

SHAP3D® 3D Printing Filament

SHAP3D® is a thermoplastic 3D printing filament developed by Orfit Industries for use in Fused Deposition Modeling (FDM) 3D printing process. It is intended for the fabrication of custom-made devices, including orthotic applications, by qualified professionals.

The material allows for low-temperature post-print remolding, enabling localized adjustments during fabrication. SHAP3D® combines ease of printing with efficient post-print processing and finishing.

Sharing similar material properties with Orfit's trusted low-temperature thermoplastic sheets, SHAP3D offers the reliability and performance expected from Orfit thermoplastic solutions.

This product is latex-free. It is supplied non-sterile. Refer to the *MAINTENANCE* section for cleaning and disinfection instructions.

SHAP3D® is supplied as a filament for 3D printing and is not a medical device. Therefore, it is not CE-marked under Regulation (EU) 2017/745 (MDR).

The regulatory status of any final product manufactured using SHAP3D® is determined by its intended use and remains the responsibility of the user.

Orfit, as manufacturer of SHAP3D® filament, is responsible for:

- Supplying filament that meets defined quality and material specifications
- Providing Technical Data Sheet (TDS), Safety Data Sheet (SDS), and material-related information
- Providing biocompatibility data for the filament as supplied

The user is responsible for:

- The design and intended use of any final product manufactured using SHAP3D®
- The 3D printing process, post-print processing, and final product performance
- Verifying that the final product is safe and suitable for its intended application

If the final product is intended for medical use, the user becomes the manufacturer of that product and is responsible for ensuring compliance with applicable regulations.

SHAP3D® has been tested for biocompatibility in accordance with ISO 10993-1:2025 and is suitable for prolonged contact with intact skin at the material level. The biocompatibility assessment applies to SHAP3D® filament and remains applicable when processed according to the recommended printing conditions. The user remains responsible for ensuring the suitability of the final printed product for its intended application.

SHAP3D® filaments are available in rolls in different colors, diameters and lengths to meet a wide range of 3D printing requirements.

Article Number	Color	Weight	Diameter	Length
3DF/B/1.75/1	Black	1 kg (2.2 lbs)	1.75 mm	365 m (399 yd)
3DF/OR/1.75/1	Orange	1 kg (2.2 lbs)	1.75 mm	365 m (399 yd)
3DF/W/1.75/1	White	1 kg (2.2 lbs)	1.75 mm	365 m (399 yd)
3DF/B/1.75/0.5	Black	0.5 kg (1.1 lbs)	1.75 mm	182.5 m (199.5 yd)
3DF/OR/1.75/0.5	Orange	0.5 kg (1.1 lbs)	1.75 mm	182.5 m (199.5 yd)
3DF/W/1.75/0.5	White	0.5 kg (1.1 lbs)	1.75 mm	182.5 m (199.5 yd)

1. **Do not dry or preheat the filament.** It is ready to print.
2. The filament softens and becomes sticky at temperatures above 45 °C (113 °F).
3. Ensure the filament diameter matches the 3D printer's nozzle specifications.
4. Trim the filament tip before loading to prevent jams or blockages.
5. This filament must be printed using a direct feed system with an external spool holder only.
Do not use the printer's AMS accessory, as it may cause filament jams or print defects.
6. Ensure the printing plate is clean and correctly installed.
7. Use a smooth print plate for easier part removal and a textured plate for better adhesion.
8. The workplace must be well-ventilated.

1. Load the filament onto the external spool holder according to the printer manufacturer's instructions.
 2. Import your model/design in STL format into the 3D printing software (slicer).
 3. Ensure that the size of the 3D model is compatible with the print volume of your printer.
 4. Ensure the model is correctly oriented in the slicer for optimal printing results.
 5. Select your printer and filament within the slicer.
 6. Adjust the print parameters according to specifications in the filament Technical Data Sheet.
⚠ Do not exceed the recommended printing speed to prevent nozzle blockages and ensure high-quality prints.
 7. Slice the model using the slicer and save the project under the .3mf format.
 8. If the slicer is connected via Wi-Fi, send the print file directly through the printer's desktop or mobile application.
 9. If not connected, generate a G-code file from the slicer, transfer it to the printer, and start the print.
- For detailed guidance on advanced settings—such as supports, infill, and other parameters—consult your printer manual.

