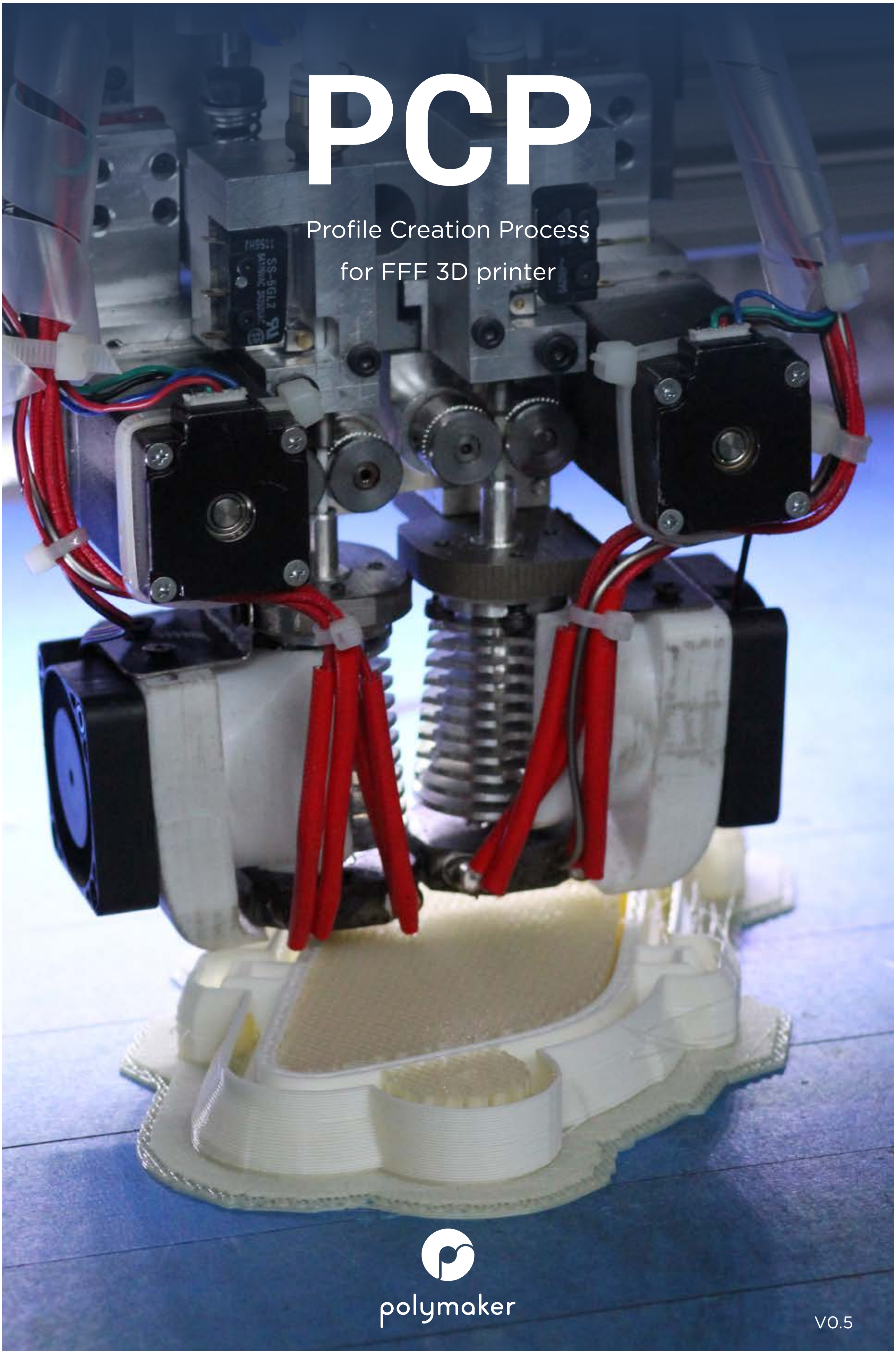
PCP

Profile Creation Process

for FFF 3D printer



polymaker

V0.5

CONTENT

What is the Profile Creation Process from Polymaker?

