

Support Filament Usage Guide

1. Introduction

Support filaments are typically used for complex models that require preserving fine overhang details. If your print demands smooth support contact surfaces and uncompromised detail reproduction, understanding how to select and use the right support filament can help you achieve better results with less effort.

With the growing popularity of single-extruder multi-color 3D printers, printing with soluble support filaments presents several challenges. Because the support and model filaments differ, frequent purging is required, which dramatically increases filament consumption and print time while also raising the risk of nozzle clogging. Multi-extruder 3D printers largely eliminate these issues by reducing filament waste, shortening print times, and improving both print quality and stability.

2. Support Filament Selection and Preparation

Support Filament Types

Support Filament Type	Compatible Model Filament	Features
PVA	PLA、TPU PLA, TPU	Highly hygroscopic; must be kept dry at all times; water-soluble; low heat deflection temperature
BVOH	PLA、TPU、PETG PLA, TPU, PETG	Highly hygroscopic; must be kept dry at all times; water-soluble; low heat deflection temperature (slightly higher than PVA)
HIPS	ABS/ASA	Slower moisture absorption; easy to sand and remove; soluble in limonene; good heat resistance
FusRock S-Multi	PETG-CF、PET-CF、PPS-CF、PPA-CF、ABS/ASA-CF PETG-CF, PET-CF, PPS-CF, PPA-CF, ABS/ASA-CF	Highly hygroscopic; must be kept dry at all times; easy to remove without water or solvents; good heat resistance

Multi-extruder 3D printers also make it possible to expand support filament options. For example, using PLA as a support filament for PETG or TPU prints offers significant cost and speed advantages over dedicated PVA. For more information, see our related article.

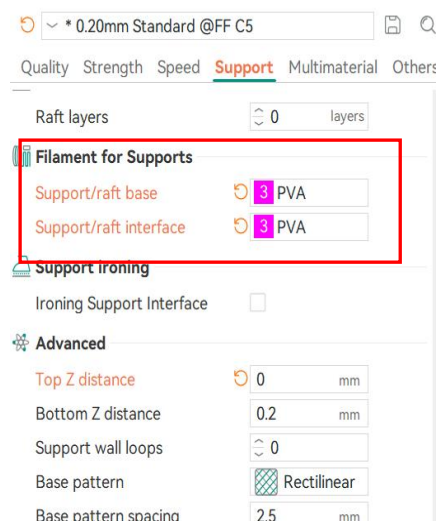
Support filaments require dedicated drying equipment and must be kept dry throughout the printing process. Suitable drying solutions include the Flashforge's VDS or a third-party dry box with similar functionality. Follow the drying presets provided by the drying equipment or the drying parameters listed on the filament spool. Note: Some filament spools are not designed to withstand high-temperature drying. Always check the instructions on the spool before drying.

3. Printer Preparation

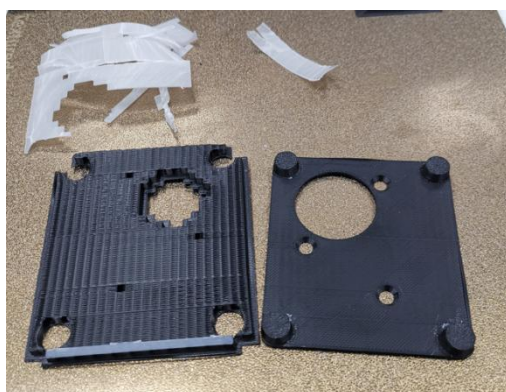
When printing support filaments such as PVA or BVOH, the ambient temperature should generally not exceed 45°C, as higher temperatures can easily cause nozzle clogging. When printing these filaments, ensure the printer is in a well-ventilated environment. For enclosed printers, open the front door and top cover to maintain an appropriate chamber temperature. In addition, water-soluble support filaments may soften and deform over time during extended exposure while serving as support structures, which makes maintaining a dry environment essential. If printing in a humid environment is unavoidable, consider using breakaway support filaments instead. These are less sensitive to moisture, offer stronger mechanical properties, and provide more stable support for taller models.

