Flashforge PLA Usage Guide

1. Filament Overview

1.1 Filament Introduction

PLA is an environmentally friendly 3D printing filament derived from renewable plant sources such as corn and sugarcane. It is non-toxic, odorless during printing, easy to print, and offers decent strength, though it has low heat resistance. PLA is widely used for models, prototypes, and educational applications.

1.2 Filament Types

In addition to standard PLA variants (PLA Basic, PLA Pro, PLA Lite, PLA Matte, PLA Silk, and HS PLA), Flashforge also offers the following specialty PLA filaments:

PLA-CF: A composite reinforced with carbon fiber, enhancing its mechanical properties and heat resistance

PLA Marble: An aesthetic filament containing large polymer particles in specific colors, creating a finish that mimics natural marble

PLA Wood: An aesthetic filament containing natural wood powder, producing a finish resembling wood grain

PLA Galaxy: An aesthetic filament containing ultra-fine powder, resulting in a dense, speckled, sparkling effect

PLA Sparkle: An aesthetic filament containing flake-shaped glitter, creating a sparse, evenly distributed sparkling effect

PLA Aurora: An aesthetic filament containing pearlescent powder, producing a shimmering, pearl-like effect

PLA Crystal: A unique Flashforge aesthetic filament that uses rapid cooling and ultra-fine powder to achieve enhanced transparency and a crystal-like appearance

PLA Metal: An aesthetic filament containing ultra-fine metal powder, creating a finish that simulates metallic textures

PLA Color Change: A thermochromic filament containing ultra-fine heat-sensitive materials, allowing for reversible color changes in response to temperature variations

PLA Luminous: A light-storing filament that contains phosphorescent powder and emits a faint glow after exposure to UV light

2. Hardware Compatibility

PLA is an easy-to-print, beginner-friendly filament. PLA series is compatible with all supported Flashforge 3D printers. For more compatibility information, refer to the table below:

Filament	Nozzle Compatibility	BuildPlate Compatibility	Component Compatibility (IFS)
PLA Basic/Matte/Lite/Metal/Color Change	Compatible with all standard nozzles	All build plates compatible	Fully compatible with IFS
PLA-CF	Only compatible with hardened steel nozzles ⩾ 0.4mm	All build plates compatible	Fully compatible with IFS
PLA Silk	0.4mm nozzles recommended only	Low-temperature build plates not recommended	Fully compatible with IFS
PLA Galaxy/Crystal/Aurora/Luminous	Nozzles ⩽ 0.25mm not recommended	All build plates compatible	Fully compatible with IFS
PLA Marble/Wood/Sparkle	Only compatible with standard nozzles ⩾ 0.4mm; hardened steel nozzles recommended	All build plates compatible	Fully compatible with IFS

Note:

1. Due to surface gloss characteristics, PLA Silk is recommended for use with a 0.4mm nozzle only. Using 0.25mm or 0.6mm nozzles will not properly achieve the silk-like sheen. Additionally, PLA Silk has weaker interlayer adhesion. When printed on low-temperature build plates with excessive bed adhesion, the base may separate from the model during removal.
2. PLA-CF contains carbon fiber particles, which can cause rapid wear on standard stainless steel nozzles. The particles are also incompatible with nozzles smaller than 0.4mm in diameter.
3. PLA Galaxy/Crystal/Aurora/Luminous contain fine particles. Long-term printing with these filaments may occasionally lead to clogging in 0.25mm nozzles; therefore, 0.25mm nozzles are not recommended.
4. PLA Marble/Wood/Sparkle contain large particles (flakes, fibers, etc.), which are very likely to clog 0.25mm nozzles. Only 0.4mm or larger nozzles are recommended. For PLA Marble and PLA Wood, which prioritize overall texture over fine detail, a 0.6mm hardened steel nozzle is recommended to improve printing efficiency, minimize layer lines, and prevent clogs.

3. Preparation Before Printing

PLA Basic is one of the easiest filaments to print and can typically be used straight out of the package without special adjustments. When removing the filament end secured in the spool holes, avoid loosening it to prevent the filament from unraveling. Some specialty PLA filaments require additional attention due to their unique composition. To further ensure print quality, please follow the preparation recommendations below.

3.1 Filament Drying

PLA has relatively low hygroscopicity and can be stored normally at 50%-60% ambient humidity without frequent drying, unlike PETG or nylon. However, due to their specific compositions, the following filaments are recommended to be dried before printing to prevent issues like stringing or voids caused by moisture absorption.

Filament Type	Air Drying Oven	Heated Bed
PLA Silk/Wood/Crystal /PLA-CF	50℃， 6h	70℃， 10h

Note: When using a heated bed for drying, please flip the filament over every 3 hours and cover it with the original filament packaging box or a PC enclosure.

