fillamentum forMAKERS

VERSION UPDATED IN 8/2024

SO LET'S CREATE AWESOME

3D PRINTING GUIDE
PETG

Plug in and print. Compatible with high speed printers.

Second life Sustainable spool

Dry before use it

BASIC OVERVIEW

HARDNESS

IMPACT RESISTANCE

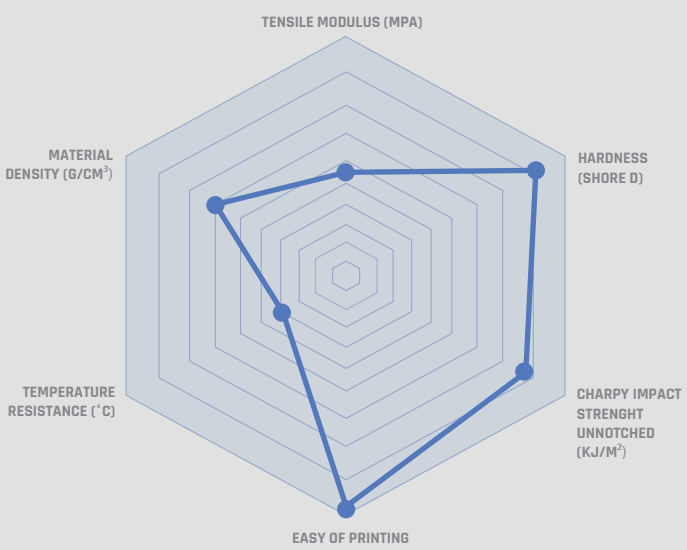
FLEXIBILITY

EASY OF PRINTING

WEATHER RESISTANCE

WEAR AND ABRASION RESISTANCE

DETAILED VIEW



BASIC NON HIGH-SPEED PRINTERS SETUP

Print Temp: 230 - 275 °C

Bed Temp: 65 - 75 °C

Printing Speed: 40 - 60 mm/s

Cooling Fan: 0 - 30 %

HIGH SPEED PRINTERS SETUP

GLOSSY AND MATTE FINISHES WITH TWO SETUPS

When aiming for consistent surface finishes, the printing temperature plays a crucial role. A common issue arises when using the same printing temperature, as the extrusion temperature can vary significantly, resulting in different surface finishes.

GLOSSY FINISH

Print Temp: 255 - 285 °C

Printing Speed | Outer line: 100 - 300 mm/s

Bed Temp: 65 - 80 °C

Cooling Fan: 0 - 40 %

MATTE FINISH

Print Temp: 235 - 255 °C

Printing Speed | Outer line: 100 - 300 mm/s

Bed Temp: 65 - 80 °C

Cooling Fan: 0 - 40 %

DISCLAIMER:
Drying: highly recommended 65 °C for 4 hours
Storage 15 - 25 °C with low humidity.

NOTE:
THE glossy finish can slightly (up to 5 %) affect the mechanical properties of the final print. Conversely, for a matte look, adjust your setup to achieve a uniform, non-reflective finish without compromising the mechanical integrity. This guide will help you fine-tune both setups for the desired aesthetic and functional outcomes.

TIPS FOR OPTIMAL PETG PRINTING

1 Bed Adhesion:

Best Surfaces: Use PEI, mirror, or glass bed surfaces for strong adhesion.

Adhesives: Apply Magigoo or PVA glue for extra hold. PETG adheres strongly, so avoid printing directly on glass without an adhesive layer to prevent damage.

Layer Settings: Use a brim (5 mm) to improve bed adhesion and reduce the likelihood of edge lifting.

2 Cooling and Overhangs:

Cooling Fan Control: PETG typically doesn't require high cooling; start with 0 - 15 % for regular prints. Increase up to 40 % for sections with extensive overhangs to reduce stringing or if youre printing very fast.

Print Slow: Slower speeds (around 40 - 80 mm/s) are helpful for achieving high-quality overhangs.

3 Layer Bonding:

Higher Temps for Strength: For stronger layer bonding, print at the higher end of the temperature range, around 250 - 270 °C. This temperature helps enhance layer adhesion.

Infill Overlap: Set the infill overlap slightly higher (20 - 30 %) to improve inner structure bonding.

4 Moisture Management:

Drying: PETG is hygroscopic, so dry it at 65 °C for 4 - 5 hours if it has absorbed moisture. Moisture can lead to stringing, oozing, and poor layer adhesion.

