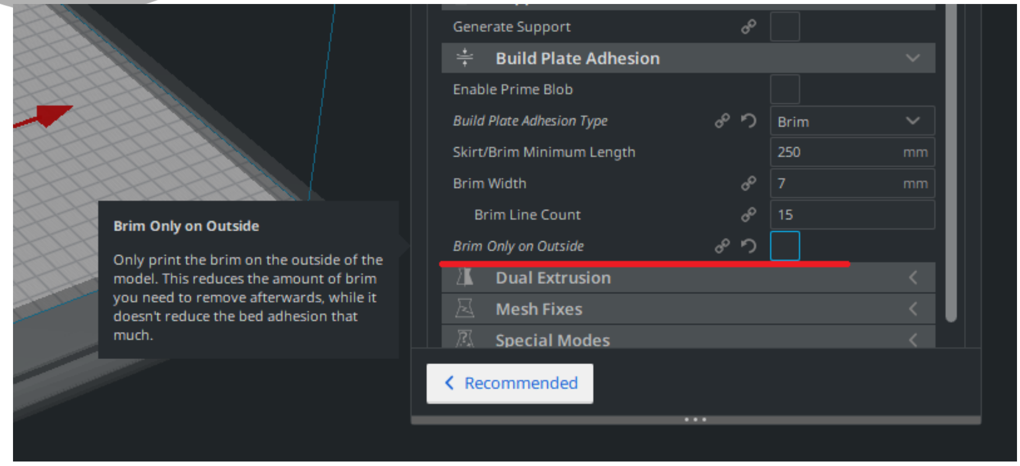


Figure 30: Printing using a brim helps improve first layer adhesion.



Brim Only on Outside

Only print the brim on the outside of the model. This reduces the amount of brim you need to remove afterwards, while it doesn't reduce the bed adhesion that much.

Generate Support ☐
Enable Prime Blob ☐
Build Plate Adhesion Type: Brim
Skirt/Brim Minimum Length: 250 mm
Brim Width: 7 mm
Brim Line Count: 15
Brim Only on Outside ☐

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.

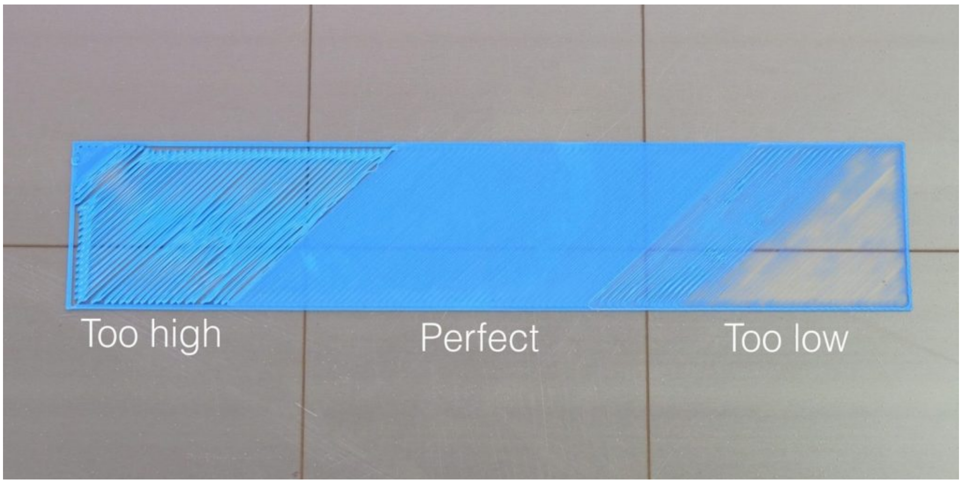


Figure 32: Image showing results of the nozzle being too far away, at an ideal level and too close from/to the build plate – image via Prusa Printers

- **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.

5.2 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 10 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® Pro 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**.

5.3 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)