4. Printing Precautions

4.1 General Reminders

If you have previously printed with high-temperature filaments such as ABS or PC, or carbon fiber filaments such as PLA-CF or PET-CF, it is recommended to perform a cold pull first. This helps prevent clogs or print failures caused by residual material. Please refer to the detailed cold pull procedure.

Cold Pull Procedure:

① On the printer's main interface (location varies by printer), select the nozzle to enter the temperature setting interface;

② Manually set the temperature to 250°C (increase to 260-270°C if previously printing PET-CF or other high-temperature filaments);

③ Remove the Bowden clip and filament guide tube from the corresponding nozzle;

④ Cut a section of PLA filament (PLA Basic/Matte works) about 20cm long;

⑤ Slowly feed the cut PLA filament into the open filament inlet. The old residual filament will be extruded first. Continue until the new PLA filament comes out;

⑥ Feed a small additional amount to flush, and then stop. Set the nozzle temperature to 100°C and wait for it to cool down;

⑦ Pinch the end of the inserted PLA filament and pull it out of the inlet quickly;

⑧ Check the tip of the pulled PLA filament for any residue from previous filaments (different colors or carbon fiber particles);

⑨ If residue is found, repeat the above steps until the pulled filament tip is clean.。

4.2 Filaments Requiring Special Attention

If using PLA Wood or PLA-CF, please note the following precautions.

4.2.1 PLA Wood

PLA Wood is not compatible with 0.25mm nozzles, as the fine wood powder can cause clogs.

The addition of wood powder produces a mild wood scent during printing. This odor is non-toxic and non-irritating. If discomfort occurs, ensure adequate ventilation.

PLA Wood is prone to moisture absorption, which can lead to stringing (see image below). Please dry the filament before printing to prevent stringing and ensure optimal print quality.

4.2.2 PLA-CF

PLA-CF does not support nozzles smaller than 0.4mm and must be printed with hardened steel nozzles to prevent excessive wear and maintain print quality.

For long-term printing with PLA-CF, it's recommended to regularly perform cold pulls (see Section 4.1) to maintain a clean nozzle.

5. Common Printing Issues

3D printing issues such as poor overhangs or bridging are common across many filament types. This section focuses only on common issues encountered when printing PLA.

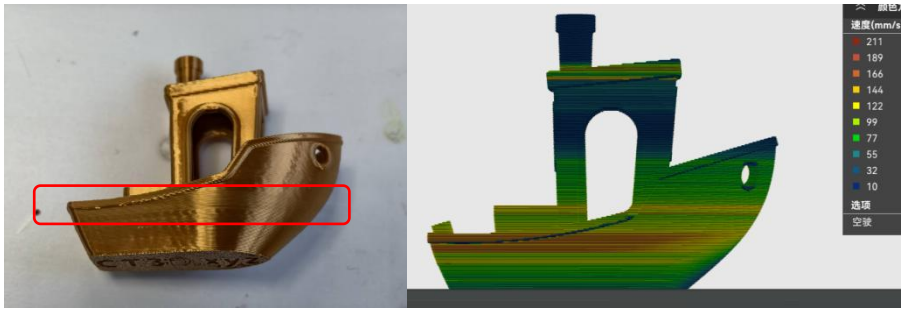
5.1 Heat Creep Causing Extruder Clogs

PLA has low heat resistance, and most PLA variants have a heat deflection temperature below 60°C. In high ambient temperatures or when the extruder is close to a heated bed, internal heat buildup can cause PLA to soften prematurely inside the extruder, leading to jams.

It is recommended to print PLA in a well-ventilated environment with ambient temperatures between 5-30°C to prevent internal heat buildup. For enclosed printers, please open the top cover or door during PLA printing.

5.2 PLA Silk Print Issues

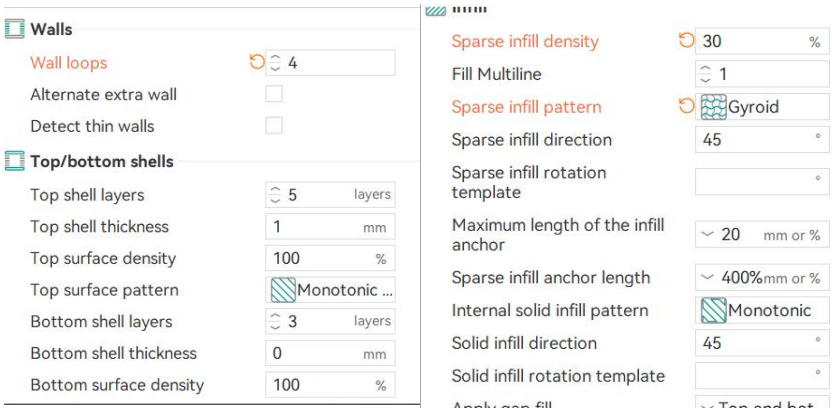
5.2.1 Uneven Gloss



Significant variations in print speed across different areas can cause uneven gloss. For example, the area outlined by a red box in the left image shows reduced gloss corresponding to a zone with excessively high speed in the slicing preview. Refer to Section 6.1 for parameter adjustment guidance.