SPECIFICATION	VALUE
DIAMETER	1.75 mm
TOLERANCE	± 0.05 mm
PRINTING (NOZZLE) TEMP	160 - 200 °C
NOZZLE SIZE RECOMENDATION	0.4 mm
BED TEMPERATURE	25 - 35 °C (PEI) 40 - 45 °C (Others) Tip: Increase bed temperature up to 55 °C for better adhesion.
FLOW	100 - 120%
LAYER THICKNESS	0.1 - 0.2 mm
COOLING FAN (RECOMMENDED)	80 - 100%
PRINTING SPEED	<180 mm/s (for 1.75 mm)
RETRACTION	0.5 - 1 mm
SHRINKAGE	1.25%
SPOOL WEIGHT	1 kg/ 500 gr
COLOR RANGE	White, Orange, Black

Removing the Printed Object

Once the printing process is complete:

1. Carefully remove the printed object from the build plate.
2. Remove any support structures according to the requirements of the printed design.

Post-Print Processing (Finishing and Adjustments)

Recommended temperature range: 55–65 °C (131–149 °F)

• **Finishing**

For surface smoothing and edge refinement, apply local heat by immersing the required area in water at 55–65 °C (131–149 °F) for 10–15 seconds, depending on the thickness and infill of the printed object.

⚠ Do not fully immerse the printed object, as this may affect its overall shape.

• **Local Reshaping and Adjustments**

To locally reshape or modify the printed object, immerse the required area in water at 55–65 °C (131–149 °F) for 15–25 seconds.

If the material is not sufficiently softened, repeat immersion in short intervals until the required flexibility for adjustment is reached.

Note: Lower temperatures require longer heating times. For example, at 55 °C (131 °F), approximately 3 minutes may be required to soften a 1.6 mm thick printed object sufficiently for adjustment.

⚠ Overheating Warning:

Do not immerse the printed object in:

- 55 °C (131 °F) water for more than 3 minutes
- 65 °C (149 °F) water for more than 25 seconds

Exceeding these recommended heating times may result in unintended deformation or melting.

Cleaning the Print Plate

Heat the build plate to 65–70 °C (149–158 °F) or immerse it in water at the same temperature range. Remove residual material using a suitable tool, such as a metal spatula.

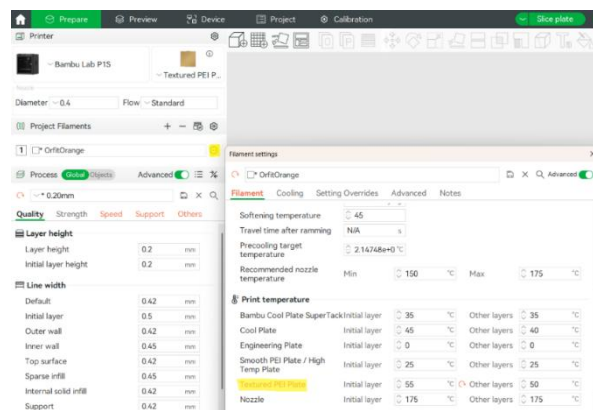
Note: This guide is based on **Bambu Studio software**. As the software is regularly updated, some features and settings may look different from those shown in the screenshots below.

