PETG Usage Guide

1. Filament Overview

PETG is a widely used thermoplastic filament for FDM 3D printing, known for its excellent strength, toughness, and printability. Compared to PLA, PETG is better suited for functional parts that require a certain level of strength and toughness.

2. Hardware Compatibility

As an easy-to-print, entry-level filament, PETG is compatible with all printers. For more information on hardware compatibility, please refer to the table below:

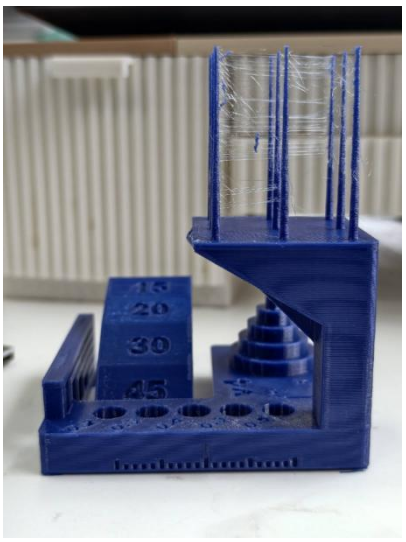
Filament	Nozzle Compatibility	Build Plate Compatibility	Accessory Compatibility (IFS)
PETG Basic/PETG Pro/HS PETG	Compatible with all standard nozzles	Compatible with all build plates	Fully compatible with IFS
PETG-CF	Only compatible with hardened steel nozzles ≥ 0.4mm	Compatible with all build plates	Fully compatible with IFS
PETG Transparent	Compatible with standard nozzles ≥ 0.4mm	Compatible with all build plates	I Fully compatible with IFS

Note:

1. PETG-CF contains carbon fiber particles, which can rapidly wear down nozzles and easily clog nozzles smaller than 0.4mm. Therefore, it should only be used with hardened steel nozzles of 0.4mm or larger.
1. 。
2. PETG Transparent requires a nozzle size of 0.4mm or larger to achieve optimal transparency.

3. Preparation Before Printing

Like PLA, PETG is generally easy to print straight out of the box. However, PETG absorbs moisture from the environment much more easily than PLA. This can lead to stringing and significantly affect print quality, as shown in the figure below.



3.1 Filament Drying

PETG is highly hygroscopic. If the filament has been opened for a long time and has not been stored under dry conditions, it is recommended to dry it thoroughly before printing to avoid issues such as stringing, bubbles, and surface defects. Recommended drying conditions:

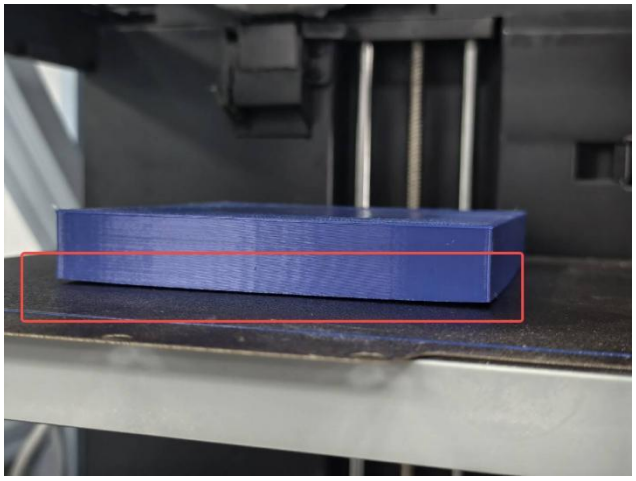
Filament Type	Air Drying Oven	Heated Bed
PETG Basic/PETG Pro/HS PETG/ PETG-CF/PETG Transparent	60-65℃ 8h 60-65°C, 8h	80℃ 12h 80°C, 12h

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.

4. Common Printing Issues & Solutions

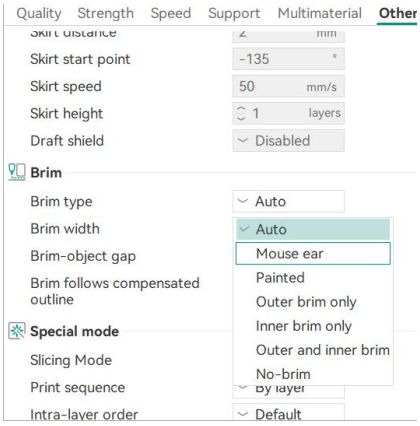
4.1 Model Warping

PETG has a higher shrinkage rate than PLA, making it more prone to warping. When printed layers cool too quickly, internal stress builds up, causing corners to shrink and lift from the build plate, as shown below.