5.2.2 Insufficient Strength Leading to Cracking

The flake particles in PLA Silk make it more brittle than standard PLA, with weaker interlayer adhesion along the Z-axis, leading to a higher risk of cracking. For non-load-bearing parts, default settings are usually fine. For load-bearing applications, increase wall loops, adjust infill pattern, and increase infill density to improve strength.

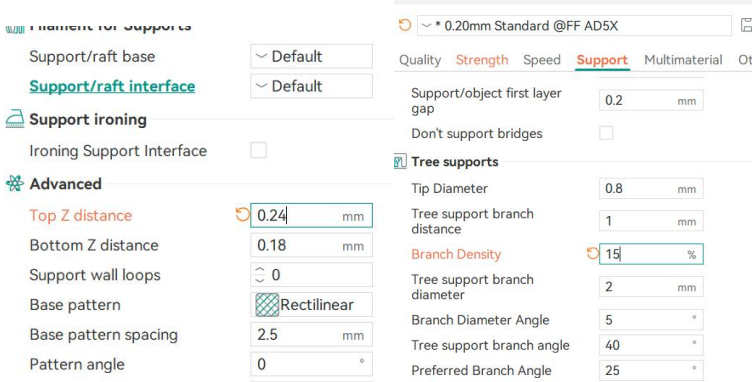


As shown above, depending on strength requirements: increase [Wall loops] to 4-6, increase [Sparse infill density] to 30-40%, and adjust [Sparse infill pattern] to Gyroid or Honeycomb.

Tip: For better interlayer adhesion, consider increasing the nozzle temperature by 5-10°C.

When printing models with thin branches using PLA Silk, although supports are easier to remove due to brittleness, stress during support removal can concentrate on the thin branches, causing them to break from the main model instead of the support.

In such case, slightly increase the [Top Z distance] (e.g., from 0.2mm to 0.24mm/0.28mm) and reduce the [Branch Density] (e.g., from 30% to 15-20%). This reduces adhesion between support and model, lowering the chance of damaging the model during support removal.



6. Advanced Print Settings

6.1 Achieve a Silk-Like Texture

To achieve a smooth, silk-like finish with PLA Silk, the following adjustments are recommended:

6.1.1 Reduce the Outer Wall Speed

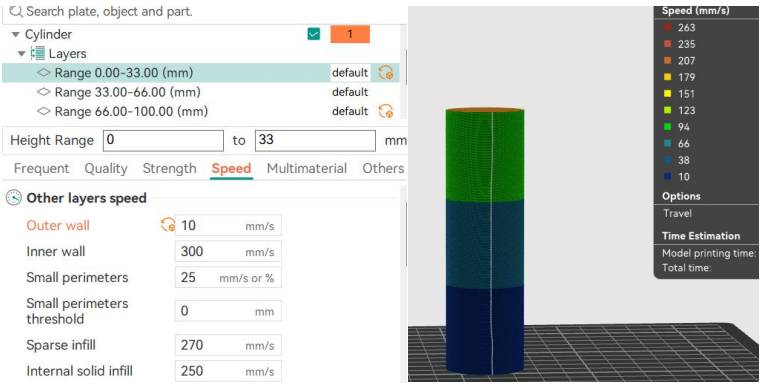
Since gloss levels vary by brand and color, optimal print speeds will differ. We recommend printing a gloss test piece before starting your project. The procedure below uses Flashforge PLA Silk (Gold) as an example:

① Open Orca-Flashforge and add a 100mm cylinder;

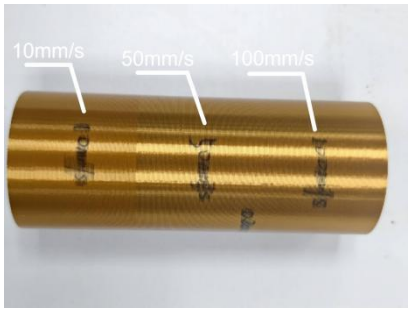
② Right-click to add a Height Range Modifier, dividing it into three sections: 0-33mm, 33-66mm, and 66-100mm. Set the outer wall speed for each section to 10, 50, and 100mm/s respectively (you can use more speed gradients if desired; this example is just for demonstration);

③ Preview the sliced model to check if the print speeds are achievable (if not, it is likely limited by the max volumetric speed of the filament);

④ Print the model and observe the surface finish.