The profile creation process (PCP) allows users to rapidly develop a printing profile for a given material/ printer.
It is important to consider all of these factors to build a profile:

Material
Printer
Environment
Model geometry
Model purpose

Polymaker came up with a process which allows users to build their own profile considering the material, printer and environment. This base profile will then be used to create the custom profile taking into account the models geometry and purpose. The process is also designed to educate users about the 3D printing process and provide them with the skills and knowledge to optimize their own profiles.

The PCP is divided in 4 steps:

Step 1: Extrusion Flow
Step 2: Flow Management
Step 3: Cooling Fan
Step 4: Warpage

The PCP can be completed in less than 7 hours, using no more than 300g of materials

Each of these steps has a specific objective and introduces an important concept about the FFF 3D printing process. Every concept is introduced with a scientific explanation and a basic print profile (inc model).

The steps can be repeated multiple times, should the user wish to refine their print results.

CONCEPT 1

This concept highlights the difference between the settings provided in the slicer and the actual printing conditions. tt

Common terms

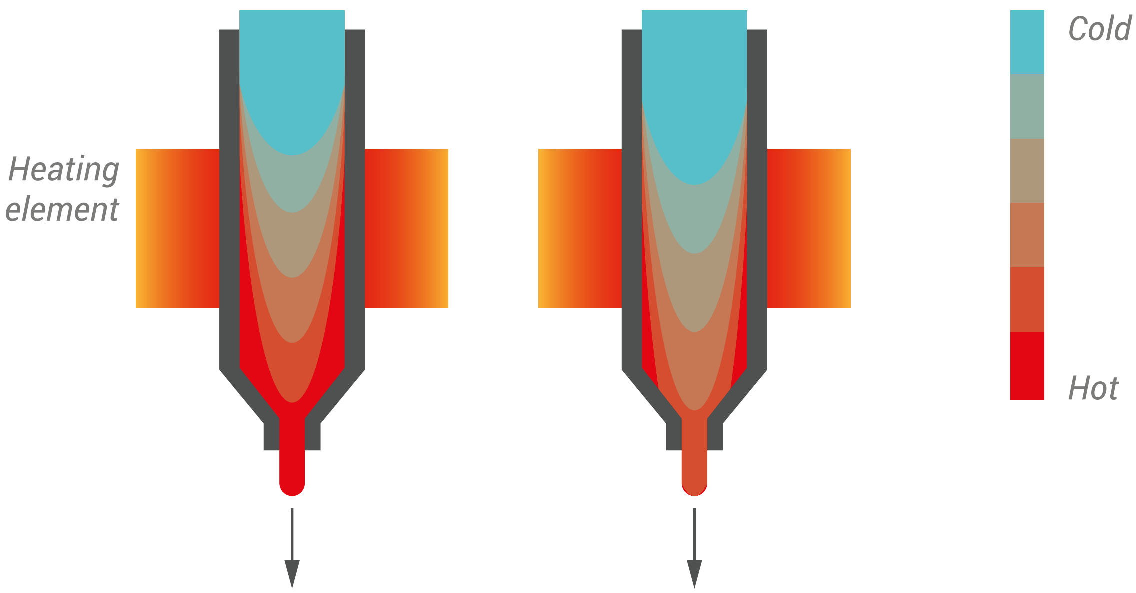
Printing temp.:	Usually used to define the heat block temperature, given in °C
Printing speed:	Usually used to define the print head speed when printing, given in mm/s

Useful terms

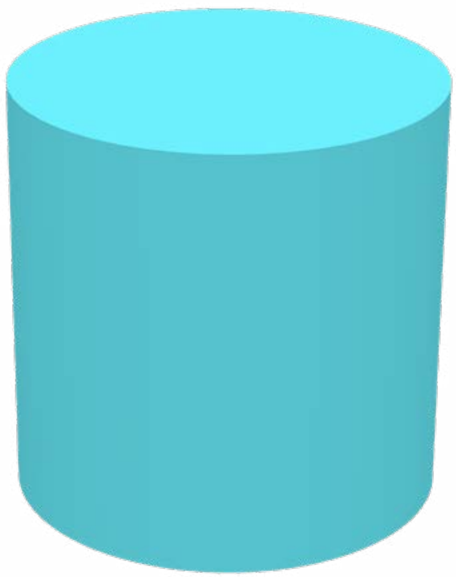
Extrusion Temp.:	The temperature at which the plastic exit the nozzle, given in °C
Extrusion Speed:	The speed at which the plastic is extruded from the nozzle, given in mm3/s

Same printing temperature/speed → Different extrusion temperature/speed

Printing Temperature = 200°C Printing Speed = 50mm/s Layer height = 0.1mm	Printing Temperature = 200°C Printing Speed = 50mm/s Layer height = 0.2mm



Extrusion Speed = 2mm3/s Extrusion Temperature = 200°C	Extrusion Speed = 4mm3/s Extrusion Temperature = 195°C



Printing settings

Recommended settings for PLA material. Please refer to the manufacturer recommended settings for any other material.

