[PETG Filament - Overview, Step-by-Step Settings & Problems Resolved](#)

Posted by Ed Tyson

(Image courtesy of Tom's Filaween review of our Orange PETG at toms3d.org)

In this guide you'll learn why PET-G filament is fast becoming a favorite. We'll explain what it is and what's it's used for.

Then we'll cover the **best practices to printing with it perfectly**, including how to avoid any pitfalls when using it.

Is PLA or ABS not cutting it anymore? Perhaps you need something more durable, or something that can handle higher temperatures than PLA, but is easier to print than ABS?

PETG filament might be your next favorite to print with, we know it's been rapidly gaining in popularity since it was launched. Let's take a look at what is PETG, why you'd use it and how to print with it.

What is PETG Filament?

PETG is a durable copolyester (a combination). The PET stands for [polyethylene terephthalate](#) (think plastic bottles) and the G means it's been glycol modified for extra durability.

And what is PETG Filament used for?

In short this is a really tough material, it's **extremely durable and prints without odour**. Once you've dialed in the correct print settings, it prints nicely too. Users report similar finish quality to PLA.

Here are the main benefits to printing with this material and common PETG filament properties:

- Very durable, it's more flexible than PLA or ABS, but also a little softer. You'd have **a hard job breaking it in half**, so if an 'unbreakable' case or enclosure is what you need, PETG trumps pretty much everything (except, [Nylon 12](#)).
- It has **very low shrinkage**, and therefore no warping. Ideal for printing big stuff.
- PETG is also **very strong, it's not brittle** but can be scratched more easily than ABS which is harder.
- PETG plastic makes a terrible support structure, because it sticks so well. But because it sticks so well, **layer adhesion is fantastic**, so prints come out strong.
- It sticks well to the print bed too, so be careful when you're removing it after printing.
- It has a **great chemical resistance**, along with alkali, acid and water resistance.
- Odourless when printing.

Typically Polyethylene filament is supplied in a range of translucent colours, and prints with a nice glossy finish.

It makes it ideal for printing anything that needs to be shatterproof or translucent. Many are taking the leap from using PLA or ABS to just using PETG.

It's unlikely you'd use standard [PET filament](#) when the more durable glycol modified version is now widely available.

In Tom's petg filament review, he reported the weighted strength test of our PETG to be 'off the scale', meaning in the direction of the layers, he couldn't break it. This isn't to brag (honest!) but should hopefully indicate the high durability of the material.

But what does this mean for practical applications? What objects are ideal for you to be printing with PETG?

Well, while we don't recommend printing everything with it, due to it's flexibility (which isn't always desirable, depending on your application) there are instances where you need a print with great flex, but where printing in a fully flexible material (like TPU, Flexi PLA, or Nylon) wouldn't provide enough support.

This makes it ideal for 'real world' applications and end-use parts.

What's PETG Glass Transition Temperature?

It's 80C - so that's worth taking into consideration when deciding what material to make your next project out of. This is significantly lower than ABS's Tg of 105C, but higher than PLA which is as low as 55C.

Below is one of our customer's applications, which is a bumper for his micro quadcopter. As you can imagine, the bumped needs reasonable stiffness to resist impacts, but yet plenty of durability to absorb the force of any severer crashes.

We think, this is a perfect example of the types of prints you may wish to print with this material. Essentially, it's a great addition to your existing 3D printing filament arsenal.

Great example of the practical use for PETG's durability. Crash bumper by Otto.

Here are a few data stats about PETG. Please note, these stats are from our own PETG and are not representative of the average:

- Density of 1.27g/cc, that's just higher than PLA and about 20% more dense than ABS.
- Rockwell hardness of R 106, which is pretty high for PETG. (Our ABS, which is very hard is rated R 110).
- Our raw pellets for PETG have been GREENGUARD INDOOR AIR QUALITY CERTIFIED®. This is a certification under license from the GREENGUARD Environmental Institute (GEI), who are an independent, non-profit organisation that test low-emitting products and materials for indoor environments.

Now you know why you're likely to go for PET-G filament, vs the more traditional materials. It's nice to print with and produces excellently tough prints that will last.

Let's look at how to get the best results from **this underutilized filament**, so that you can spend the least amount of time setting up, and the more time producing ultra-durable high use prototypes, models or end-use parts.

(another screenshot from [Tom's epic 90 second PETG review](#))

Here's how to print PETG:

As with all 3d printing materials, you need to take note of the specific traits that material adheres to – which issues are caused by what?