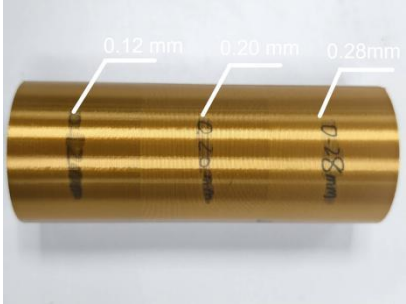
Here's how the printed model looks in terms of surface gloss:



Therefore, when printing at a 0.2mm layer height, the outer wall speed for Flashforge PLA Silk (Gold) should be set below 50mm/s to achieve optimal surface gloss.

6.1.2 Increase the Layer Height Appropriately

Similarly, layer height is another key factor affecting the gloss of PLA Silk. In this section, we again use Flashforge PLA Silk (Gold) as an example. The cylinder is divided into three sections with layer heights of 0.12mm, 0.20mm, and 0.28mm, while the outer wall speed is set to 20mm/s for all sections (illustration omitted for simplicity). The resulting surface gloss of the printed model is shown below:

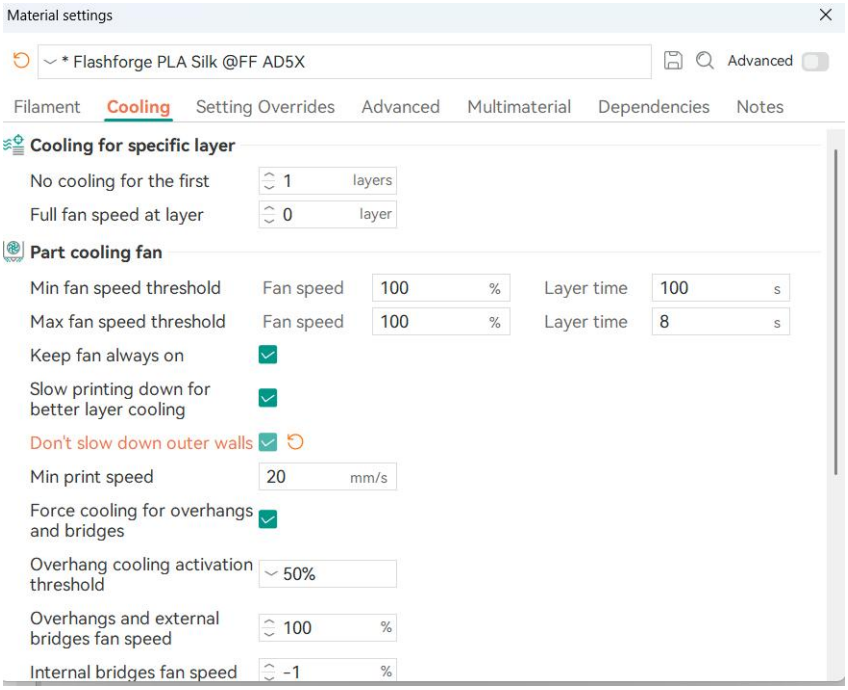


Therefore, when printing Flashforge PLA Silk (Gold) at an outer wall speed of 20mm/s, a 0.28mm layer height significantly enhances the surface gloss.

6.1.3 Combined Testing

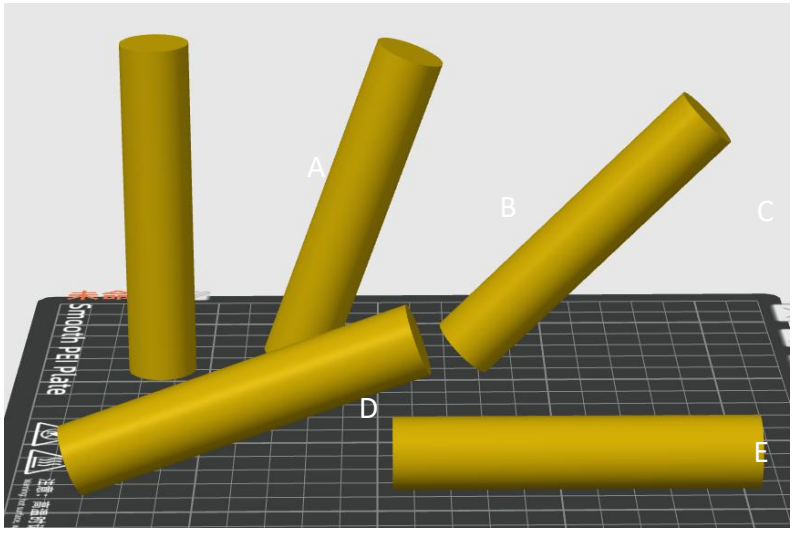
As mentioned above, gloss depends on factors such as filament brand, color, layer height, and outer wall speed. Therefore, there is no universal optimal setting. To obtain the best gloss, cross-testing outer wall speed and layer height is strongly recommended before final printing.