Printing Temperature	210˚C
Printing Speed	50mm/s
Extrusion Width	0.4mm
Layer Height	0.2
Top/Bottom layer	0/3
First layer speed	20mm/s
Cooling Fan	100%

Print in vase mode (single outline, 0% infill)

Test

OBJECTIVE

The objective of this test is to verify the right printing speed and temperature to ensure a good extrusion flow.

TEST

The test consists of printing a cylinder in vase mode and checking the quality of the print.

NOTE

Make sure the cooling settings won't override the printing speed.

MODEL

The Extrusion Flow model is a cylinder of 5cm height and 5cm diameter. This simple model has a rounded, wide surface to easily adhere to the bed whilst ensuring a consistent printing speed, thus a consistent extrusion.

Results

When the printing has finished, you can study two different elements of the print:

CONSISTENT EXTRUSION:

The surface must be smooth and even. (free of holes, bubbles or other artifacts). If its not, the holes can be explained by under extrusion which can be solved by increasing the extrusion temperature. The bubbles can be explained by moisture issues which can be solved by lowering the extrusion temperature or drying the material. Aditional artifacts can be caused by other factors, please contact Polymaker for help.

LAYER ADHESION:

You must be able to pinch the model without it cracking along the layers.

If it breaks along the layer, you can increase the layer adhesion by increasing the extrusion temperature or lowering the model cooling system.

CONCEPT 2

This concept explains the reasons behind material oozing in 3d printing and the subsequent creation of stringing or other artifact issues. Once the root causes of such a phenomenon are well understood, there are several methods available to prevent this issue.

Oozing causes:

Residual Pressure

Residual pressure is a result of the printer building up pressure within the nozzle to extrude at a certain volumetric speed. This pressure can never be 100%, completely discharged from the nozzle and therefore, it will keep extruding slightly. Therefore, it will keep slightly extruding.

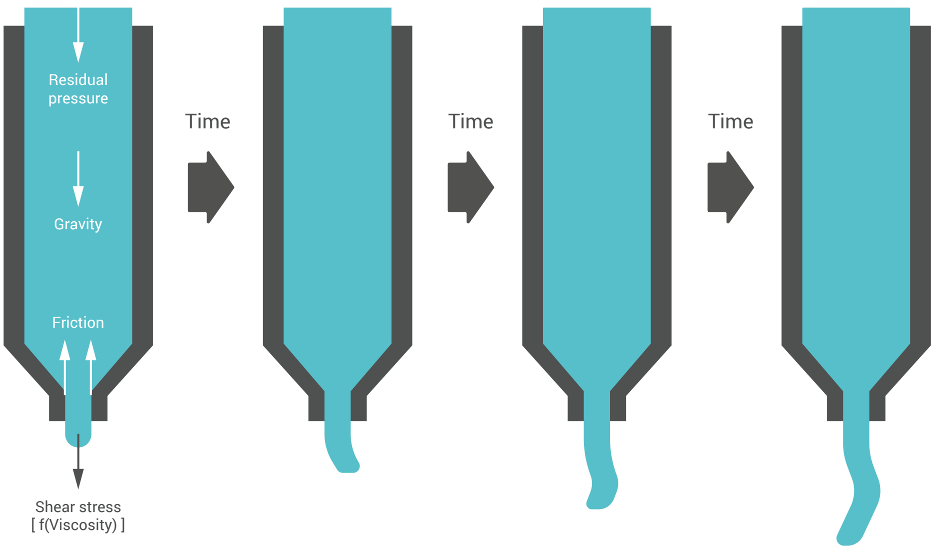
Gravity

Gravity will always pull the filament out of the nozzle, and if the gravitational force is stronger than the friction of the plastic against the nozzle's internal surface (and shear within the plastic), it will ooze out. Note that the friction between the internal surface of the nozzle and the plastic can be increased

Time

The amount of material oozing from the nozzle also depends on the amount of time the nozzle is inactive. The greater the duration, the larger amount of material there is.

