Glass Fiber Filament Usage Guide

1. Filament Overview

Glass fiber filaments are composite filaments reinforced with glass fiber. Common types include ABS-GF, ASA-GF, PA6-GF, and PPA-GF.

ABS-GF and ASA-GF are glass fiber reinforced versions of ABS and ASA, respectively. ABS-GF retains the excellent mechanical properties of ABS, while ASA-GF retains the outstanding weather resistance of ASA. Both also provide greater rigidity and significantly reduced warping. ABS-GF is ideal for printing structurally stable parts, while ASA-GF is well suited for outdoor applications.

PA6-GF combines PA6 with glass fiber reinforcement, preserving the material's excellent dimensional accuracy, wear resistance, and hardness while increasing rigidity and reducing warping for easier printing. It is ideal for printing mechanical components subject to friction and wear.

PPA-GF is reinforced with glass fiber based on PPA, a material known for its exceptional heat resistance. Parts printed with PPA-GF exhibit excellent ductility and impact resistance, along with outstanding dimensional stability, electrical stability, and aging resistance in high-temperature, high-humidity environments.

2. Hardware Compatibility

Because glass fibers are relatively large and abrasive, they can wear against the inner walls of the nozzle. Therefore, these filaments are compatible only with hardened steel nozzles of 0.4mm or larger, and a 0.6mm hardened steel nozzle is recommended for optimal print results.

Filament	Nozzle Compatibility	Build Plate Compatibility	Accessory Compatibility (IFS)
ASA-GF、ABS-GF、PA6-GF、PPA-GF ASA-GF, ABS-GF, PA6-GF, PPA-GF	Only compatible with hardened steel nozzles ≥ 0.4mm; 0.6mm hardened steel nozzles recommended	Glue required; not compatible with Cool Plate	Not compatible with IFS

3. Preparation Before Printing

Health Notice: ABS-GF and ASA-GF may release harmful fumes during printing. Do not print in an enclosed room for extended periods, and ensure the printing area is well-ventilated.

3.1 Filament Drying

All glass fiber filaments must be dried before printing to prevent issues such as stringing, bubbles, and surface defects caused by moisture absorption. Recommended drying conditions:

Filament Type	Air Drying Oven	Heated Bed
ASA-GF	80℃ 6h 80°C, 6h	80-100℃ 12h 80-100°C, 12h
ABS-GF	80℃ 6h 80°C, 6h	80-100℃ 12h 80-100°C, 12h
PA6-GF	100℃ 8h 100°C, 8h	/
PPA-GF	100℃ 12h 100°C, 12h	/

Note: When drying on the heated bed, flip the spool every 3 hours and cover the filament with its packaging box or a PC box to ensure even heating.

3.2 Heated Chamber Requirements

To reduce warping, warp-prone filaments should be printed in a heated chamber.

For ABS-GF and ASA-GF, set the chamber temperature to 50-60°C. For PA6-GF, set the chamber temperature to 60-70°C.

Thanks to its specially modified formulation, Flashforge PPA-GF can be printed successfully even without a heated chamber. For more stable printing, setting the chamber temperature to 60°C is recommended.

3.3 Feed Filament from the Top

Glass fiber filaments offer excellent rigidity, but they are also more brittle and prone to breaking. Therefore, common glass fiber filaments are not recommended to be fed through the IFS.

The correct method is to place the filament above the filament inlet and keep the guide tube as straight as possible to minimize bending and prevent the filament from snapping inside the feed path.

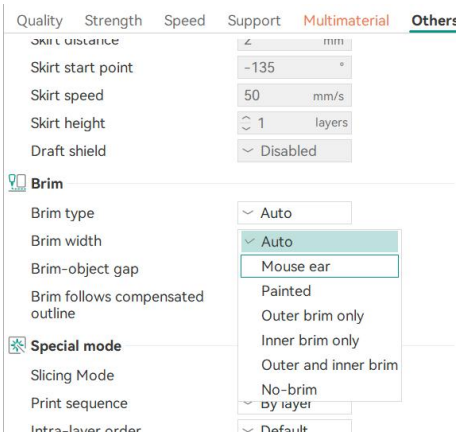
4. Common Printing Issues & Solutions

4.1 Model Warping

Due to their unique physical properties, filaments like ABS-GF, ASA-GF, and PA6-GF have higher shrinkage rates and are more prone to warping.