Storage: Store PETG in an airtight bag with silica gel to keep it dry.

5 Stringing Reduction:

Retraction Settings: Adjust retraction to around 6 - 7 mm for Bowden extruders and 3 - 5 for direct drive extruders. Speed set to 20 - 25 mm/s which can reduce stringing without causing under-extrusion.

Temperature Control: Lowering the print temperature slightly can also reduce stringing. A good range is 230 - 240 °C for string-prone prints.

Test Models: Use a simple stringing test model to fine-tune settings before starting on complex prints.

TIPS BEFORE YOU START

HEATED BED SURFACE:
PEI, mirror/glass

ADHESIVE:
Magigoo, 3Dlac, PVA glue

RAFT/SKIRT/BRIM:
Skirt / Brim 5 mm

HEATED CHAMBER/ ENCLOSURE:
Not needed

COOLING:
It is not recommended to use more than 30 % of fan speed, as fast cooling could lead to improper layer bonding. We recommend using no cooling for standard objects, which leads to stronger parts. For bridges and big overhangs, it is possible to go up to 50 % for the desired layer.

DATASHEETS AND MORE... 24/7 AVAILABLE

OTHER TIPS

Surface Preparation: For best adhesion, use a PEI surface with a light adhesive such as Magigoo PETG. Avoid printing PETG directly on glass without a release adhesive, as it can adhere too strongly.

Bed Heating and Brim/Skirt: A heated bed is essential for PETG to avoid warping; a temperature of 70 - 75 °C is generally effective. Adding a skirt or brim (5 - 10 mm) provides extra stability to the base layer.

Enclosure Not Required: An enclosed printer is not necessary for PETG. It generally prints well without a heated chamber, though a stable ambient temperature can improve performance slightly for larger models. Increasing temperature inside chamber won't make prints better or make printing easier.

Avoiding Blobs and Zits: High temperatures can cause PETG to form small blobs on the surface. Adjusting retraction and using coasting settings in your slicer can help minimize these artifacts.

Optimized Cooling and Speed: High-speed printing of PETG can be achieved with printing speeds up to 100 - 200 mm/s, depending on the printer's capabilities. For speeds above 100 mm/s, it's recommended to use stronger cooling (around 40 %) but only after the first few layers. A bed temperature of 70 - 80 °C should be maintained.

Use of Hardened Nozzles: At high speedsis better to use hardened steel nozzle. They are recommended for such high-speed PETG applications to ensure longevity.

Layer Height and Wall Thickness: For better layer adhesion at high speeds, use larger layer heights (0.2 - 0.3 mm) and increase wall thickness to improve strength and reduce vibrations during high-speed movements.

VASE-mode: When printing PETG in vase mode, it's essential to use consistent extrusion with a well-tuned flow rate to achieve smooth, watertight walls. Set the Spiralize Outer Contour option in your slicer, which creates a continuous, single-layer wall to reduce seams and layer lines. Use a lower print speed (generally 20 - 40 mm/s) and a slightly higher nozzle temperature (around 240 - 250 °C) to ensure proper adhesion. Avoid cooling fans.

Conclusion: Following these practices should help you get the best results with PETG, especially when balancing speed and surface (matte X glossy) quality. We are happy to provide you with more information and specialized slicer settings on our helpdesk or chatbot.

ARE YOU MISSING THE RIGHT ANSWER?

CHOOSE THE PLACE YOU'D LIKE TO CONNECT WITH US.

CLICK FOR FILLA FELLA CHAT BOT
24/7 INSTANT ANSWER

HELPDESK@FILLAMENTUM.COM
REACTION TIME 12 - 16 HOUR

CALL US IF YOU WANT +420 725 463 731
BETWEEN 8 AM AND 4 PM CENTRAL EUROPEAN TIME.

WE GUARANTEE THE BEST QUALITY WITH CPK PROCESS MEASUREMENT.

At Fillamentum, we go beyond achieving a lower filament diameter. We focus on CPK (Process Capability Index) could be known as a Sigma within Industry. It is a crucial measure that ensures every spool of filament meets the highest standards. Here is Why CPK is essential for you and why it is more important than just diameter.

WE PROVIDE FILAMENT INSPECTION

RECOMMENDED

3D Printing Industry AWARDS 2021 WINNER OF 2021