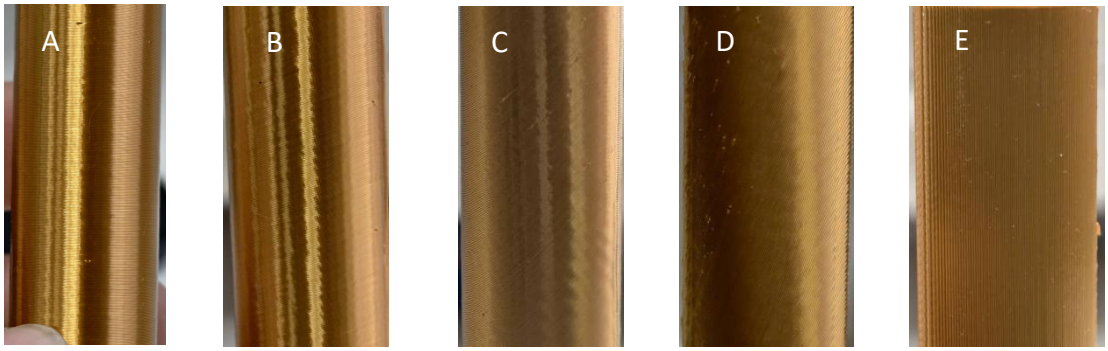
Tip: For models like 3DBenchy, where print speed can fluctuate significantly due to layer time, click [Cooling] and enable [Don't slow down outer walls] in the Material settings interface to significantly reduce variations in surface gloss.



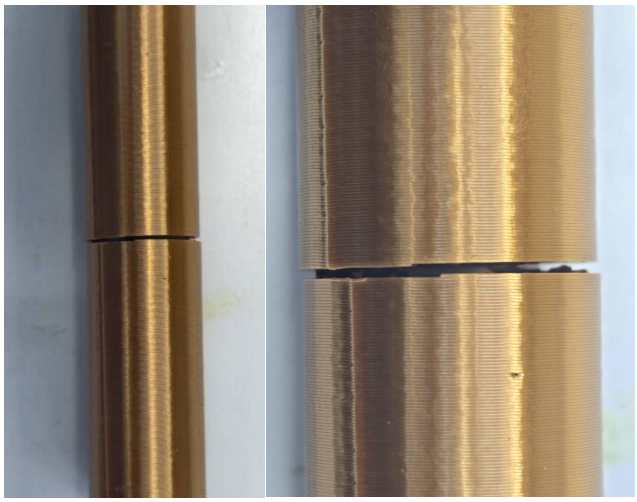
6.2 Improve PLA Silk Strength

Due to its composition, PLA Silk has weaker interlayer adhesion and is more prone to breaking along the Z-axis compared to standard PLA. For printing load-bearing components with PLA Silk, in addition to conventional approaches such as increasing wall loops and adjusting infill pattern and density, it is also recommended to print the model at a tilted angle to improve strength in the load direction.





These images illustrate cylinders oriented at 90°, 70°, 45°, 20°, and 0° relative to the build plate and the corresponding print results. When the load is applied parallel to the build plate, Cylinder A is the most likely to fracture, as shown below.



For Cylinders B-E, the direction of interlayer bonding forms an angle with the load direction. As a result, part of the applied force is borne by the filament itself, which has higher intrinsic strength. This reduces interlayer peeling stress, making layer separation less likely. Under horizontal loading, the strength ranking of Cylinders A-E from highest to lowest is: E > D > C > B > A.

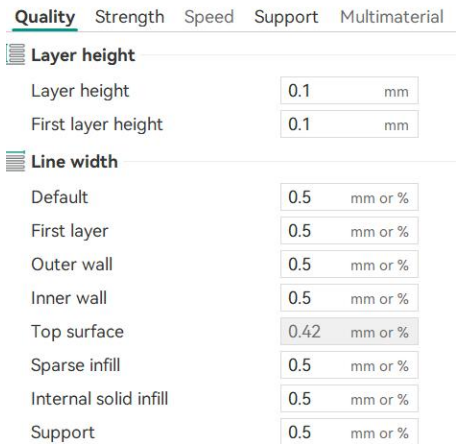
Users should choose the print orientation based on the expected load direction and the surface quality requirements of the model, balancing strength and appearance. In most cases, and

unless there are special requirements, it is generally recommended to orient the model so that the primary load direction forms an angle of approximately 45° to the build plate.

