TPU/PEBA Usage Guide

1. Filament Overview

1.1. Filament Introduction

TPU (Thermoplastic Polyurethane) and PEBA (Polyether Block Amide) are flexible filaments known for their high elasticity, wear resistance, and tear resistance. They are widely used in FDM 3D printing for producing elastic parts, shock absorbers, and seals.
Hardness ranking (from hardest to softest):
77D > 70D > 68D > 64D > 55D > 95A > 90A > 85A > 83A > 80A > 75A > 70A

1.2. Filament Properties

Filament	Hardness	Print Difficulty	Typical Applications
TPU-95A	Shore 95A (relatively rigid)	Easy	Phone cases, protective gear, rigid elastic structural parts
TPU-90A	Shore 90A	Easy	Cable protectors, shock-absorbing pads, general elastic parts
TPU-85A	Shore 85A	Medium	Wristbands, insoles, soft seals
TPU-90-75A	Shore 90A → 75A	Easy	Flexible joints, irregular elastic structures
TPU-Aero-90A	Shore 90A	Medium	Drone accessories, elastic components for RC aircraft, impact-resistant parts
TPU-Aero-85A	Shore 85A	Medium	Shock absorbers for RC aircraft, flexible connectors
TPU-64D	Shore 64D (near-rigid)	Easy	Wear-resistant gears, rigid elastic housings, mechanical buffers
PEBA-90A	Shore 90A	Medium	Athletic shoe midsoles, high-end shock absorbers

1.3. Hardware Compatibility

	Printer Compatibility	Nozzle Compatibility	Build Plate Compatibility	Accessory Compatibility
TPU-95A	Adventurer 5M Adventurer 5M Pro AD5X Guider 3 Ultra	Recommended: 0.4mm, 0.6mm, 0.8mm Not supported: 0.25mm	Textured PEI Plate	**Not compatible** with IFS
TPU-64D	Adventurer 5M Adventurer 5M Pro AD5X Guider 3 Ultra	Recommended: 0.4mm, 0.6mm, 0.8mm Not supported: 0.25mm	Textured PEI Plate	
PEBA-90A	Adventurer 5M Adventurer 5M Pro AD5X Guider 3 Ultra	Recommended: 0.4mm, 0.6mm, 0.8mm Not supported: 0.25mm	Textured PEI Plate	

2. Preparation Before Printing

2.1. Filament Drying

Moisture absorption can cause bubbling, stringing, and poor layer adhesion when printing TPU or PEBA. Drying before printing is essential.

Filament	Air Drying Oven	Heated Bed
TPU	70℃，8h 70°C, 8h	90℃，16h 90°C, 16h
PEBA	80℃，8h 80°C, 8h	90℃，16h 90°C, 16h

When drying on the heated bed, flip the spool every 6 hours and cover the filament with its packaging box or a PC box to ensure even heating.

2.2. Nozzle

Clean the nozzle thoroughly and ensure it is free of debris.
Important: Before printing TPU or PEBA, verify nozzle compatibility carefully. It is highly recommended to use a brand-new, dedicated nozzle. Avoid using nozzles that have previously printed carbon fiber or high-temperature engineering filaments, as residues can increase the risk of clogging and print failure. If a specific nozzle must be used, perform a cold pull 3-5 times before printing.
Cold pull: Heat the nozzle to the printing temperature, manually feed a short length of filament, then let it cool to approximately 90°C (PLA) or 150°C (PETG). Quickly and steadily pull out the filament to remove carbon buildup inside the nozzle.

2.3. Build Plate

1. Clean the build plate to ensure the surface is free of debris or contaminants.
2. Apply glue: We recommend applying official glue to improve bed adhesion and increase print success rates.

2.4. Support Material

1. Use PLA as support: Removing TPU supports from TPU models can be difficult and may deform or damage the model surface. PLA is recommended as a support material because it is easier to remove while maintaining model stability.
2. Support material color: Choose a color close to your main model color to prevent color bleeding or mixing on the model surface.

2.5. Reduce Feeding Resistance

Because TPU and PEBA are flexible, elastic, and slightly tacky, they deform easily, which can cause feeding issues and print failure. Try the following methods to reduce feeding resistance:
Filament path: The shorter, straighter, and smoother the path, the lower the resistance.