Oozing amount = f(residual pressure, 1/plastic viscosity, time)

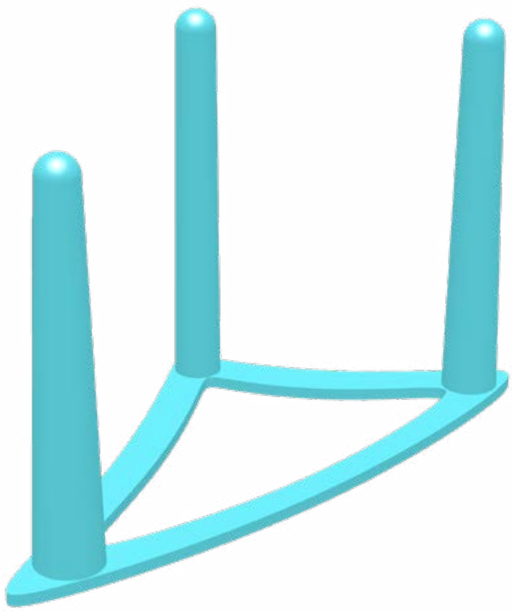


To decrease the Oozing amount you need to:

Decrease residual pressure: Increase retraction settings; Increasing coasting; Decrease extrusion speed; Increase the printing temperature

Decrease time: Increase the travel speed

Increase plastic viscosity: Decrease the printing temperature



Printing settings

Recommended settings for PLA material. Please refer to the manufacturer recommended settings for any other material.

Printing Temperature	210˚C
Printing Speed	30mm/s (shell)
Printing Speed	50mm/s (infill)
First layer speed	20mm/s
Extrusion Width	0.4mm
Top/Bottom layer	3/3
Shell number	3
Cooling Fan	100%
Infill	20%
Layer Height	0.2
Top/Bottom layer	3/3
Shell number	3
Cooling Fan	100%
Infill	20%
Layer Height	0.2

Test

OBJECTIVE

The objective of this test is to verify the right printing settings to ensure a good flow management of the plastic.

TEST

The test consists of printing a series of pillars separated by various distances and checking the quality of the print.

NOTE

Make sure the cooling settings won't override the printing speed. Some slicers have specific settings or intrinsic ways to manage the oozing. Make sure you reckon them during your test.

MODEL

The Flow Management model is 3 pillars, 5cm high and linked by thin rounded plates. This geometry forces the printer to process a series of extrusion and non-extrusion steps, allowing the performance analysis of the printer's flow management with the material.

