Guide: How to Print with PETG

After the novelty of making Benchys and Marvins in PLA has worn off, you may start to consider more practical applications for your 3D Printer. Each material has its own share of pros and cons, where ABS for example is durable, easy to print and can even be vapor smoothed, yet this plastic emits toxic fumes when melted. PETG in comparison is both strong and flexible while also being completely safe to print. It is widely used in the manufacturing industry for functional components, where it can be injection molded, vacuum formed and even 3D printed. If that isn't enough to convince you, it also comes FDA Approved, making it great for anything from food containers to medical devices.

We are going to look at the most important differences to be aware of when moving from PLA to PETG. Small adjustments will make a world of difference, often those which are overlooked or lack general explanation. Once your settings are tweaked and configured correctly, PETG can print just as well or better than the traditional choices.
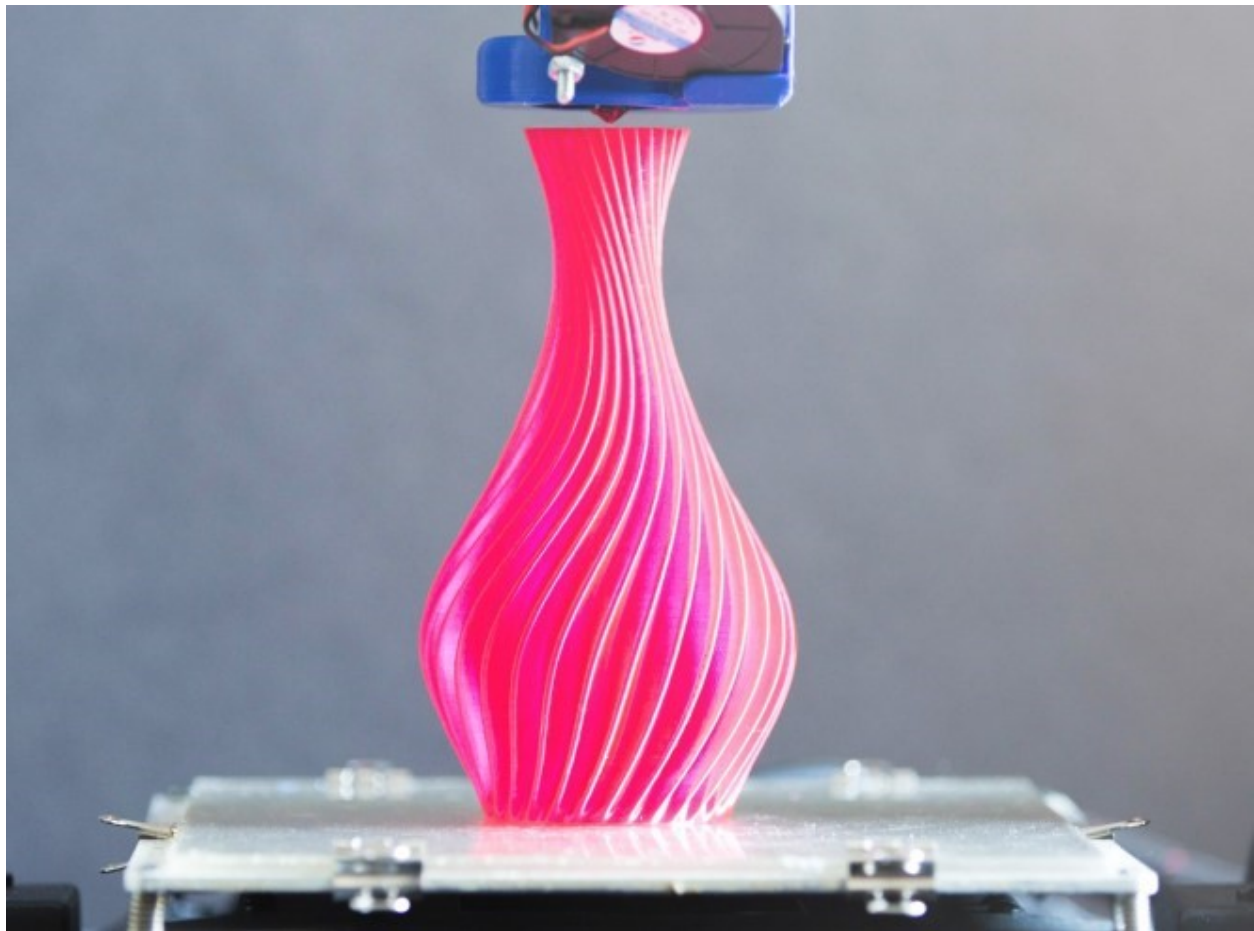

PETG Vase

PETG Brands

I've used several options available on Amazon with mixed results. Although I found Hatchbox to be the easiest to print, the surface quality was somewhat mediocre and often had a rather bland finish. Inland was an improvement but seemed to string and ooze the most, requiring some additional work to remove the excess plastic. My personal preference is the eSUN brand when it comes to PETG, where the Semi Transparent line of filament produces gorgeous results and has the least issues with stringing during print.