PROBLEM 1: WARPING/ LIFTING CORNERS

When the print's edges lift or curl up from the build plate

Cause 1: Bed temperature too low

Solution 1: Raise bed temperature



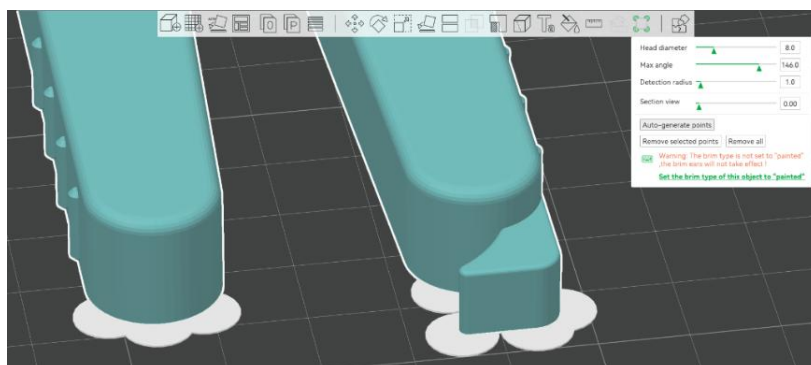
Solution 2: Add brims

• **Automatically generate brim corners**

Bambu lab studio → Prepare → Click on Brim Ears → Click on Auto-generate points

• **Manually generate brim corners**

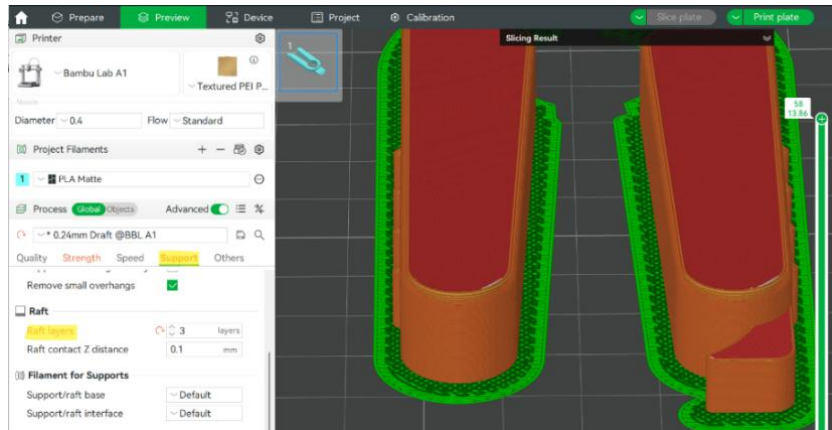
Bambu lab studio → Prepare → Click on Brim Ears → Adjust points manually



Note: Brims help the part stick to the build plate and reduce warping, but they take extra time to remove. Use brims only where needed (e.g., corners) to reduce cleanup. If corner brims are not available, use a full brim.

Solution 3: Add Raft Layers (less recommended)

Bambu lab studio → Prepare → Support → Raft Layers (set to a value >0, e.g.: 3)



Note: Rafts provide strong adhesion but require more cleanup and may not work for all shapes. Use only if brims are not effective.

Cause 2: Bad adhesion

Solution 1: Use adhesive (e.g. Magigoo Pro FLEX)

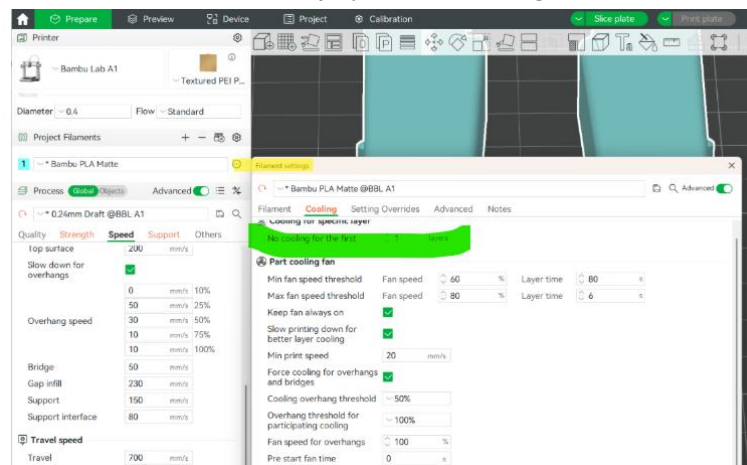


Solution 2: Use textured plate (e.g. Textured PEI Plate)