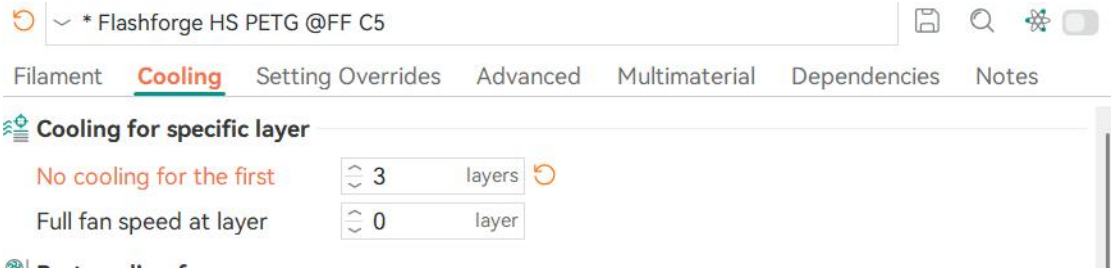


To prevent warping, try the following:

- Increase bed adhesion by applying glue stick or bed adhesive to the build plate.
- Enable a brim based on model geometry (typically choose [Outer brim only]; for sharp-cornered models that warp easily, select [Mouse ear]).



- Control cooling for the first few layers: Set [No cooling for the first] from 1 to 3 to improve the adhesion between the first layer and the bed.



4.2 Stringing

The primary cause of stringing in PETG is moisture absorption. When wet filament passes through the high-temperature nozzle, the moisture rapidly vaporizes, causing the filament melt to expand. This can lead to issues such as oozing, holes, and stringing. If severe stringing occurs, dry the filament according to the instructions in Section 3.1.

If stringing persists after thorough drying, consider increasing the retraction length and retraction speed appropriately.

4.3 Supports Hard To Remove

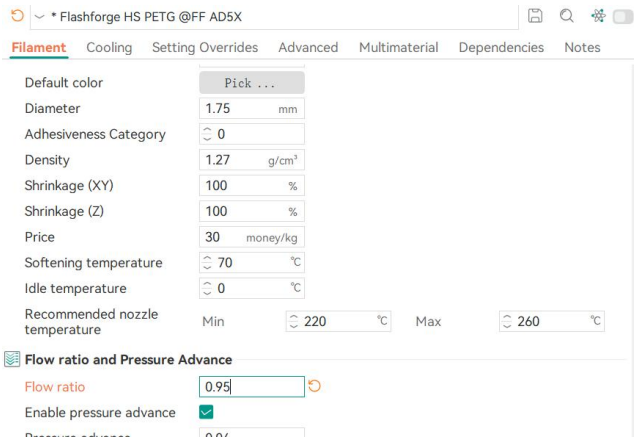
PETG supports tend to bond strongly with the model, making them difficult to remove. This is mainly caused by:

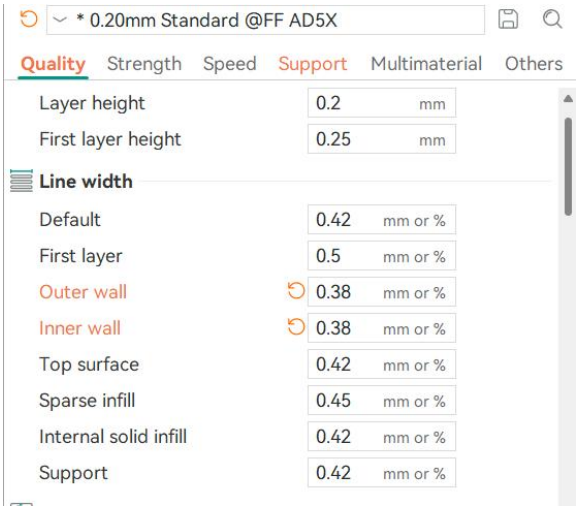
- Moisture in the filament vaporizing and expanding during extrusion, causing supports to adhere to the model's bottom surface
- Excessive flow ratio leading to over-extrusion and fusion between supports and the model
- PETG's inherently strong layer adhesion, which improves strength but also makes supports stick firmly to the model

If supports are hard to remove, optimize in the following order of priority:

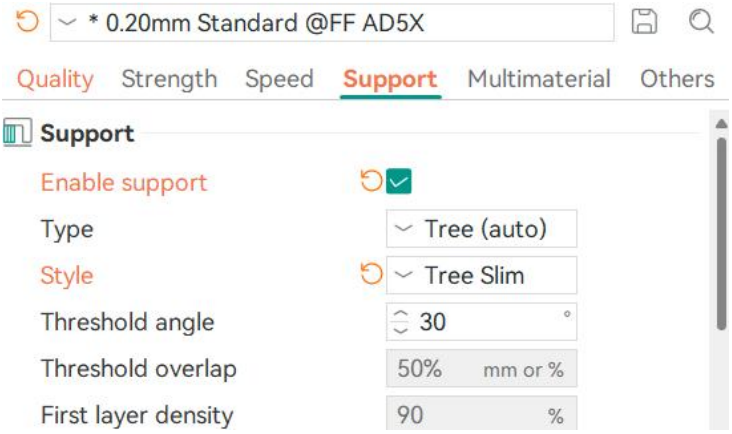
1) Thoroughly dry the filament to ensure stable print quality.
2) Increase the top Z distance to control the gap between the support and the model (0.2-0.3mm recommended to balance support removal and surface quality).

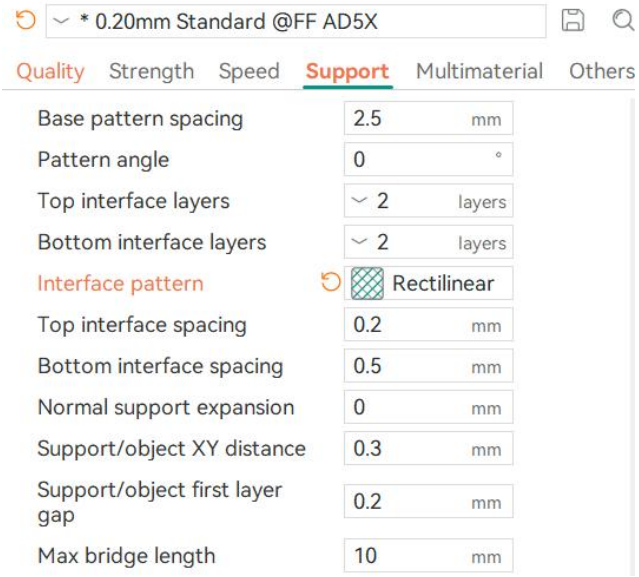
3) Slightly reduce the flow ratio or line width, and recalibrate the flow ratio if necessary.





4) Change the support and support interface types. Use low-adhesion options such as Tree Slim supports or support interface patterns like Rectilinear.