To prevent warping, try the following:

- Increase bed adhesion by applying a glue stick or bed adhesive to the build plate.
- Use a printer with a heated chamber and set the chamber temperature to 40-60°C.
- Enable a brim based on model geometry (typically choose [Outer brim only]; for sharp-cornered models that warp easily, select [Mouse ear]).



4.2 Filament Not Extruding

Glass fiber reinforced engineering filaments typically require relatively high nozzle temperatures. When switching from a high-temperature filament to one that prints at a lower temperature, residual high-temperature filament left inside the nozzle may not extrude properly. To resolve this, raise the nozzle temperature to the printing temperature of the high-temperature filament, purge the remaining material, then continue purging for a while. Once all residual high-temperature filament has been purged from the nozzle, the lower-temperature filament will extrude normally.

4.3 Oozing

If you encounter a situation similar to what is shown in the image, it indicates the filament has absorbed moisture, causing filament to ooze from the nozzle. Dry the filament thoroughly as

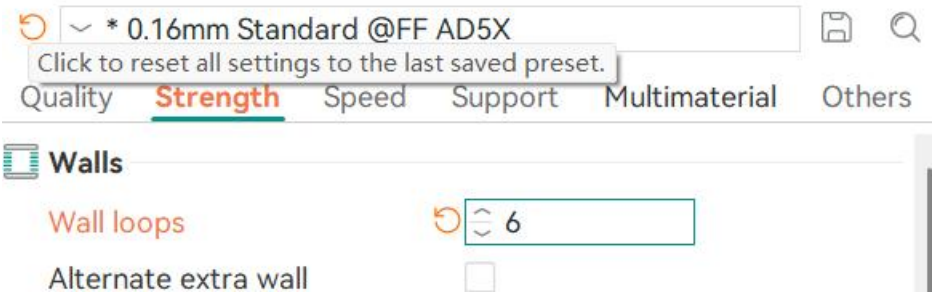
described in Section 3.1.

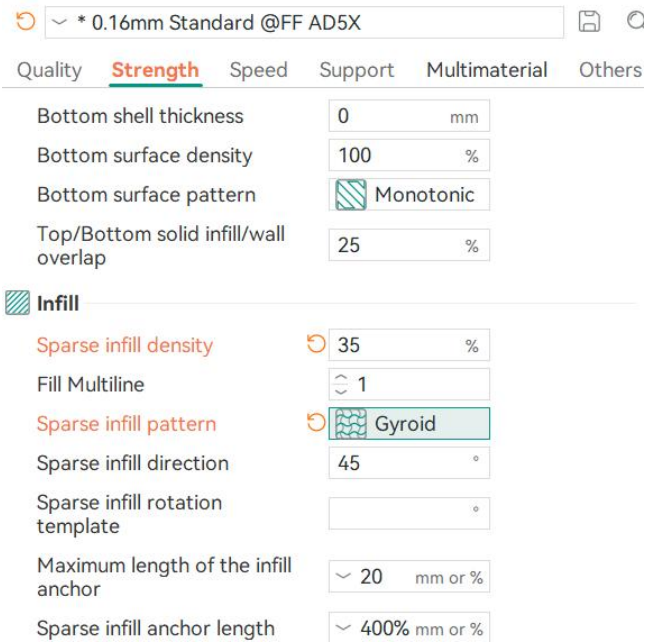
5. Advanced Print Settings

5.1 Improve Model Strength

5.1.1 Increase the Wall Loops and Sparse Infill Density, and Change the Sparse Infill Pattern

You can improve model strength by increasing the wall loops and the sparse infill density, and by selecting a stronger infill pattern. We recommend increasing the wall loops to 3-6 (default: 2), raising the infill density to 20-50% (default: 15%), and changing the infill pattern to "Gyroid" (default: "Grid"). Further increases in wall loops or infill density are not recommended, as excessive model density may increase the risk of warping.





