SHAP3D
3D Printing Filament

GENERAL PRODUCT INFORMATION

SHAP3D is Orfit's innovative, patented 3D printing filament specifically developed for orthotic applications. This low temperature thermoplastic filament is optimized for standard FDM 3D printing technology.

SHAP3D is used to fabricate custom orthoses and medical aids that support patients in their daily activities, providing durable, precise, and high-performance solutions.

The filament is biocompatible. It is tested and certified according to ISO 10993-1:2018 and approved for prolonged use on intact skin.

Key features include easy remolding at low temperatures, allowing clinicians to adjust the orthosis after printing for an optimal fit and comfort.

Sharing similar properties with Orfit's trusted low-temperature thermoplastic sheets, SHAP3D provides the reliability and performance clinicians expect in medical and rehabilitation applications.

PRODUCT RANGE

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

Article Number	Colour	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)

PRECAUTIONS BEFORE PRINTING

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. For easier removal of printed parts, a smooth print plate is recommended; For better adhesion, a textured plate is recommended.
8. The printing environment must be well-ventilated.

PRINTING TECHNIQUE

1. Load the filament manually onto the external spool holder of your printer, following 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 to the printer via Wi-Fi, send the print order directly via 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's manual.

TECHNICAL SPECIFICATIONS

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:*** *Bed temperature can be increased 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) <100 mm/s (for 2.85 mm)
RETRACTION	0.5 - 1 mm
SHRINKAGE	1.25%
SPOOL WEIGHT	1 kg/ 500 gr
COLOR RANGE	White, Orange, Black

WORKING TECHNIQUE

Removing the Printed Orthosis

Once the printed orthosis is ready,

1. Carefully remove the orthosis from the build plate.
2. Detach any support structures.

Post-Print Processing (Finishing and Adjustments)

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

- **Finishing:** For surface smoothing and edge refinement, briefly apply heat locally by immersion in water at 55–65 °C (131–149 °F) for 10–15 seconds, depending on thickness and infill.
  ⚠ Do not fully immerse the orthosis, as this may affect its overall shape.

- **Local Adjustments**
  For fit adjustment or reshaping, immerse locally the specific 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), 3 minutes is sufficient to adjust a 1.6 mm thick orthosis.*

⚠ Overheating Warning:

Do not immerse the printed orthosis in:

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

Exceeding these times may cause unintended deformation or melting.

Cleaning the Print Plate:

Heat the plate to 65–70 °C (131–158 °F) or immerse it in water at the same temperature, then remove excess material using a metal spatula.

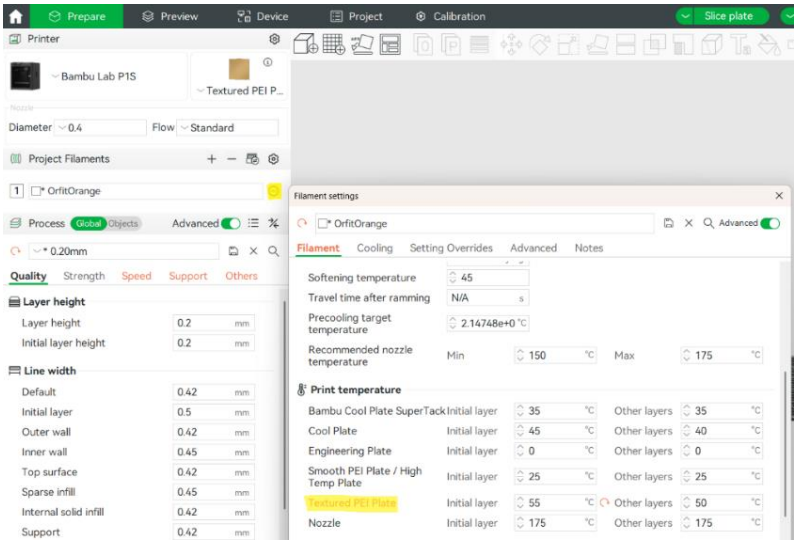
***Note:** This guide is based on **Bambu Studio software**. As the software is regularly updated, some features and settings may look different from those shown in the screenshots below.*