4. Parameter Settings

Selecting the appropriate support filament will automatically apply recommended print settings in most cases. However, manual adjustments may sometimes be required to achieve optimal results. For example, when using PVA as the support filament, it is recommended to use PVA for both [Support/raft base] and [Support/raft interface]. This ensures superior interlayer bonding and prevents the model from collapsing due to poor adhesion between the support filament and the model filament.

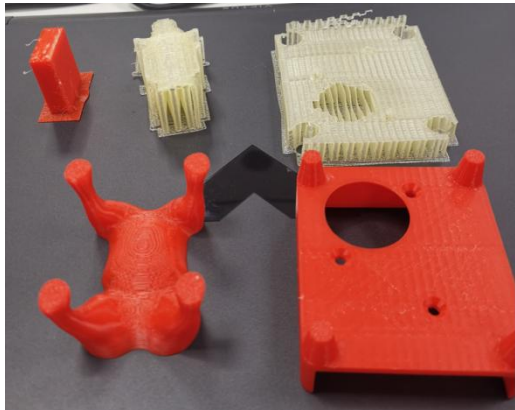


5. Support Removal



Breakaway supports can be removed directly from the model, either by hand or with the aid of

simple tools.



Water-soluble supports can also be removed manually. Alternatively, they can be dissolved in warm water, provided the water temperature remains below the heat deflection temperature of the model filament to prevent deformation.

6. Troubleshooting

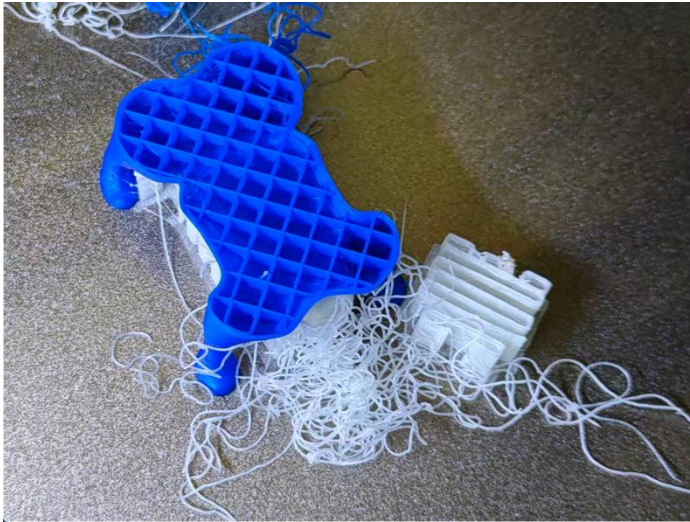
- Nozzle Clogging



Clogs may occur if highly hygroscopic support filaments are not dried properly or kept dry during printing, or if the ambient temperature exceeds the filament's maximum limit (softening point). Residue from previously printed high-temperature filaments left in the nozzle without proper cleaning or cold-pull procedures may also cause clogging.

Prevention: Keep the filament thoroughly dry, maintain an ambient temperature below the filament's heat deflection temperature, clean the nozzle regularly, and consider using a dedicated nozzle for support filaments. If clogging occurs, contact Customer Support for guidance on removing the hotend, inspecting components, and resuming printing.

- Support Collapse



Using insufficiently dried filament or failing to protect it from moisture during printing can reduce layer adhesion within the support structure and increase oozing. This may eventually cause the support structure to crack or be knocked over by the nozzle, leading to collapse.

Prevention: Thoroughly dry the filament, ensure the build plate is clean and free of contaminants that may reduce adhesion, verify that first-layer height calibration is accurate, and confirm that the printing temperature is appropriate for the filament being used.

- Support Contaminating the Model



Insufficiently dried filament can lead to excessive oozing, increasing the likelihood of unwanted filament adhering to the model surface.

Prevention: Keep the filament dry and protected from moisture. If possible, use a support filament that matches the model color, or choose a light-colored support filament to minimize the visual impact of any accidental adhesion.

- Model Not Adhering to Support



This issue is usually caused by undried filament, incorrect support filament selection, or improper temperature settings.

Prevention: Thoroughly dry the filament and protect it from moisture, strictly follow the recommended guidelines when selecting support filaments, and use the suggested temperature settings.