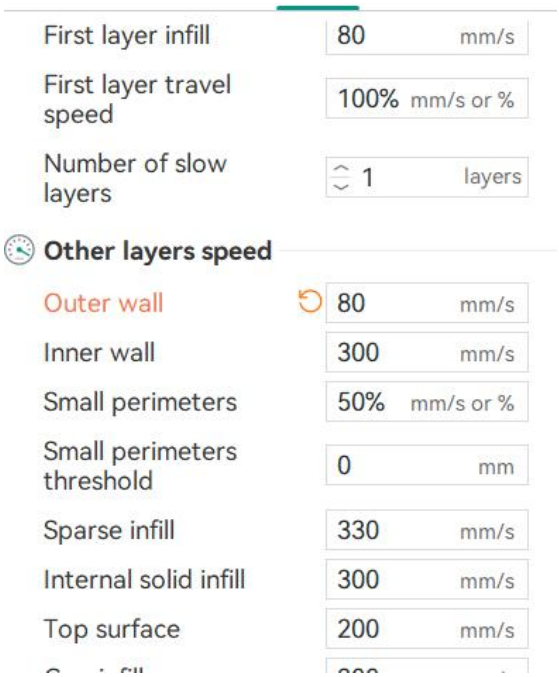
Tip: If warping occurs with high-density prints, refer to Section 4.1 to reduce the risk.

5.1.2 Orient the Model Based on Load Direction

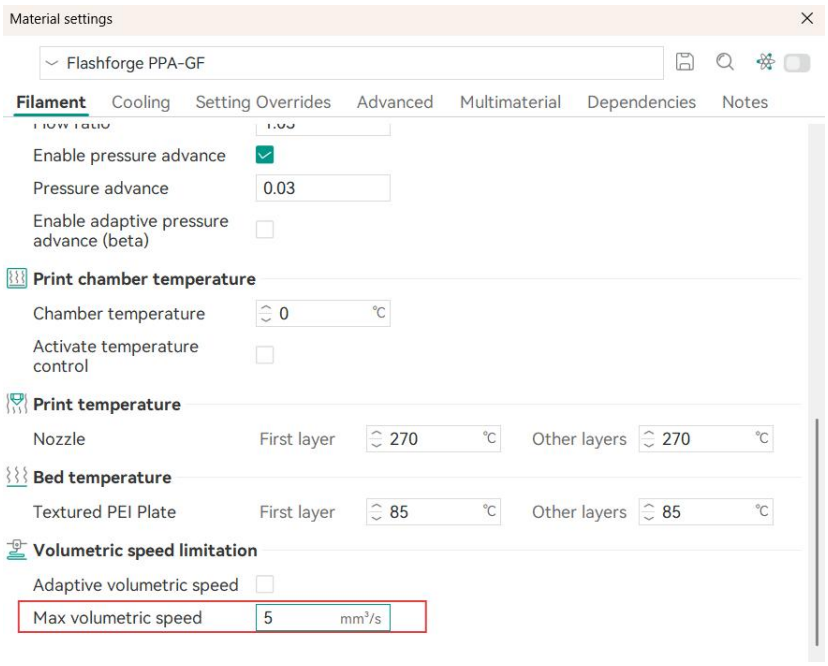
Due to the layer-by-layer nature of 3D printing, parts are inherently weaker along the Z-axis, where interlayer bonding is limited. As a result, fractures are more likely to occur between layers. For load-bearing parts, orient the model so that the Z-axis is not perpendicular to the primary load direction. For detailed guidance, refer to Section 6.2 of the PLA Usage Guide.

5.2 Improve Surface Quality

When printing high-temperature filaments, maintaining a consistent print speed helps improve surface quality. It is recommended to maintain a consistent outer wall speed of no more than 100mm/s.



To enhance overall printing stability and quality, reduce the max volumetric speed for these engineering filaments to 5-8mm³/s.



5.3 Prevent Bridge Sagging

High-temperature filaments require high extrusion temperatures and are typically printed with low cooling fan speeds to improve layer adhesion. As a result, they cool and solidify more slowly during printing. When printing bridges, noticeable sagging may occur. We recommend enabling supports during slicing to prevent bridge sagging. (Note that supports for bridges are not always generated automatically, so you may need to add them manually.)