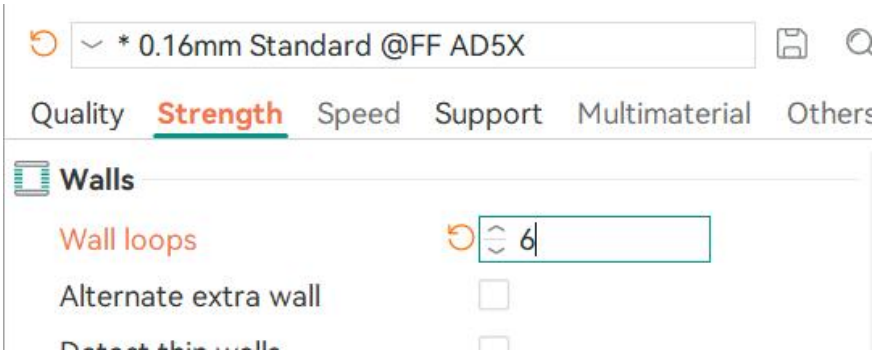


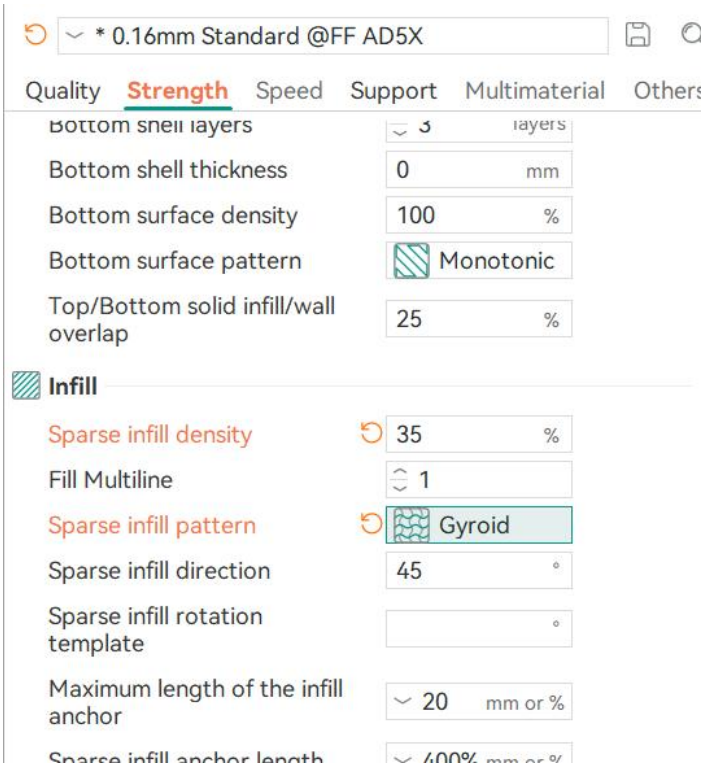
5. Advanced Print Settings

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

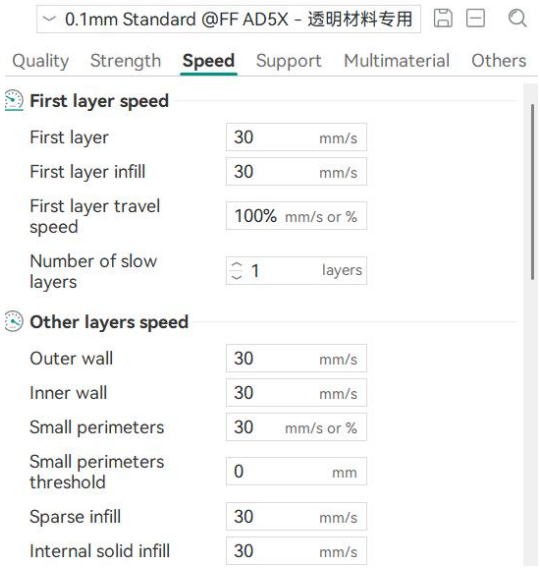
Due to the layer-by-layer nature of 3D printing, parts are inherently weaker along the Z-axis, where interlayer bonding is limited. As a result, fractures are more likely to occur between layers. Although PETG offers better strength than PLA, load-bearing parts should still be oriented so that the Z-axis is not perpendicular to the primary stress direction. For detailed guidance, refer to Section 6.2 of the PLA Usage Guide.

5.2 Print PETG Transparent

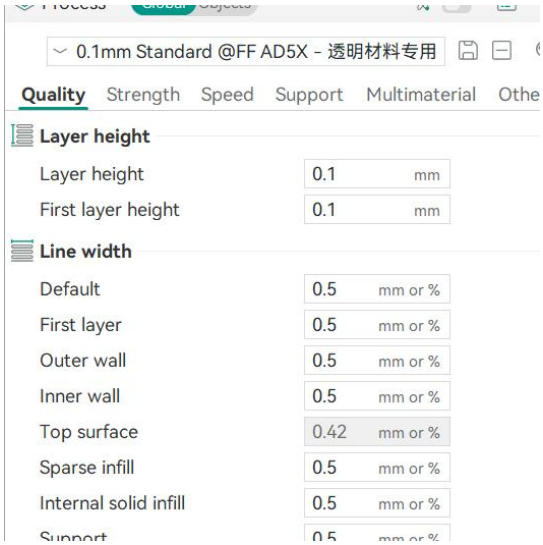
Flashforge offers a specialized PETG Transparent filament designed for clear prints. To maximize transparency, follow the recommendations below to reduce light scattering:

- Dry the filament: This is essential. Moisture in the filament can cause bubbles and voids that disrupt light transmission.
- Optimize print parameters (0.4mm nozzle):