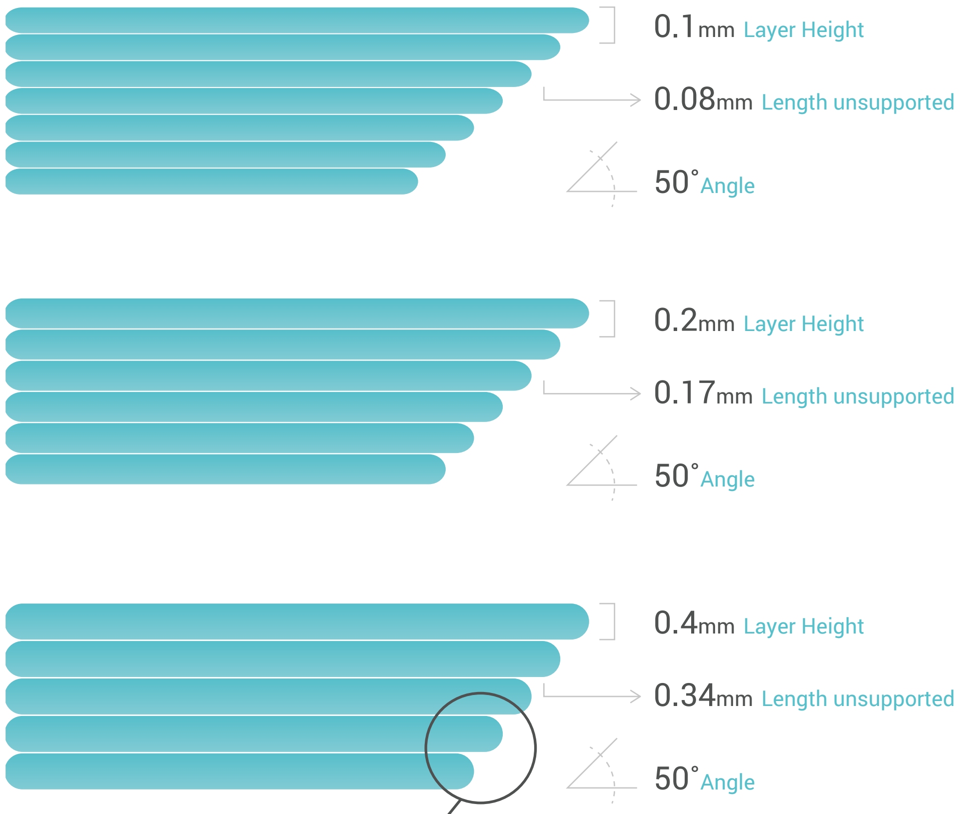
Results

When the printing has finished, you can study:

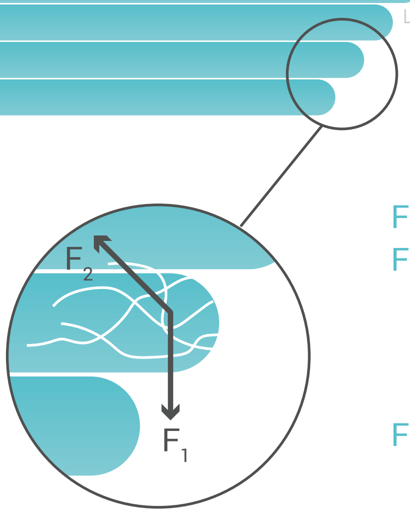
STRINGING: The print must be free of plastic strings. If it is not, the strings can be explained by an excess of residual pressure or simply the gravitational pull of material out of the nozzle. the plastic out of the nozzle. These issues can be solved by increasing the retraction settings, increasing the travel speed or lowering the extrusion temperature.

CONCEPT 3

This concept explains the different challenges encountered when trying to print overhanging surfaces. It is important to understand that the factors affecting overhanging areas are the same as those that affect bridging. Once the challenges are well identified, it is easier to define the factors and modify them to improve the overhanging surfaces.



Same overhang angles
Different unsupported area



F1 = f(mass, 1/rigidity of the polymer) = f(layer height, overhang angle, temp.)

F2 = f(polymer stress) = f(1/temp., extrusion speed)

It is impossible to balance F1 and F2 so the idea to have good overhang surfaces is to have F1=F2=0

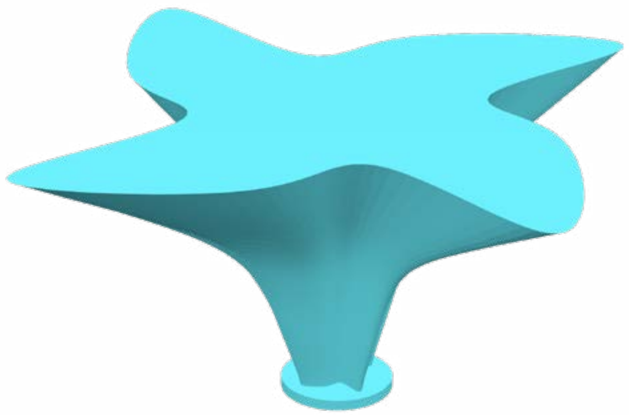
$F_1=0$

Decrease the layer height, decrease the overhang angle, decrease extrusion temperature or increase cooling.

$F_2=0$

Increase extrusion temperature, decrease extrusion speed

Other factors should be taken into account when adjusting to extreme values. It is possible to print “on air” when using the right settings



Printing settings

Recommended settings for PLA material. Please refer to the manufacturer recommended settings for any other material.

Printing Temperature	210˚C
Printing Speed	30mm/s (shell)
Printing Speed	50mm/s (infill)
First layer speed	20mm/s
Extrusion Width	0.4mm
Top/Bottom layer	3/3
Shell number	3
Cooling Fan	100%
Infill	20%
Layer Height	0.2

Test

3

OBJECTIVE
The objective of this test is to verify the right printing settings to ensure overhang surface quality above a certain threshold angle to avoid using support structure.