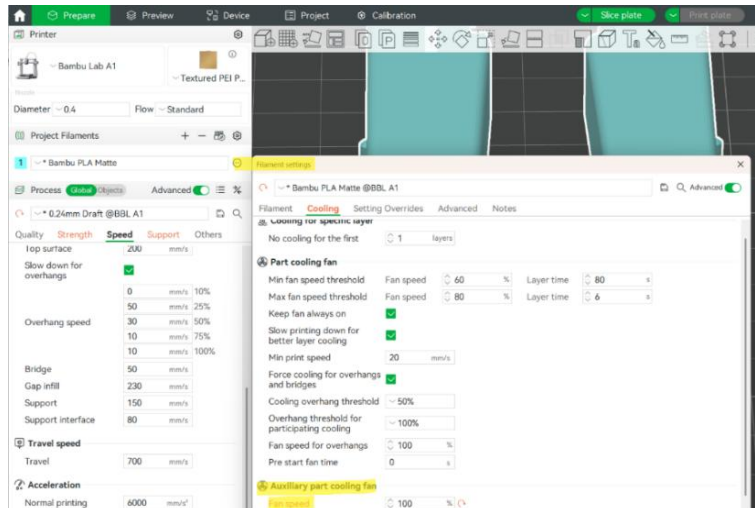


Cause 3: Uneven cooling

Solution 1: Increase the number of layers without cooling



Solution 2: Reduce the percentage of auxiliary cooling to 60-80%



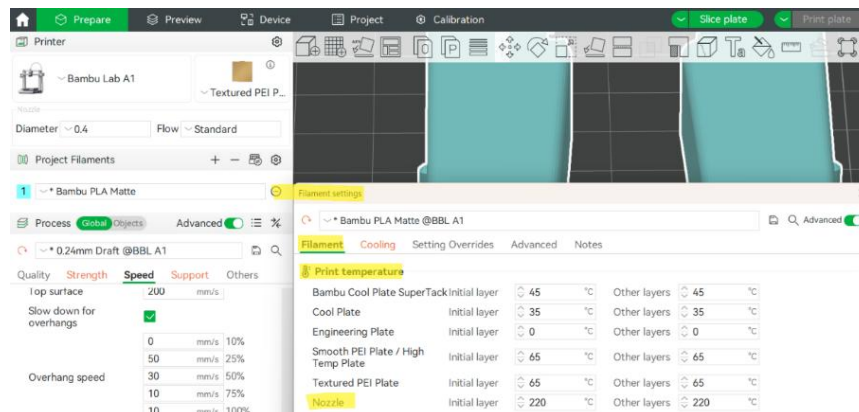
Solution 3: Use enclosed printer (like Bambu Lab P1S)