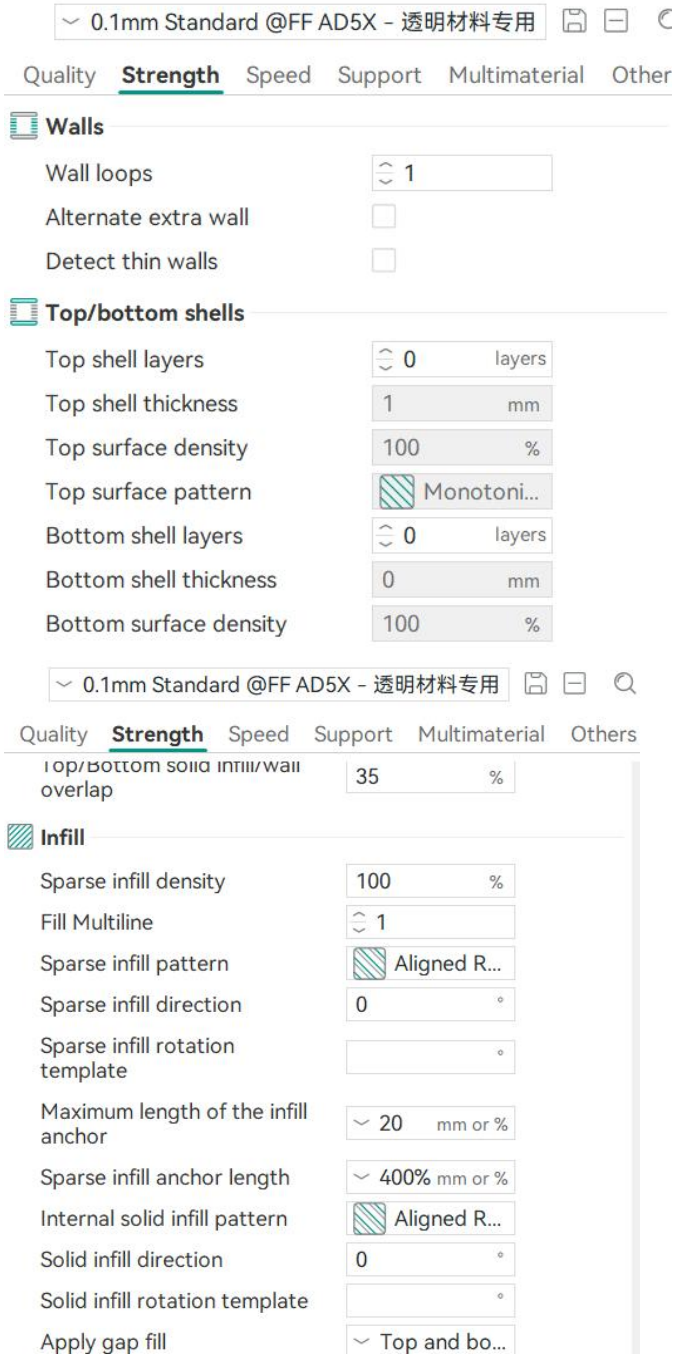
1) Reduce the print speed: When printing at a low speed, extrusion is more stable. It is recommended to reduce the print speed to 30mm/s.



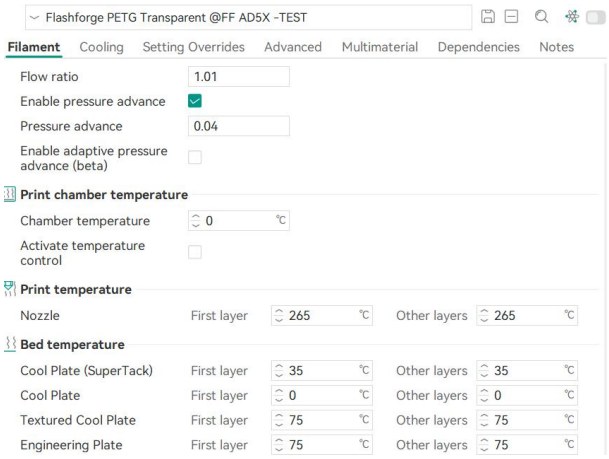
2) Lower the layer height and increase the line width: Thinner layers improve bonding between layers, while wider lines improve bonding between adjacent extrusion lines within the same layer. Both help reduce gaps and minimize light scattering. It is recommended to set the layer height to 0.1mm and all line width parameters to 0.5.



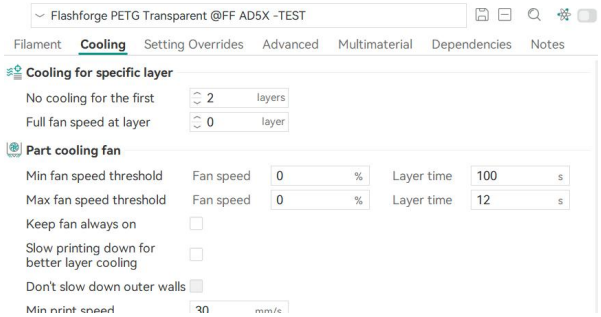
3) Shell and infill settings: To achieve uniform material distribution and reduce light scattering, set [Wall loops] to 1, [Top shell layers] to 0, [Bottom shell layers] to 0, and [Sparse infill density] to 100%. Choose "Aligned Rectilinear" for the infill pattern and set [Solid infill direction] to 0°.



4) Increase the print temperature and flow ratio: Higher temperatures and flow ratios improve bonding between and within layers, reducing voids. Try increasing the nozzle temperature to 265°C and the flow ratio to 1.01.



5) Turn off the cooling fan: Disabling the fan improves layer adhesion. We recommend turning the fan off completely when printing PETG Transparent.



➢ Thickness reference: Transparency decreases as thickness increases. The image below shows transparency performance at 1mm, 3mm, 5mm, and 7mm under recommended settings. Choose the appropriate thickness based on your application.