For first timers however, I would advise starting with the absolute cheapest (reputable) option available. You may very well go through several rolls before you have the settings tuned as desired. In my case, I wasted an entire spool when creating a set of functional parts, only to later find they were as brittle as potato chips, splitting into pieces under very little pressure.

Bed Leveling

Since PLA is often the first material every new maker uses, most will be familiar with using a sheet of notebook paper to space the nozzle from the bed. PLA essentially needs to be smushed into the build plate, where a very small gap is required for optimal extrusion.

PETG on the other hand works best with a distance between the nozzle and bed, where the extrusion lays down much like squeezing toothpaste. If the nozzle is too close to the bed, it won't extrude properly and may cause the filament feed gear to jam. Spaced too far and the filament won't properly adhere to the build plate. Using the same approach for leveling as PLA, substitute the thin notebook paper with a thicker sheet such as an index card or your ex's wedding invitation.
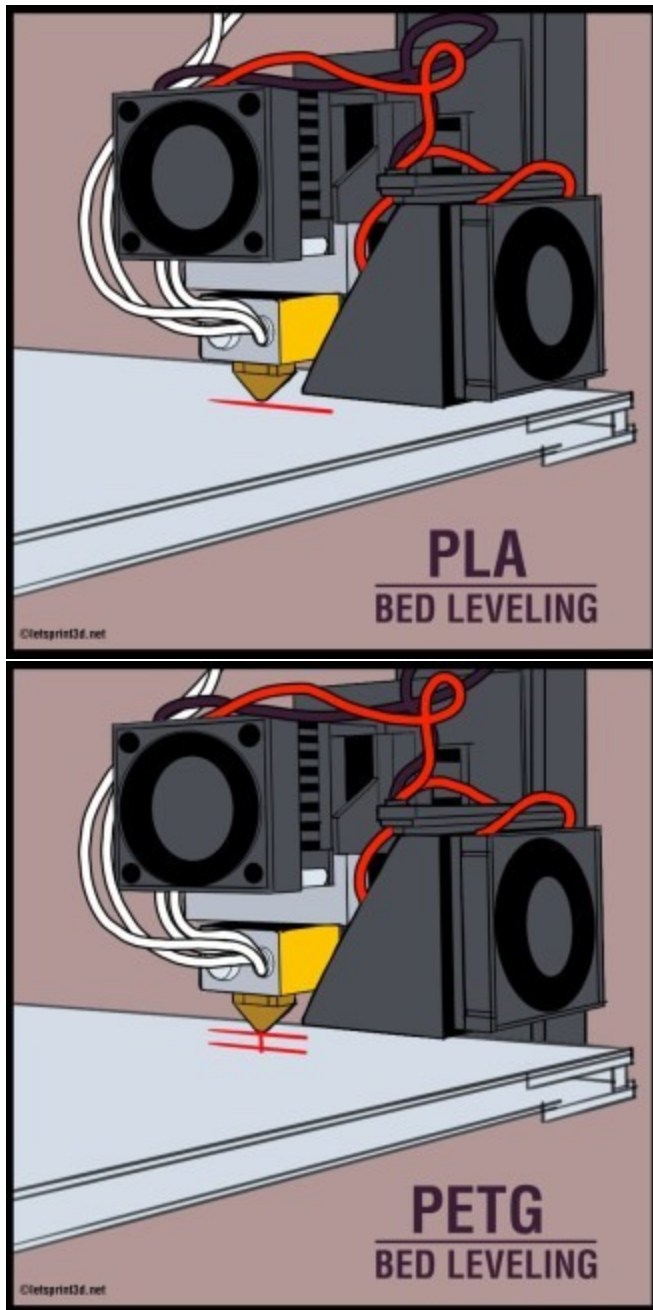
Print Temperatures

PETG has a higher melting point than traditional filaments such as PLA, where it flows best between 240-260C and sticks to a heated bed at 75-95C. Every spool of filament is different to some degree, but it will include the Manufacturer's optimal working temperatures listed on the side. I would suggest printing a Temperature Tower first, where this will give you an idea how it looks, feels and prints at incremental temperatures. If the nozzle is too cold, PETG parts will have weak structure and delaminate with ease. If the nozzle is too hot, it will cause excessive stringing, oozing and blobbing.

Nozzle Temperature: 240C – 260C
Bed Temperature: 75C – 95C

Note: Perform a PID Tune on your extruder when switching to PETG. This will prevent heat fluctuations and keep the nozzle as close to the desired temperature as possible.

