

PETG

PRINTING GUIDE

VERSION: 1.1 - 2020

PREPARED BY:
STANDARD PRINT CO.
www.standardprintco.com
info@standardprintco.com
Suite 5 / 1-5 Woodburn St
Redfern, 2016, NSW



The Essentials - Checklist

Tip 1: Increased Z Offset



+ 0.02mm - 0.06mm



Allow for an extra 0.02mm to 0.06 mm offset when printing in PETG. Unlike PLA and ABS, PETG doesn't perform well when it is squeezed onto the print bed. Failure to do so often causes the nozzle to skim over the material that has just been printed. This material accumulates before being deposited on the print as light strings or globs that solidify and increase the chance of your nozzle colliding with the print. Small holes can also form on the skin of your print where material has been picked up by the nozzle.

Tip 2: Proper Temperature Control



Nozzle: 220°C - 240°C

Bed: 70°C - 80°C



PETG is prone to stringing and oozing. When printing PETG start with the lowest suggested temperature recommended by your filament manufacturer. For Standard Print Co. PETG we recommend starting at 220°C and increasing the print temperature only if extrusion problems become evident. A bed temp of between 70°C - 80°C is recommended. Print the first two layers with the bed at 80°C if possible. Should adhesion problems occur, try using blue painter's tape or hairspray.

Tip 3: Proper Fan Control



0 - 100% Fanspeed



Rapid cooling of the molten filament by increasing fan speed will ensure a clean, crisp print. A fan speed of 100% can be used to ensure the best surface resolution possible when printing PETG. If a strong print is what you are after, disabling your fan and setting the fan speed to 0% will ensure your print has great interlayer adhesion. Be careful though as this may come at the cost of surface resolution.

Tip 4: Retraction



Retraction Speed: 30-80mm/s

Retraction Distance: Bowden: 2 - 5mm

Direct Drive: 1 - 2mm



Increasing the retraction speed and distance will decrease that amount of material that oozes out of the nozzle between print moves. Turning on retraction settings in your slicer will tell your printer to actively rewind some filament at the end of a print move, causing a negative pressure to build up in the nozzle. We recommend starting with a retraction speed of between 30-80mm/s and a retraction distance of Between 2-5 mm for a Bowden setup or 1-2mm for a direct drive printer.

Tip 5: Print Slower



30-80mm/s Print Speed



If you are having problems with interlayer adhesion or print quality, try printing a little slower. We've found that printing at 55mm/s or slower yields great results. Printing at speeds greater than 60mm/s and your printer may struggle to lay filament down fast enough. Layers may have problems sticking to one another and in more extreme cases, causing blobs, zits, stringing and holes in the print..

Tip 6: Flow Control



Decrease Flow in 5% Increments



PETG is sensitive to overextrusion. If you've tried all the steps outlined above and you're getting excessive stringing/blobing, the problem may be linked to overextrusion. Simply reduce the flow of filament in 5% increments to check that the filament is extruding correctly and not overextruding.

Tip 7: Unload Filament



Unload Filament When Not In Use



Whenever possible, when not printing with PETG (and all other filaments for that matter), unload the filament from the printer and store it in an airtight container in a cool dry place to keep it dry. Ensure the desiccant that comes with the filament is also stored in the same container to keep the filament in optimal printing condition.

Tip 8: Keep Filament Dry



Keep Filament Dry



PETG is prone to absorbing moisture from the air when left out (being hygroscopic) and hydrolyses when wet, resulting in weaker interlayer bonding on the molecular level. Wet filament can be rescued by dehydration in an oven at 60°C for a minimum of 6 hours or by storing the filament in a container with ample amounts of desiccant for a minimum of 24 hours.



Printing Issues?

We've got you covered.

**Check out our full PETG
troubleshooting guide [here](#).**