PETG overview

Dalton Durst edited this page on Nov 27, 2016 · 3 revisions

Random thoughts on PETG...

According to rigid.ink and the #reprap IRC channel...

- PETG is more durable than PLA and is more flexible than PLA or ABS. Indeed, from my experience, it is extremely difficult to snap PETG when bending with the layers.
- It sticks to itself... a lot.
- No odor!
- Harder to print than PLA, but easier than ABS. Still, not a filament for the weak.

What to do with it

This is according to rigid.ink (linked above) and airwolf3d. With more experimentation, I'll be able to say which of these are 100% correct and which need some tweaking.

- PETG sees a lot of buildup on the nozzle which is deposited at random parts of the print. The hotend will skim this deposit and throw off the print. (How to fix this will be discussed below)
- Experiment with temperatures for your machine. Between 235 and 260C may work well. Keep the heated bed at 70C (80C for Printbite)
- Use 100% fan if you want good prints. Less fan will give you a stronger print but there will be a lot of stringing.
- Never go above 30mm/s retraction speed.
- Don't print more than around 55mm/s, and absolutely not more than 60. PETG simply can't be pushed through the hotend fast enough.

Preventing Blobs

My experience with this filament echoes many others': it makes a lot of blobs and strings. There are a couple of ways to fix this, you should do both:

- Level your bed

This goes for any filament, but it's really important for PETG. Consider using a feeler gauge. You can find one in [Essential Tools](#)

- Set your Z offset

The Z offset raises (or lowers) the print head relative to the bed before beginning the printing process. You use it to ensure that a height of 0.25mm in your file is *actually* 0.25mm in physical space. The point of this process is to keep the nozzle from skimming the part during printing. PETG does not like to be smashed (unlike PLA) and will instead build up on the nozzle.

In Slic3r, Z offset is located in Printer Settings -> General -> Size and Coordinates. Increment it by 0.02 until you get results that you're happy with. Cura does not have this option. So, if you're using Cura, read on.

Z offset basically just moves the bed a little bit and then sets the new location as Z=0. So we'll do this manually using the start.gcode on our printer.

In Cura, select the Start/End-GCode tab and ensure that `start.gcode` is selected in the list that appears near the top. The text that you see below that begins with `;Sliced at: {day} {date} {time}` is the GCODE that will be added to the beginning of any file that you slice.

Locate the line, `G28 Z0` . After that line, add the following two lines:

```
G1 Z0.01 F{travel_speed} ;Our Z offset
G92 Z0 ;set this location as zero
```

Then try to print. You'll need to revisit the `z` value, increment it by 0.01 until you get an acceptably small number of blobs in your print without terrible adhesion.

- Turn down the extrusion

Calibrating the amount of extrusion is important for every filament, but it is compulsive for PETG. See [Extrusion calibration](#) for more.