- Keep the filament guide tube as short as possible: ≤60cm. Longer tubes are more prone to kinking and resistance.
- Keep it straight: Avoid sharp 90° bends or tight-radius turns.

1. Extruder: Loosen the extruder gear spring slightly to prevent excessive compression, which may deform the filament and block feeding.

3. Printing Precautions

3.1. Monitor Filament Feeding

Monitor filament feeding throughout the print. If filament slipping occurs (the extruder gear leaves visible bite marks), slightly increase the extruder spring tension or reduce the print speed.

3.2. Avoid Frequent Pauses

Flexible filaments are prone to feeding issues and layer separation after pause-and-resume cycles. Ensure a stable power supply and avoid unnecessary interruptions during printing.

3.3. Part Removal

After printing, wait for the build plate to cool down to room temperature before removing the model to avoid burns. Gently flex the plate to release the print. If the model is difficult to remove, spray a little alcohol into the gap between the model and the build plate to reduce adhesion before removal.

4. Post-Processing

4.1. Support Removal & Cleaning

1. PVA supports: Soak in warm water (40-60°C) for 2-4 hours until fully dissolved, then rinse thoroughly with clean water and let air dry.
2. PLA supports: Remove manually with gentle force. If the adhesion is strong, soak the contact area in warm water (40-60°C) and slowly peel away the supports.

4.2. Deburring

1. Carefully remove any remaining support material using a craft knife, or lightly sand the surface with 200-400 grit sandpaper. Then gently heat the surface with a low-temperature heat gun (50-60°C) to soften and smooth out fine burrs, improving the overall finish.