Problem 1: WARPING/ LIFTING CORNERS

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

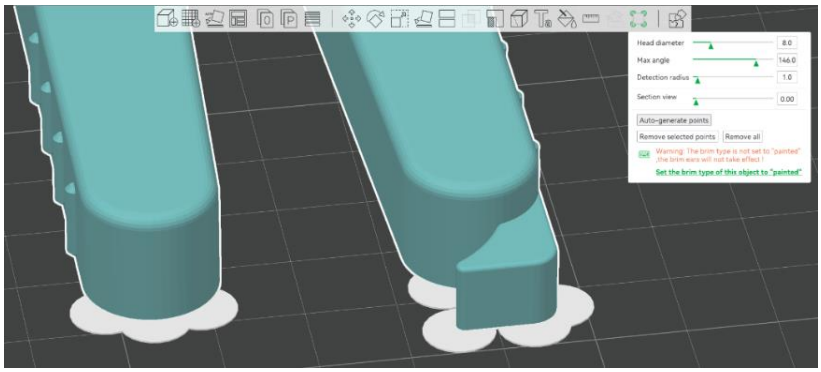
Cause 1: Bed temperature too low

***Solution 1:** Raise bed temperature*



***Solution 2:** Add brims*

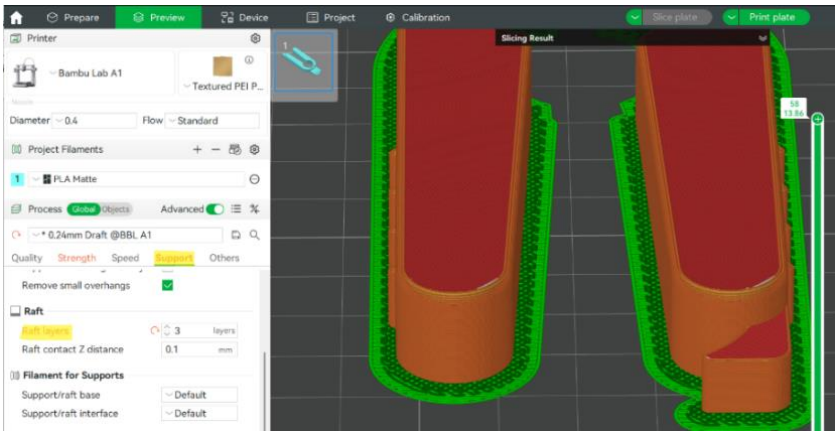
- ***Automatically generate brim corners***
  Bambu lab studio → Prepare → Click on Brim Ears → Click on Auto-generate points
- ***Manually generate brim corners***
  Bambu lab studio → Prepare → Click on Brim Ears → Adjust points manually