TEST
The test consists of printing a model presenting different degrees of overhang surfaces in different shapes.

NOTE
Even for materials that do not recommend the cooling fan it is important to note that only 10% of cooling fan can dramatically increase the printing quality of the model whilst having minimal affect on the layer adhesion.

MODEL
The Cooling Fan model is a star shape gradually scaled up while extruded. It forces the printer to print overhanging surfaces at different degree but also with different shapes and distances between the attach points. The model has an added disk at the bottom of the model to improve the stability of the model during the printing process.

Results

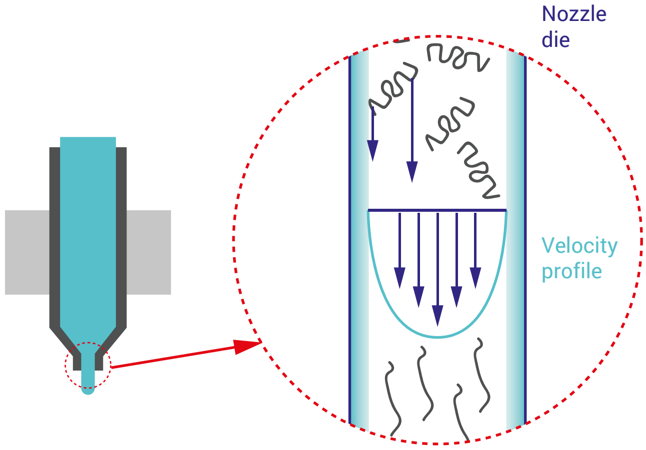
When the printing has finished, you can study the overhanging surfaces:

Depending on your need, you can define the threshold angle at which the material should provide a good surface. Note the model provide overhangs from 0˚C to 60˚ overhang. In most cases the threshold is 45˚. To improve the overhang surface you can:

Decrease the layer height
Increase cooling
Decrease extrusion speed

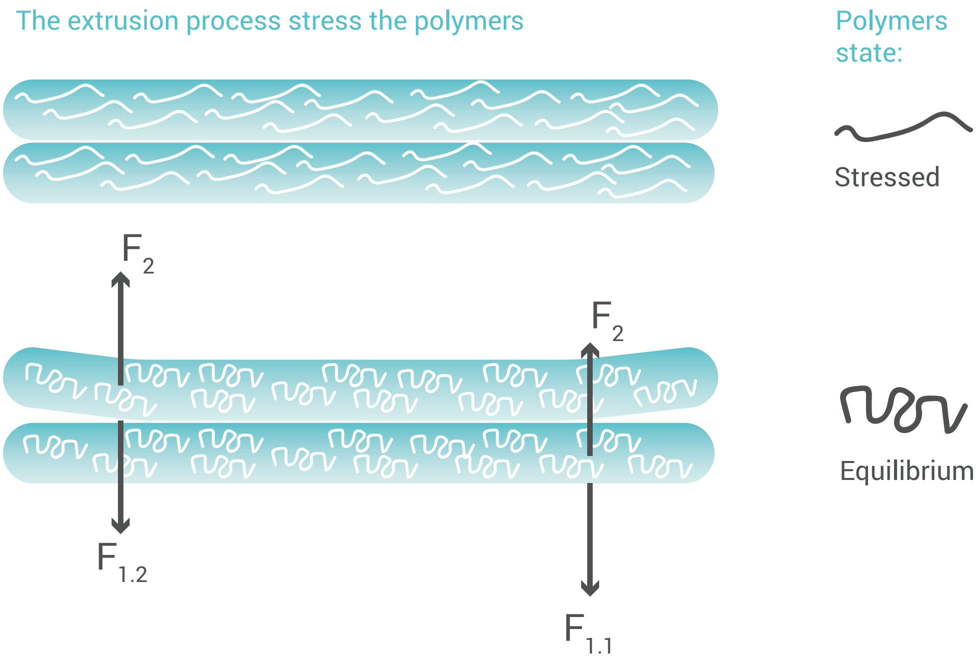
CONCEPT 4

This concept tackles the science behind warping issues which is one of the most common challenges in 3D printing. Once the root causes of this phenomenon are well understood, it is easier to highlight all the possible solutions to prevent this deformation from happening.