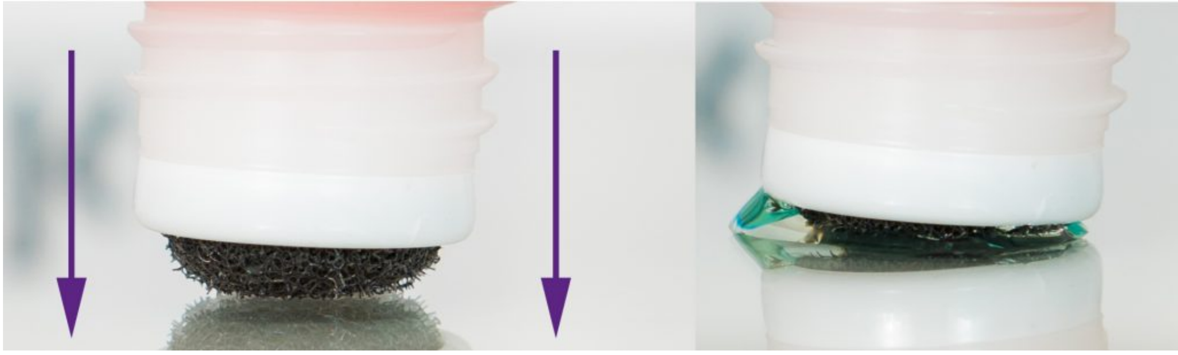


Figure 33: Depress nib on build plate to activate nozzle, Magigoo should then easily flow out!

- 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.

5.4 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 considering the higher cost of a failed print and given that 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 taking into account the material, time and energy cost for a single failed print.

Having said so, 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 its better 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 at least anyways.

5.5 The foam on my applicator is damaged, what can I do?

In some cases, the foam application tip can wear off or tear. Please note that the lifetime of the foam tip is significantly reduced if you apply Magigoo® on a build-plate which is hotter than 40 °C. The lifetime of the foam tip is also reduced if the foam is dirty or glue is left to dry out on the foam tip. If for some other reason your foam tip is damaged please contact us on **feedback@magigoo.com** so we can look into the issue. In the meantime, Magigoo® can still be poured onto the build-plate by gently pressing the bottle whilst pushing the valve against the build-plate and can be spread to the required printing area using a sponge tipped brush, regular paint brush or even a paper towel.

5.6 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.

5.7 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 if water does not suffice, we recommend using ethanol or isopropanol.

5.8 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.

6 Using Magigoo with engineering filaments

6.1 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

6.2 Nylon

6.2.1 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.

6.2.2 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.

6.2.2.1 *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!

6.2.2.2 *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 suggested 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.

6.2.2.3 *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 the 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.

6.3 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.

6.3.1 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/m²)	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.

6.3.2 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!

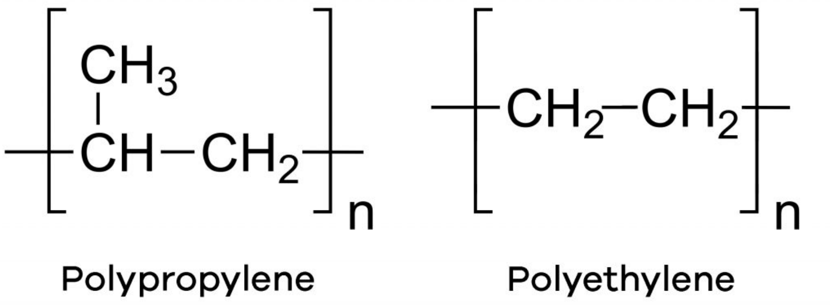
Another requirement for successful printing with PC is a printer enclosure, especially for medium and larger sized prints. Using an enclosure retains some of the heat generated by the heated build-plate. This will reduce the tendency for poor layer adhesion and for warping.

Professional and engineering grade materials tend to have a narrow range in which they show optimum print-ability 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.

6.4 Polypropylene

6.4.1 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

6.4.2 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.

6.4.3 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

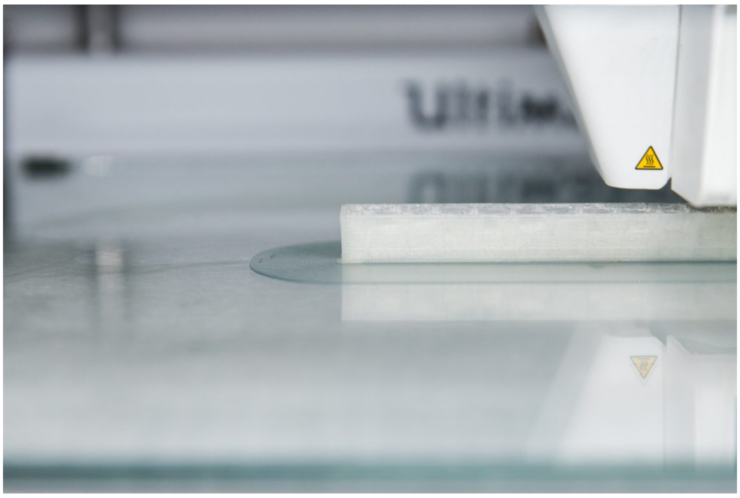
6.5 How to succeed when printing PP

6.5.1 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 regards 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.