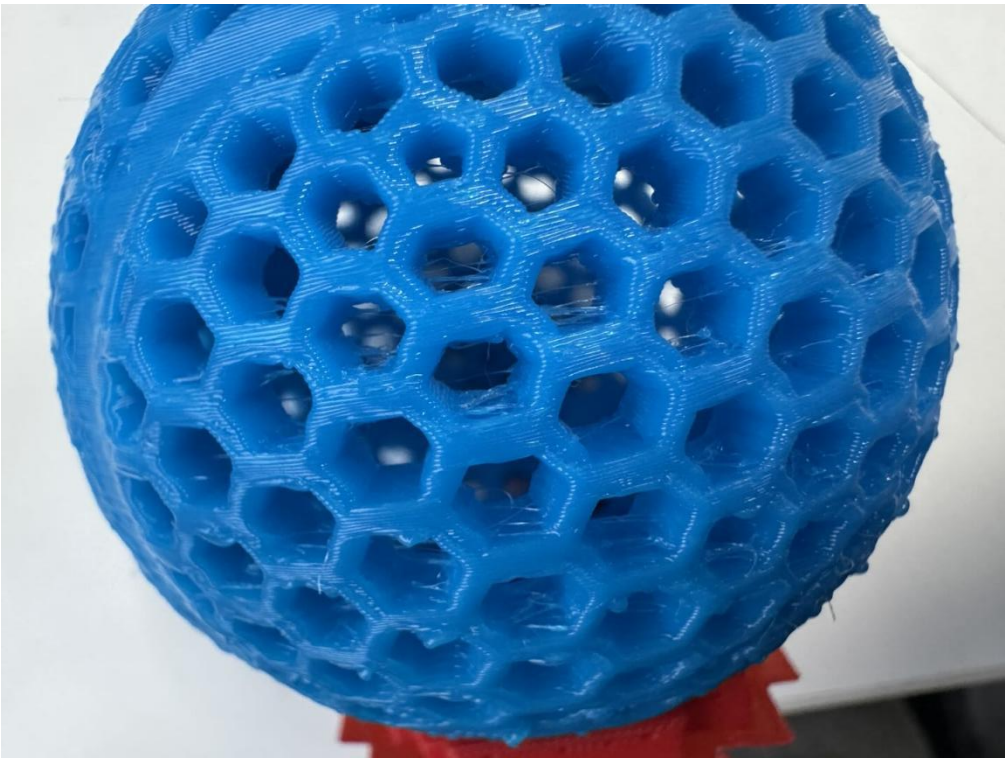
5. Filament Storage After Printing

5.1. Sealed Storage

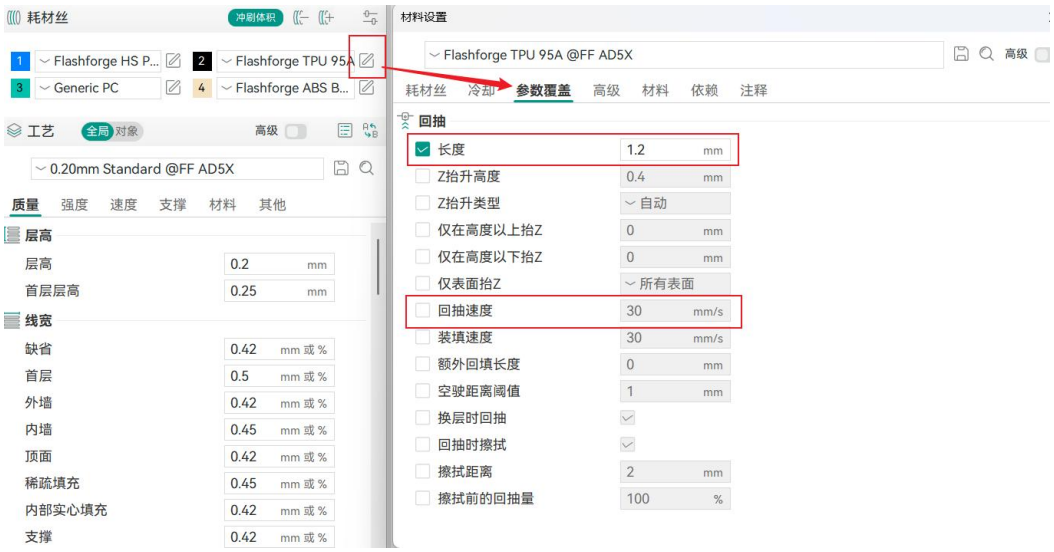
Store the filament in a sealed bag with desiccant at room temperature.
Note: It is recommended to print immediately after drying to prevent the filament from absorbing moisture again.

6. Common Printing Issues & Solutions

6.1. Stringing & Oozing



1. Dry the filament thoroughly and keep it sealed during printing. Incompletely dried filament is highly prone to stringing and oozing.
2. Fine-tune retraction settings. For severe oozing, increase the retraction length slightly (for example, by 0.2mm).



3. Switch to a larger nozzle, such as a 0.6mm nozzle.
4. Post-process the model with a low-temperature heat gun (50-60°C) to remove fine stringing on the surface.

6.2. Filament Slipping or Grinding

1. Excessive feeding resistance
   - ➢ Shorten the filament guide tube to reduce the distance to the extruder.
   - ➢ Reduce spool rotation resistance.
2. Aggressive retraction settings
   - ➢ Reduce the retraction length (for example, by 0.2mm) and lower the retraction speed (for example, to 10mm/s).

6.3. Model Too Stiff or Not Elastic Enough

1. Insufficient cooling: Check cooling fan settings and maximize the fan speed.
2. Infill density too high: Reduce the infill density appropriately.
3. Extruder temperature too high: Lower the nozzle temperature slightly (for

example, by 5°C) and reduce speed settings at the same time (reduce the max volumetric speed by 0.4mm³/s).