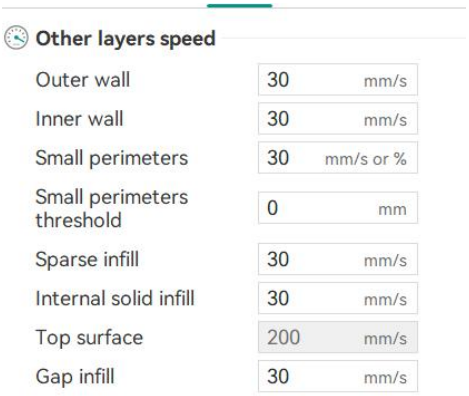
6.3 Achieve Better Transparency

Flashforge offers a transparent PLA (PLA Transparent) optimized for clear prints. To maximize transparency, adjust parameters to reduce internal light scattering and surface defects.

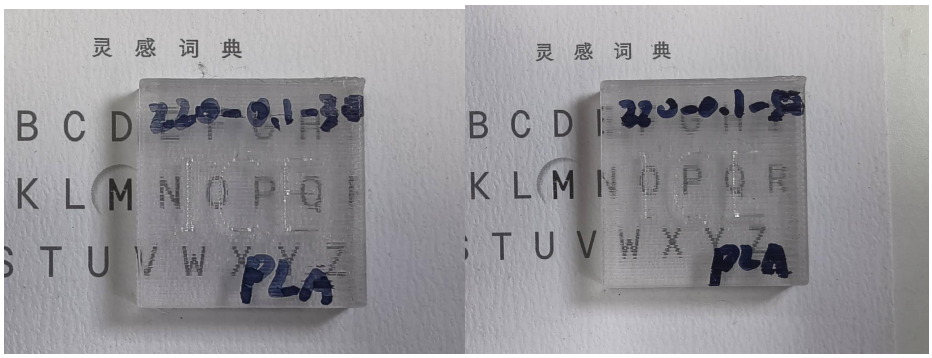
- Dry the filament: It is recommended to dry the filament before printing to prevent bubbles or voids caused by moisture, which can affect transparency.
- Optimize print parameters (0.4mm nozzle):
- Lower the layer height: Set the layer height and first layer height to 0.1mm. Thinner layers improve fusion, reduce gaps, and minimize light scattering.
- Increase the line width: Set all to 0.5mm to enhance material fusion.



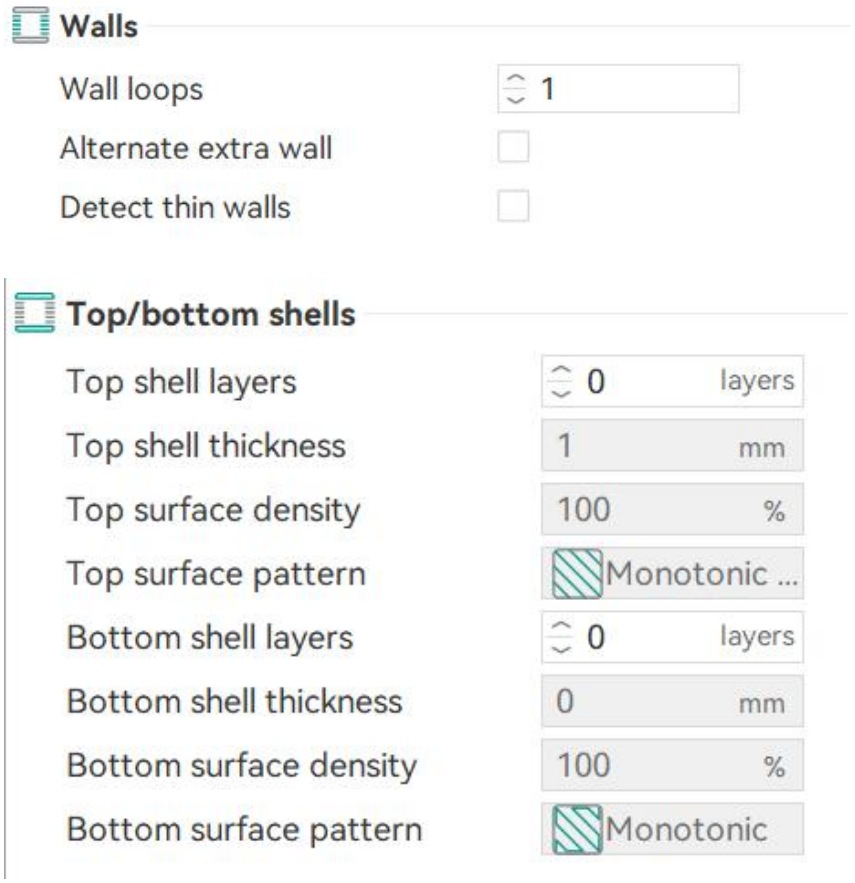
- Reduce the print speed: Lower all speeds to 30mm/s for more stable extrusion.