6.5.2 Tips for better bed adhesion



Using a brim helps prevent warping when printing with PP materials

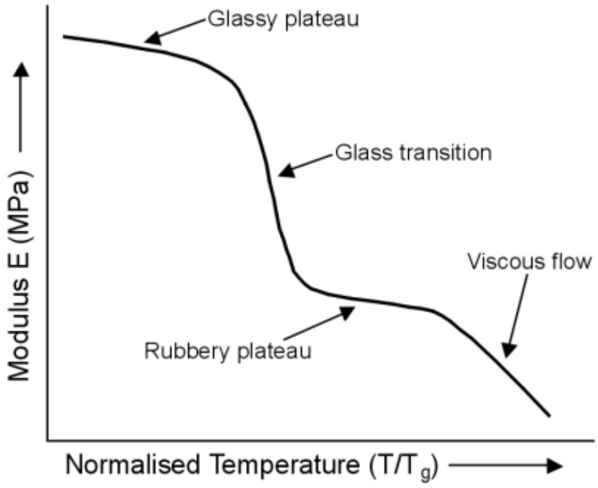
- **Use a brim:** For the majority of PP prints a 10-20mm brim is an absolute necessity for a successful print with little to no warp
- **Start high then go low:** All PP materials benefit from using an additional 10 ˚C for the first layer on both the build-plate and the nozzle. This helps the first 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)

6.6 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.

6.6.1 What makes a material Flexible?

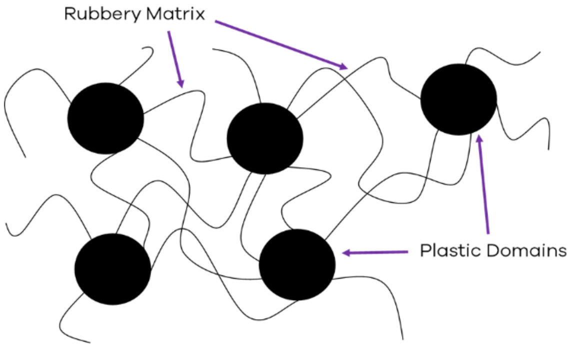
Thermoplastic materials get softer as the temperature increases since at higher temperatures polymer molecules can move around more freely. In fact at particular temperatures (or over a temperature range) thermoplastic materials will get markedly softer and start behaving more like a soft rubbery material 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 it's original shape. However once heated the plastic domains will soften and enable the material to be re-molded.

6.6.2 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.

6.7 Choosing the right TPE

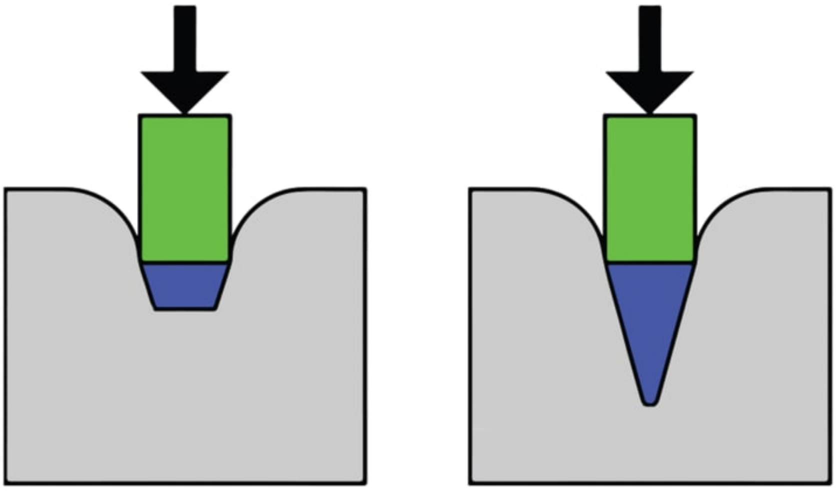
Choosing the right flexible material for your application will of course depend on the mechanical requirements. In fact one of the primary reasons TPEs are considered is their flexibility and elasticity. For this reason TPEs commonly have a clear indication of their Shore hardness, which indicates their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

6.7.1 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 (right) and Shore D (left) Durometer tips. The shore D durometer has a pointed tip and is loaded with a higher force since it is intended for used 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.

6.8 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.

6.8.1 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 tolerance. Specialised extruders often have very tight tolerances and dual drive in order to better push the filament to the nozzle.

6.8.2 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 amount 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.

6.8.3 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 strings can be easily removed by careful application of hot air from a heat gun.

6.8.4 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