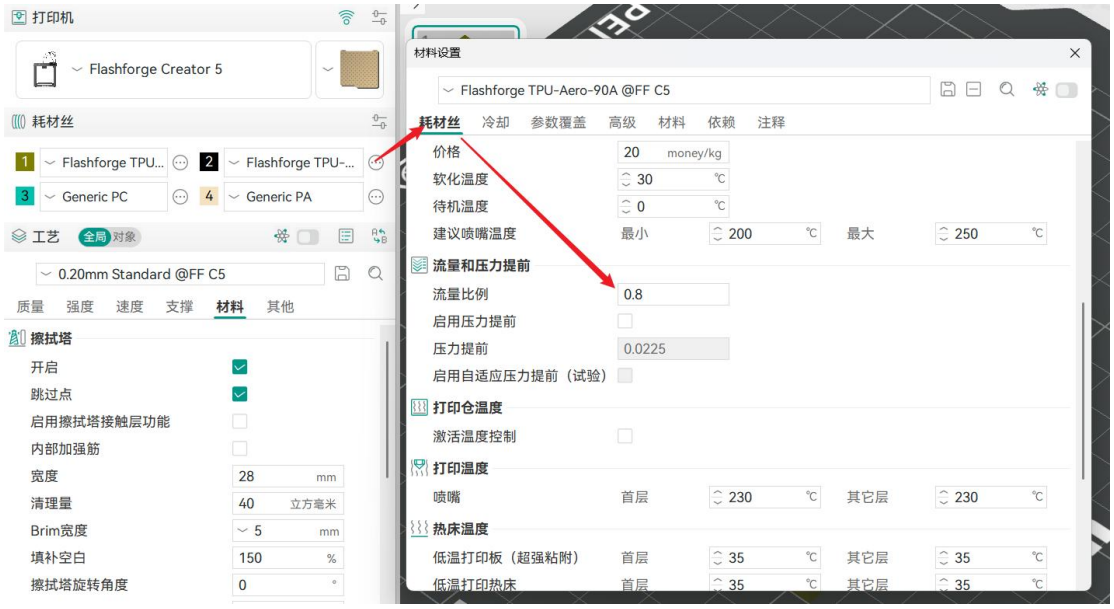
6.4. Under-Extrusion

1. Reduce print speed: Lower print speeds allow the filament to melt more thoroughly, reducing the risk of under-extrusion.
2. Check filament feeding resistance. Refer to Section 2.5 and use an appropriate feeding method.
3. Dry the filament before printing. Moisture in the filament may also cause under-extrusion.
4. Check for residual material in the hotend (e.g., high-temperature filaments). Perform a cold pull before printing to ensure the hotend is cleared of any residue.