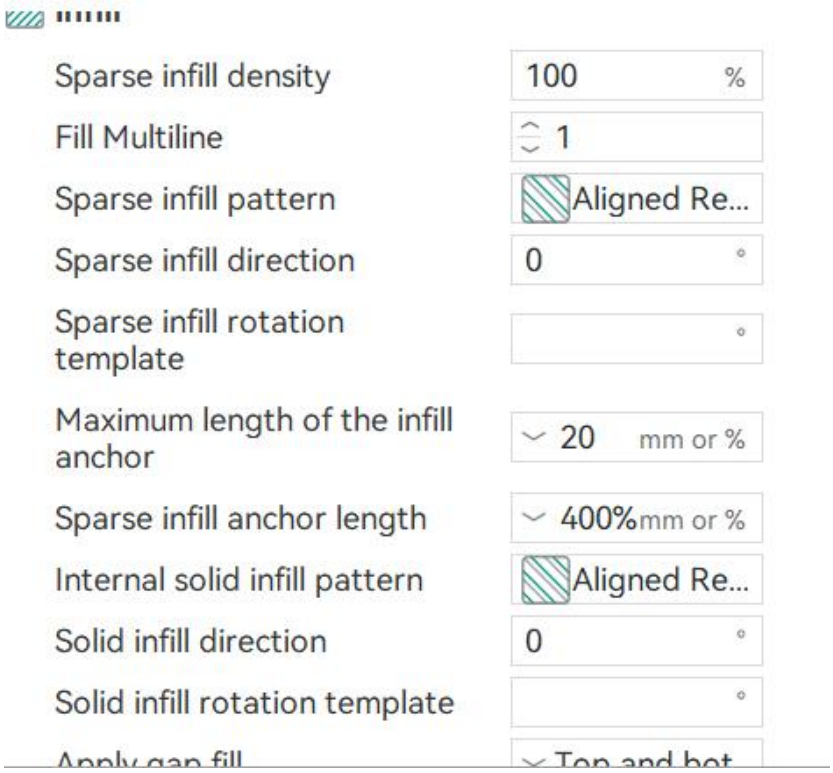


- The images below compare transparency at 30mm/s and 50mm/s. 30mm/s yields a clearer result.

