Fusion3

EDGE 3D Printer

OPERATION: ALTERNATIVES TO PETG

Revision 02/02/2023

WHY WE DON'T LIKE PETG & WHAT TO USE INSTEAD

PETG is a very popular material for hobbyist 3D printers to use. Compared to PLA, it has higher temperature resistance, better flexibility, and better chemical resistance.

Most hobbyist printers cannot reliably print materials that require higher temperatures than PETG; PETG is the best they can do. And so it gets recommended a lot for a lot of different applications.

EDGE is capable of printing many more materials, and at much higher temperatures, than hobbyist printers. Because of this, PETG is rarely a good choice for any application.

Our goal with this document is three-fold:

- To discuss & share better materials for different aspects of performance that you might be trying to get from PETG.

- To discuss the problems with printing PETG.

- To share some helpful tips and tricks to get PETG to print well if you absolutely must use it.

ALTERNATIVES TO PETG

People choose PETG for a few reasons:

- Higher temperature capability

- More flexible / more impact resistant

- Chemical / solvent resistance

- UV stability

- Food grade / food safety

Fortunately there are excellent alternative materials that can fulfill all of these requirements. We've prepared this handy table to allow you quickly & easily find the appropriate alternate material for your application.

FUSION3 PETG ALTERNATIVES LOOKUP TABLE

Use this table to locate a suitable alternative material for your specific application

Revision 02.02.2023

MATERIAL NAME	Tg (deg C)	FLEXIBILITY & TOUGHNESS	CHEMICAL RESISTANCE	UV STABLE	FOOD SAFE	DIFFICULTY LEVEL
PETG	85	X	X	X	X *	Difficult
ABS	105				X *	Easy
ASA	100			X		Easy
PC+PBT	120	X				Easy
PC	140	X			X *	Moderate
PC-ABS	115-130	X			X *	Moderate
Alloy910	90	X	X			Moderate
Nylon PA12	48	X	X	X		Moderate
Nylon 680	93	X	X		X	Difficult
MatterHackers NylonX	~85 **	X	X			Moderate

* = food safe is a complex topic and just because a material can be food safe doesn't mean the finished product is, OR that all materials in this family can be considered food-safe.

** = value not provided, estimated from other TDS information

General Material Recommendations

For general printing and most applications, **we recommend you start with ABS instead of PETG**. ABS has comparable strength and layer bond strength to PETG when printed on EDGE, and has higher temperature capabilities vs PETG. Hobbyist printers usually make weak prints with ABS because they cannot get the bed and chamber (if enclosed) hot enough. EDGE does not have this problem.

For specific applications where you need chemical resistance, UV stability, or more toughness, please use the table to find the best material for that application.

TYPICAL PROBLEMS WITH PRINTING PETG

When printing with PETG, you may experience the following aggravations and problems:

- Molten plastic wants to stick to the print head

- Difficulties with calibrating the nozzle offset due to molten filament on the print head

- Debris accumulates on the print head during the print

- Debris on the print head falls off mid-print

- Stringing

- Zits/oozing

- Poor infill quality / infill tearing

- Difficult-to-remove support

- Stripping or chewing up the filament in the extruder due to how soft PETG is

PETG PRINTING TIPS & TRICKS

If you're set on using PETG, the following tips and tricks will help your printing go as smoothly as possible.

1) Make sure the material is dry. Desiccant packs are not sufficient to dry out wet material; we recommend some sort of active drying. SEE HERE for more information.

2) Thoroughly clean the print head before starting your print. Please follow this procedure:

1. Heat the print head to 275C

2. Once up to temperature **power the printer off**.

3. Using a brass bristle brush, thoroughly scrub the nozzle, the underside of the print head, and the sides of the hot block (if needed) to remove molten filament, large chunks of debris, and carbon buildup.

4. Power the printer back on as soon as you're done.

3) Inspect your filament feed system.

- Baseline your extruder's filament monitoring section and the filament feed section. Adjust as needed and replace worn parts as needed. See [HERE](#), [HERE](#), and [HERE](#) for more information.

- Inspect/replace the PTC fittings on both ends of the bowden tube.

- Check your bowden tube and replace if there is excessive play in the fittings. See [HERE](#) for more info.

4) Inspect your print head.

- If your ANVIL tube is old, consider replacing it with a new one.

- Make sure your cold side (30mm) fan is free of debris, spins at the correct RPM, and the duct is fully seated on the cold side.

- Make sure the hot and cold side screws that clamp the tube are fully tightened and the tube is firmly clamped, for proper heat transfer.

- For more information see [HERE](#).

5) Slow your prints down. Fusion3's default PETG profiles are set up for 100mm/s print speed, which requires a high print head temperature (~300C). These speeds and temperatures may not be appropriate for all situations. Make the following changes:

- Reduce print head to 260C

- Reduce print speed to 40mm/s

- Reduce retraction to 3mm

6) During a print:

- Make sure the "ram purge" done during the nozzle offset calibration drops off the head. If it doesn't, open the door and remove it manually.

- If you see molten filament building up on the underside of the print head, pause the print and remove it.

- If molten filament drops off the head and causes a blob in the print, pause the print and remove it.
  - Sometimes you can just pluck it out with your fingers or pliers.
  - If it's stuck/embedded in the print, you may need to cut it out with a small pair of diagonal cutters.