PROBLEM 2: STRINGING / BLOBS

Thin plastic strands or small lumps that appear on the print surface

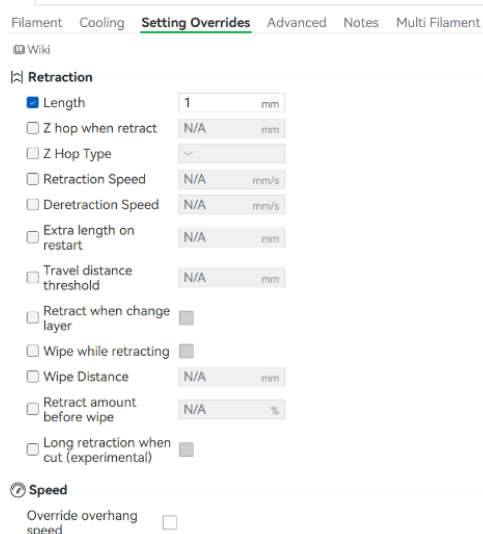
Cause 1: Extrusion temperature too high

Solution: Lower the nozzle temperature

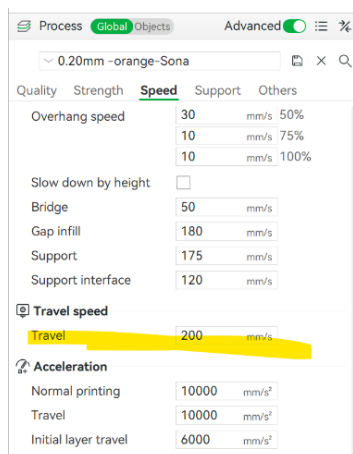


Cause 2: Retraction not set

Solution 1: Increase retraction length or speed



Solution 2: Increase travel speed



Solution 3: Add nozzle wipe

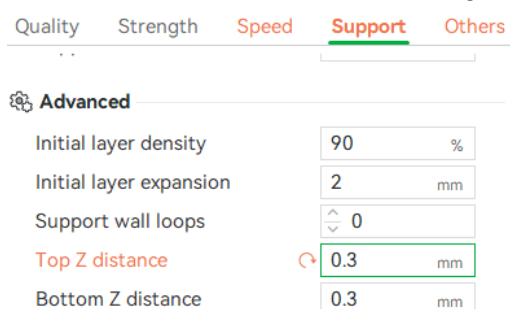
Nozzle wipe cleans the nozzle during printing to reduce stringing. This feature is not available in Bambu Studio but is supported in other slicers (e.g., Cura, Prusa Slicer, Simplify3D).

PROBLEM 3: DIFFICULT SUPPORT REMOVAL

When support structures stick too firmly to the model

Cause: Material is sticky at high printing temperature

Solution: Increase the Z-distance in slicer settings



- Products manufactured using SHAP3D® filament can be cleaned with lukewarm water and liquid soap, biological detergent, or toothpaste. Rinse well and dry thoroughly. Alternatively, pre-moistened isopropyl alcohol wipes can be used.

⚠ **If you are not sure about the cleaning fluid, do not use it.**

⚠ **Never use solvents and avoid abrasive detergents.**

⚠ **Do not put the entire finished product into the hot water \as this might cause the loss of the shape.**

- The filament *cannot* be sterilized in an autoclave. If needed, it can be done at room temperature to prevent melting.
- Disinfection is possible with alcohol, quaternary ammonium, or a solution of commercial disinfection soaps (HAC®, Sterilium®, etc.).

Unused filament, production waste, and final printed products can be disposed of with normal household waste.

The qualified professional responsible for the preparation, application, or integration of the product into the final application shall provide the user with appropriate information regarding its use, maintenance, and limitations.

The user should follow the instructions provided for the correct use of the final application. Contact the qualified professional if adjustments, modifications, or further guidance are required.

When not in use, store the filament under the following conditions:

- Store in the original packaging in a cool, dry place, protected from light, at a temperature between 10°C (50°F) and 30°C (86°F)
- Keep labels and lot numbers for traceability
- Observe the shelf life indicated on the product label
- Protect the filament from excessive heat, humidity, and direct light to maintain material performance
- Return the filament to its original packaging to preserve quality

- For external use only. Do not apply on open wounds, damaged skin, or inside the mouth.
- Do not expose finished products made from SHAP3D® to temperatures above 60 °C (140 °F). Higher temperatures may cause deformation. This may occur, for example, if the product is left in direct sunlight (e.g., behind a car windshield) or placed near heat sources such as radiators or fireplaces.
- Any serious incident related to the use of this product must be reported to your local distributor. For distributor contact details, please consult the official Orfit website.

For additional information such as distributor contact information, product brochures, Safety Data Sheets, and regulatory information, please visit our website www.orfit.com.

Note:

It is prohibited to make alterations to this text without prior approval from Orfit Industries.
ORFIT® and SHAP3D® are registered trademarks of ORFIT INDUSTRIES N.V.

ORFIT INDUSTRIES

Vosveld 9A | B-2110 Wijnegem | Belgium
T (+32) (0)3 326 20 26
welcome@orfit.com

www.orfit.com



Ref. No. 50306
VERSION 2
LAST UPDATE: 28/07/2026
REVISION DATE: 28/07/2028