PETG blobs and PETG stringing

Even though the issues of PETG blobs and PETG stringing look different from the outside, they are caused by the shared problem of over-extrusion. So, if you solve the over-extrusion problem, you would be preventing both blobs and stringing from troubling you again.

As PETG blobs and PETG stringing are often related to the settings of your 3D printer, configuring your printer with the correct values will prevent these issues from happening again. [1) 2) 3)]

Optimizing these settings can be challenging due to both the knowledge and the amount of trial and error needed to get right.

Printer Settings

- Retraction speed
- Retraction distance
- Retraction minimum travel
- Travel speed
- Nozzle temperature
- Print speed

Experimental Features

- Coasting
- Wiping
- Combing

Retraction speed

To correct the retraction speed setting, you can start by increasing the retraction speed by increments of 5 millimeters/second. If adjusting the retraction speed made the situation better, you can use smaller increments for fine-tuning. For a sprite extruder start with 0.6mm and 85mm/s (https://www.crealityexperts.com/creality-sprite-extruder-pro-upgrade-kit-install-guide#:~:text=In%20your%20slicer%20software%2C%20be,and%2085%20mm%2Fs%20speed.)

Retraction distance

To correct the retraction distance setting, start increasing retraction distance by increments of 1 millimeter. Just as you have done with the retraction speed setting, you can fine-tune with smaller increments after seeing improvements.

Retraction minimum travel

To correct the retraction minimum travel setting, start increasing the retraction minimum travel value by increments of 5 millimeters.

Set the retraction minimum travel setting to the highest possible value where you don’t experience blobs and stringing to lower the risk of filament grinding.

Travel speed

To correct the travel speed setting, increase the travel speed value by increments of 5-10 millimeters/second until you find a balanced value where you don’t experience any blobs, stringing, or ghosting.

Travel moves should be as fast as possible, at least 120 mm/s, to avoid oozing.

Nozzle temperature

PETG will typically print at temperatures between 220 and 255 °C. Start with a temperature as low as 220 °C and do some test prints. If you hear a knocking noise during printing, your extruder is skipping, and you should increase the nozzle temperature by 5 °C. Repeat the test until the extruder doesn’t skip.

Prusa suggests a printing temperature of 230 °C for the first layer and a slightly higher temperature of 240 °C for the rest of the build.[4)] Filament company MatterHackers recommends 245 °C throughout.[5)]

Print speed

PETG is very sensitive to print speed. Print too fast, and you’ll have poor layer adhesion, extruder skipping, and low print quality, but print too slow, and you’ll end up with deformed parts, stringing, and oozing.

In general, the material is best printed at a slow speed, ideally around 60 mm/s. This leads to improved bonding and cooling, and consequently better print quality. Some users report good results with faster speeds for the first layers.

To correct the print speed setting, increase the print speed value by increments of 5-10 millimeters/second until you push it to the highest value where you don’t experience any problems.

Coasting

When enabled, the coasting feature replaces the last few millimeters of extrusion with a travel move

where the printer stops extruding.

You can configure the coasting volume by increments of 0.01 cubic millimeters until you get the desired results.

Wiping

When enabled, the wiping feature causes the nozzle to move over the outer layer of the model to wipe any remaining filament off.

Wiping feature also has a parameter you can adjust, called wipe distance. You can configure this value by increments of 0.01 millimeters until you get the desired results.

Combing

Combing –> Travel Path Adjustment

You can find a feature called combing in some slicers and enabling it causes the travel paths to be re-calculated in a way where the extruder always stays over the print, even if it means that the travel distances will be greater.

Keeping the travel paths on the boundaries of the print reduces the need for retraction and can help prevent the filament from stringing.

Z-Hop

Vertical lift (z-hop) is a feature that causes the nozzle to move up by a slight margin after a retraction.

When enabled, this feature can help prevent PETG from blobbing as the extruder won't be able to leave leftover filament on the print due to the distance created.

There are a lot of mixed opinions about this setting in the 3D printing community, as some people have reported that it actually made the stringing and the blobbing issues even worse.

Experimenting with this feature if nothing else has solved your problems with PETG blobs or PETG stringing.

1) https://www.3dprintbeast.com/how-to-prevent-petg-blobs-and-stringing/
2) https://all3dp.com/2/petg-print-settings-how-to-find-the-best-settings-for-petg/
3) https://www.wevolver.com/article/petg-print-settings-adjusting-temperature-speed-retraction-to-improve-printing
4) https://help.prusa3d.com/en/article/petg_2059
5) https://www.matterhackers.com/store/l/petg-black-high-strength-filament-1.75mm/sk/M3MY2VQG