7. Advanced Tuning for TPU-Aero

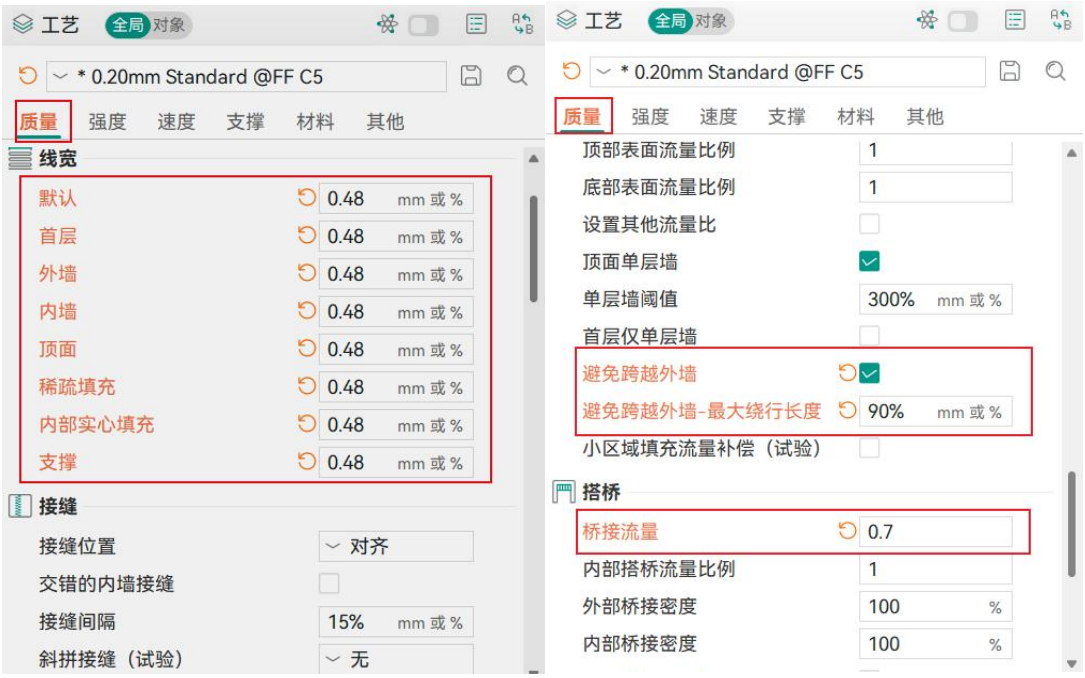
7.1. Adjust Flow Ratio

Foaming performance can be controlled by adjusting the flow ratio. A recommended range is 0.6-0.8, but the optimal value should be fine-tuned according to the specific application and machine setup.



7.2. Adjust Quality Parameters

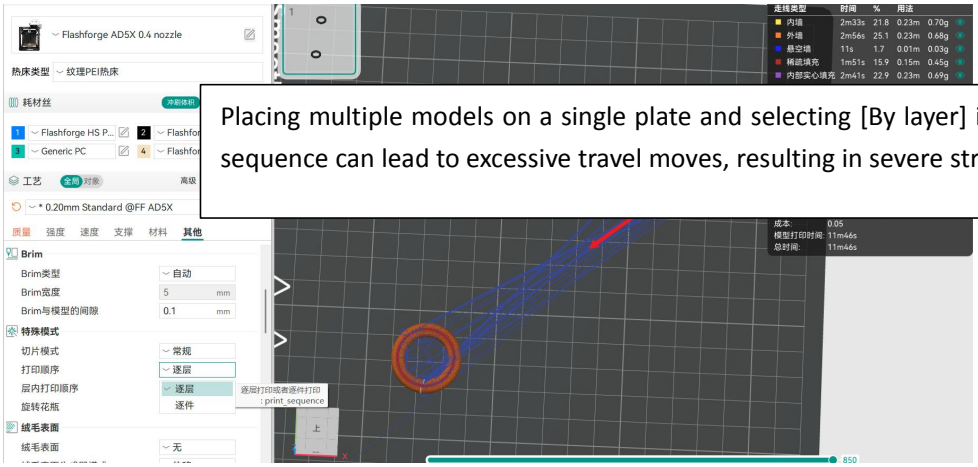
The parameters shown below can be adjusted to improve overall print quality.



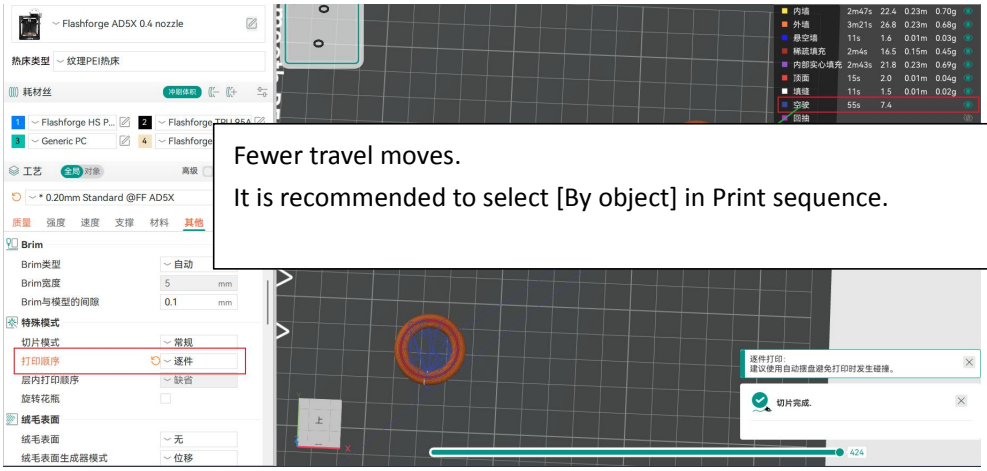
7.3. Adjust Strength Parameters

7.4. Reduce Travel Moves

Enable [By object] in the Print sequence setting to minimize unnecessary travel moves.



Placing multiple models on a single plate and selecting [By layer] in Print sequence can lead to excessive travel moves, resulting in severe stringing.



Fewer travel moves.

It is recommended to select [By object] in Print sequence.