This always saves head-scratching time when you run a material through your printer the first time, and the results aren't quite what you expected.

This plastic is just like any other, you just need to adhere to the few best practices when printing and you'll love the results.

Sometimes PET-G can take a little more setting up, [fine tuning those filament settings](#). It's just slightly more particular than something more forgiving, like PLA. That's not to say it's hard to use, just perhaps a little more patience with the setup.

But once you're set up correctly, you'll find printing with PETG a dream. No warp, odorless printing and great layer adhesion are just some of the excellent properties with printing this filament.

Be sure to use a high quality polyethylene filament and it's likely you'll just dial in your PETG temperature settings and you'll be away.

Usually Have an Extra 0.02mm to 0.05mm Gap Printing PETG
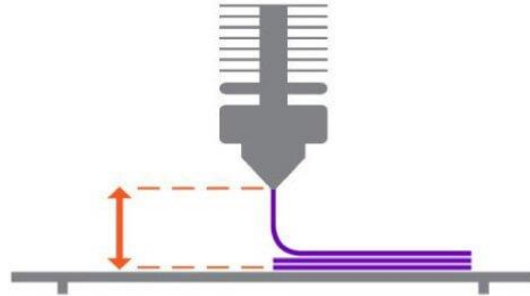
However, as with any filaments there are some pointers to make the new transition easier.

Let's look at how to get setup correctly, issues to look out for and our top PETG printing tips that'll save you time troubleshooting.

PETG Print Settings

1. We recommend printing our PETG settings at roughly 220C-245C depending on your extruder. The PETG bed temperature works best around 70-75C, a few degrees hotter perhaps for those first few layers.
2. What's the best surface to print PETG on? Well in our experience, blue painter's tape works the best. PETG bed adhesion can work great with other materials though, providing you follow the next (very important) step.
3. This material doesn't need to be squeezed onto your heated bed, you want to leave a slightly larger gap on the Z axis to allow more room for the plastic to lay down. If the extruder nozzle is too close to the bed, or previous layer it will skim and create stringing and build-up around your nozzle. We recommend starting off moving your nozzle away from the bed in 0.02mm increments, until there is no skimming when printing.
4. The Fan: essentially if you want the strongest possible print, print with no fan. The higher PETG filament temperature will aid super-strong layer adhesion. The molten PETG will stick to the previous layer ridiculously well. But if you need better detail, and no stringing, you really need to use 100% PETG cooling fan. The rapid cooling after leaving the nozzle will leave your prints detailed, with no stringing or blobbing.

This material doesn't need to be squeezed onto your heated bed, you want to leave a slightly larger gap on the Z axis to allow more room for the plastic to lay down. If the extruder nozzle is too close to the bed, or previous layer it will skim and create stringing and build-up around your nozzle.

We recommend printing **without fan for the first layer or two, and then full fan after that**. If you're still getting trouble with slight stringing and you're feeling brave, taking the guard off your fan can increase airflow further – but that's your choice.

These are some of Joseph Casha of [3D Maker Noob's](#) prints in our Red PETG

This is the sort of thing you may choose different fan settings for different prints - so that you're set up optimally for what you wanted to print. Experiment to get a good idea for how the filament reacts with your printer's fan settings.

4. Print a little slower, we recommend around 55mm/sec or less, more than 60 and it won't lie fast enough.
5. PETG can be sensitive to over extrusion (blobbing etc.) – if you're experience this, just bring in the extrusion setting on the splicer ever-so-slightly each time until it stops.

It's likely you won't have issues with all of these points, but as you can see – just like other 3d printing filaments, each material has its own set of traits to setup for.

Once you know the cause of each issue, and how to fix you'll find the printing consistent time and time over.

Here's a quick rundown of PETG vs ABS:

- PETG is more durable than ABS, but ABS is harder, and more rigid.
- PETG has a lower glass transition temperature, at 80C compared with ABS's 105C
- ABS is approximately 20% less dense than PETG.
- PETG won't warp like ABS might (if printed incorrectly) and is generally odourless.
- PETG is more chemically resistant, and so cannot be acetone smoothed like ABS.

Here's a quick rundown of PETG vs PLA:

- PLA is more brittle than PETG, unless you want to try to [anneal it.](#)
- PLA and PETG have very similar densities.
- PETG will need a heated bed, whereas PLA can be printed cold.
- Layer adhesion with PETG is typically unmatched, leaving very strong and durable prints.
- PLA prints supports easily to remove, whereas these are harder (but not impossible) to remove with PETG.