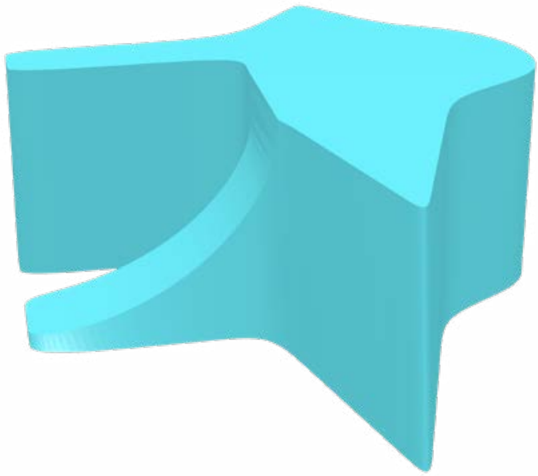


The stress built up during the process is responsible for the part deformation

$F_{1.1}$ Bed adhesion
$F_{1.2}$ Layer adhesion
F_2 Warp caused by polymer stress



The origin of the internal stress is still under debate, and depending on your 3D printer configuration, many factors may be contributing to the as-built internal stress. Here is one hypothesis which should be considered for all FDM machines: During the extrusion process the polymer is forced through a die (small hole/ nozzle), and during this step the polymer chains will be stretched to a stress state, then stuck to the build plate or a previous layer of plastic. This stress will slowly be released over time, however if the temperature does not allow the polymer to freely move enough to release the stress, or if the layer is not well stuck to the bed or the build plate, the accumulation of this stress throughout the layers will force the part to macroscopically deform.



Printing settings

Recommended settings for PLA material. Please refer to the manufacturer recommended settings for any other material.

Printing Temperature	210˚C
Printing Speed	30mm/s (shell)
Printing Speed	50mm/s (infill)
First layer speed	20mm/s
Extrusion Width	0.4mm
Brim	4mm (10 outlines)
Top/Bottom layer	3/3
Shell number	3
Cooling Fan	100%
Infill	20%
Layer Height	0.2

Test

OBJECTIVE

The objective of this test is to verify the right printing settings and environmental condition to ensure good dimensional stability during the printing process.

TEST

The test consists of printing a specific shape meant to create a lot of internal stress which could deform the model if the print settings are not properly defined.

NOTE

Make sure the bed is well leveled as it can have a big influence of the test results.

MODEL

The Warpage model is a thick design featuring multiple shapes to create different amounts of stress within the material:

Long sharp edge

Short sharp edge

Long square edge

Short round edge

Long slide shape

Results

When the printing has finished, you can study how warped the different parts of the model are.

You can reduce the internal stress which is causing the part to warp by:

- Increasing the environment and bed temperature (closer to the glass transition temperature of the material)
- Use a bed surface on which the material can stick very well
- Increase the extrusion temperature

GO FURTHER

TEST 1
To go further it is possible to use this test to establish the range of printing speeds with consistent extrusion flow for a set printing temperature, layer height and extrusion width. For this you can print a cylinder at a set printing temperature, layer height and extrusion width, then vary the speed during the printing process and finally analyze the results in the same way as above.

TEST 2
To go further it is possible to use extra settings to manage the flow:
Coasting: Coasting will stop the extrusion before the end of the layer using the oozing to complete the layer.
Wiping: Wiping will make the nozzle finish the layer, stop the extrusion and continue its trajectory above the beginning of the current layer to wipe the nozzle.
Extra restart extrusion: Extra extrusion will add extra material in the nozzle at the beginning of the next printing island to compensate the material lost during oozing and also help to quickly build up the needed pressure.
More: Slicer has other different settings to manage the flow which you can find more info on the slicer or linked website.

TEST 3
To go further it is possible to tinker with the number of shells, infill shell overlap and other settings to improve the overhang surfaces even more. Also it is important to note that even for materials that do not recommend the cooling fan.

TEST 4
To go further it is possible to repeat this test with multiple bed surfaces and temperature ranges to find out the surfaces which can match the material as well as the best range of bed/environment temperature depending on the model size and geometry. Some printers are limited in the temperature range but can still print models with specific size and geometry without any warping issues.