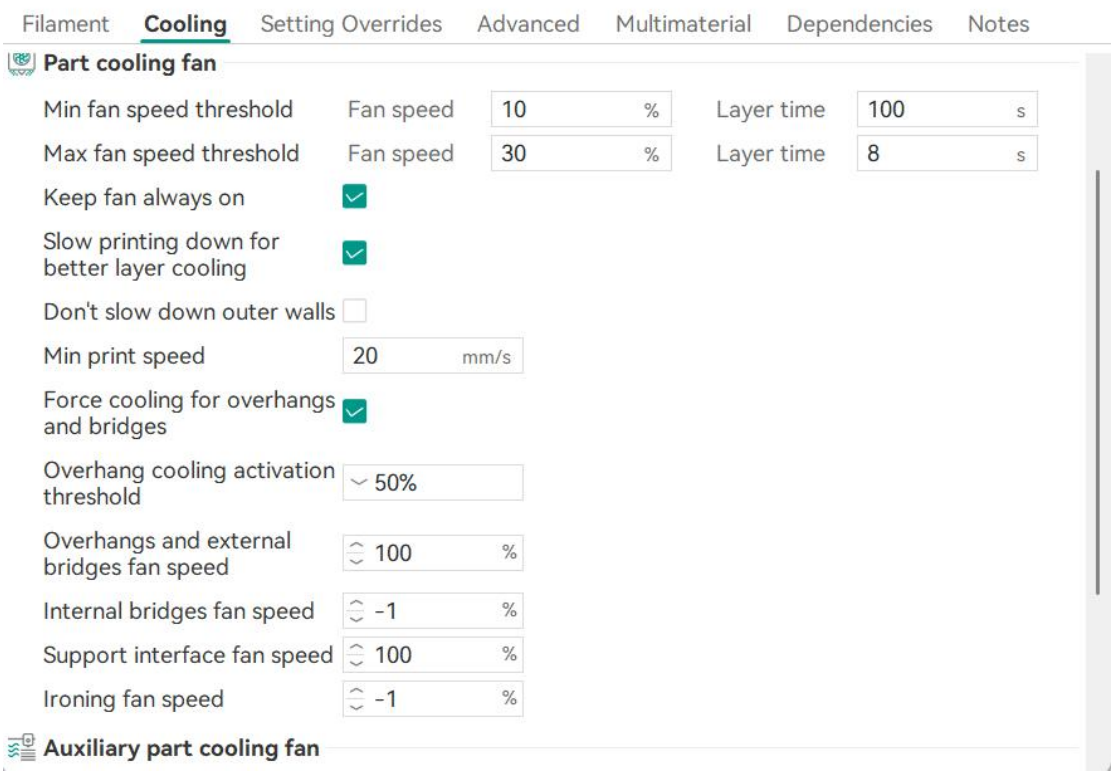


- Shell and infill settings: Set [Wall loops] to 1, [Top shell layers] to 0, [Bottom shell layers] to 0, and [Sparse infill density] to 100%. Choose "Aligned Rectilinear" for the infill pattern and set [Solid infill direction] to 0°. This ensures an even distribution of material along the light path, reducing scattering.

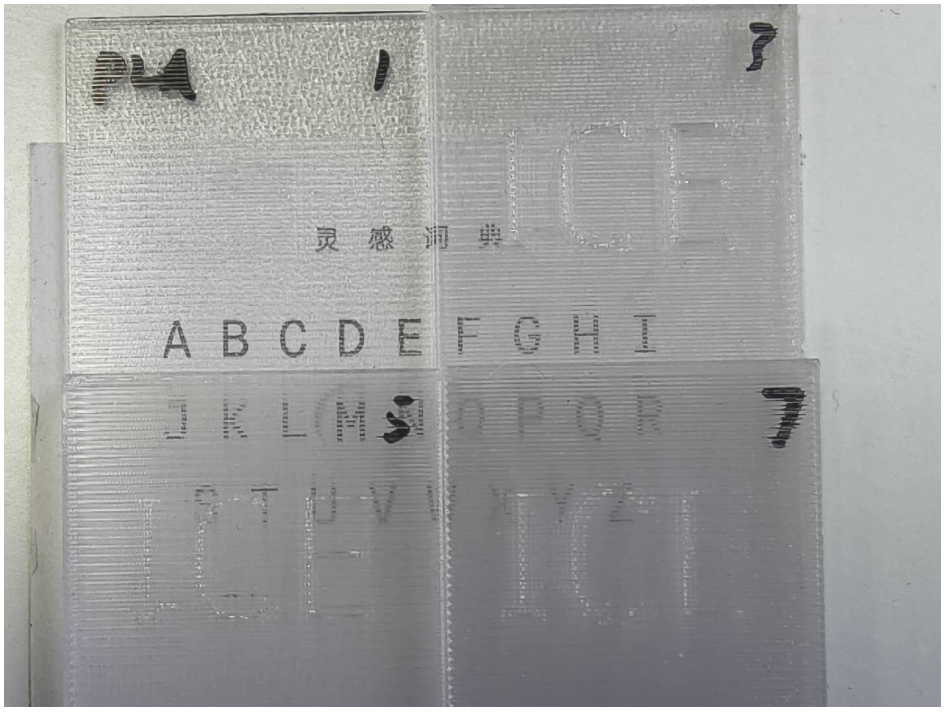




- Material settings: Slightly increase the flow ratio to 1. Raise the nozzle temperature to 220° C but do not exceed 220°C (higher temperatures can cause yellowing due to slower print speeds). This improves the melt quality and interlayer bonding. Reduce the [Min fan speed threshold] to 10% and the [Max fan speed threshold] to 30%, and turn off auxiliary fans (if present). Lower cooling rates enhance layer adhesion.



- Thickness reference: The image below shows light transmission through PLA transparent sheets of 1, 3, 5, and 7mm thickness. Use this as a guide when selecting the thickness for desired transparency.



6.4 Print PLA LW

Flashforge offers lightweight PLA (PLA LW) for model aircraft. Due to its unique foaming properties, the filament expands during printing, reducing final density and making prints lighter.

Key points when printing PLA LW:

- Expansion is proportional to the nozzle temperature. We recommend setting the nozzle temperature to 220°C, adjusting the flow ratio to 0.7, and setting the maximum volumetric speed to 8 to ensure proper foaming.
- Lower flow ratios yield lighter models, but users may need to fine-tune the nozzle temperature. Note that excessive foaming may compromise strength. Consider whether reduced durability is acceptable for the application.
- To further reduce weight, consider enabling the spiral vase mode, which prints single-wall models without sparse infill, saving filament.