As a word of caution, many 3D Printers do have a white PTFE tube liner inside of the nozzle. This plastic is not designed for excessive temperatures, where it will begin to degrade at 245C and becomes toxic at 260C. While you can still print PETG at the lower end of the recommended spectrum, this will greatly accelerate the wear of the PTFE liner. Although cheap and easy to replace, long term printing will be better served by using an aftermarket solution. The Micro Swiss All Metal Hotend and E3D V6 are both popular choices, designed to eliminate the plastic PTFE tubing and facilitate printing with higher temperatures and abrasive materials.
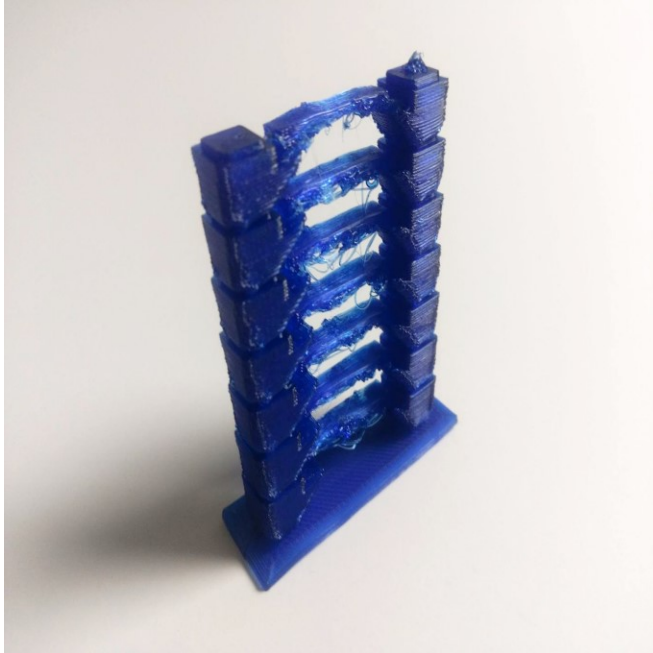
Cooling Fans

Most articles and discussions concerning PETG makes extremely different claims as to what works best in terms of cooling. Among the top search results on Google, one adamantly states a cooling fan is needed, while the next says no cooling fan should be used at all. This vast range of opinions does nothing more than impose even more confusion on which one is correct.

The truth of the matter is PETG will print anywhere from 0-100% fan speed depending on your intended goal. The amount of cooling should be chosen based on the purpose of the part.

Low Fan Speed: The less cooling used, the stronger the part will ultimately become. The molten plastic essentially melts into the previous layers, providing exceptional adhesion to one another. The downside is that this can lower the aesthetic quality by a considerable amount in exchange for stronger parts.

As can be seen in the image, bridges and overhangs will suffer without proper cooling. To address this problem, it is recommended to use the *Bridging Fan Speed Override* setting in Simplify3D or a similar feature within your desired slicer software.

High Fan Speed: The faster the part is cooled, the better it will look when finished. This is often the best choice for vanity prints such as masks, vases and other aesthetically pleasing designs. It will not offer the same level of durability as prints that set at room temperature, however the overall surface finish will be a drastic improvement.

The temperate tower on the left was printed at 100% fan with a 50mm blower fan and the Dii Cooler, where the tower on the right was printed using the low powered stock fan. The higher powered fan causes the plastic to set quickly, which prevents the bridges from sagging and overhangs from warping at the edges.

Print Settings

Printing PETG successfully is all about finding the right balance of settings. While it can be a bit more complex at first, it is a considerably better choice for most applications. There is no getting around the need to experiment, however the following should work as guidelines when building your own PETG configurations.

Print Speed: Use a slow speed for the first layer to ensure proper adhesion to the build plate. This should generally be around 20-25% of your normal print speed. Start with 30 mm/s for your PETG prints as the base speed and increase this gradually until you find the perfect speed for your printer.

Print Speed (Layer 1): 10 mm/s
Print Speed (Layer 2+): 30 mm/s

Retraction: PETG is notorious for creating strings and blobs, where finding the correct retraction values will help to minimize this. Start with these settings and tune as needed, increasing the distance if excessive stringing is present, or decreasing if you experience clogs.

Retraction Distance: 1.0 mm
Retraction Speed: 30 mm/s

Temperature: The heat applied to PETG can cause a wide range of results to both the look and strength of the material. The recommend temperatures below are middle values, where you can tweak these as needed. More heat to the extruder will increase the flow of plastic (and increase stringing), where 245C is a good starting point.

Extruder Temperature: 245C
Bed Temperature: 85C

Fan Speed: The cooling fan speed is described in depth above but make sure the first layer is set to 0% to ensure proper adhesion. For the remaining layers, choose the fan speed based on the purpose of the part. Lower fan speeds for strength and higher fan speeds for

aesthetic items. The "Bridge Fan Speed Override" is essential for good bridges regardless of your other settings.

Fan Speed (Layer 1): 0%
Fan Speed (Layer 2+): See Details Above
Bridging Fan Speed Override: 100%

First Layer Height: 90%
First Layer Width: 100%

Use Skirt: Enabled (Clears the nozzle of any excess residue)
Skirt Outlines: 2