***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.*

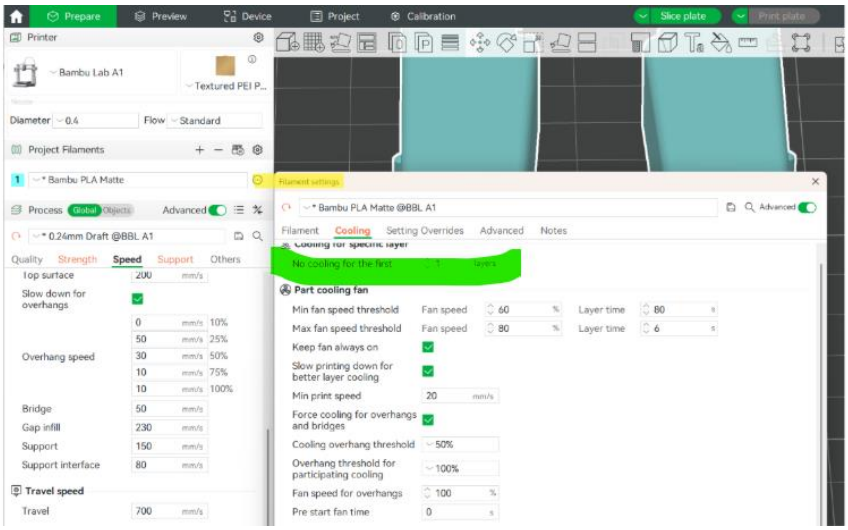
Cause 2: Bad adhesion

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

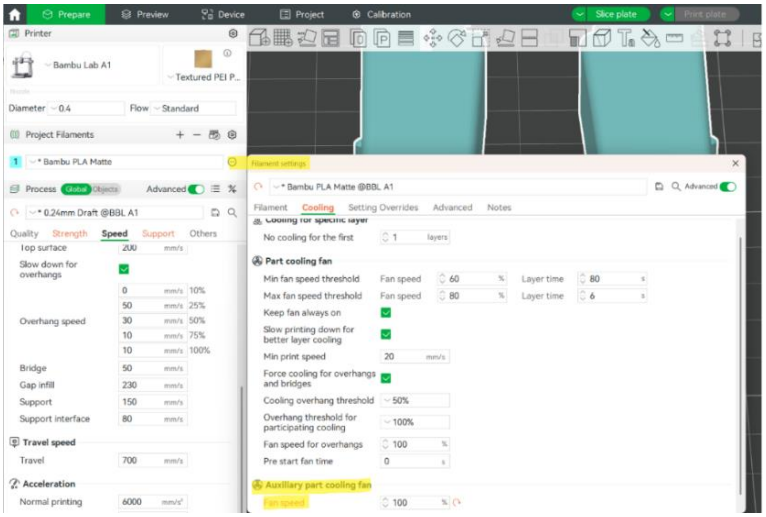
Solution 2: *Use texturized plate (e.g. Textured PEI Plate)*

Cause 3: Uneven cooling

Solution 1: *Increase the number of layers without cooling*



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



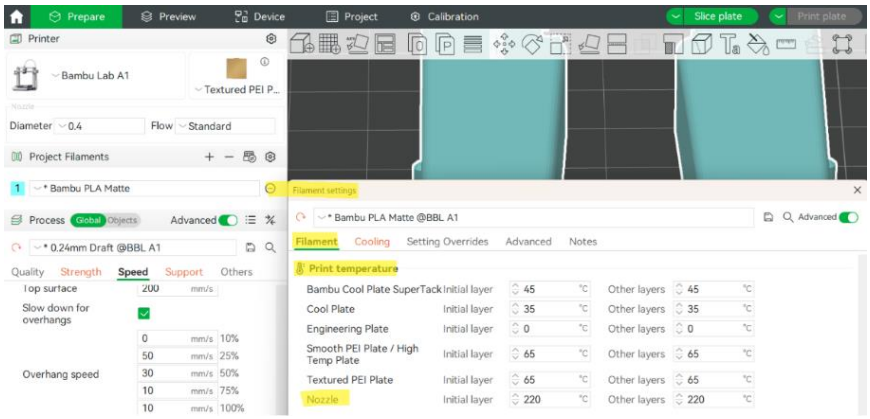
Solution 3: Use enclosed printer *(like Bambu Lab P1S)*

Problem 2: STRINGING / BLOBS

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

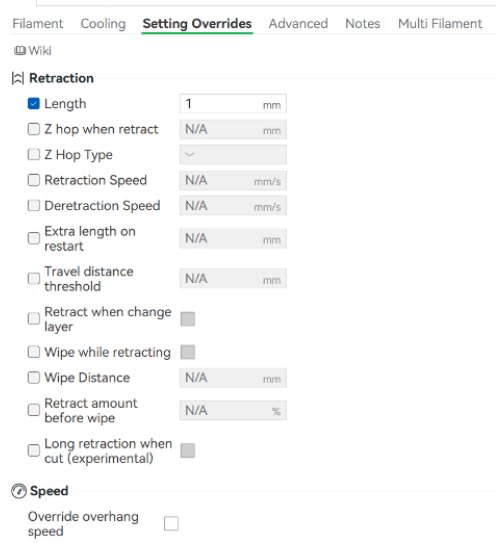
Cause 1: Extrusion temperature too high

Solution: *Lower the nozzle temperature*

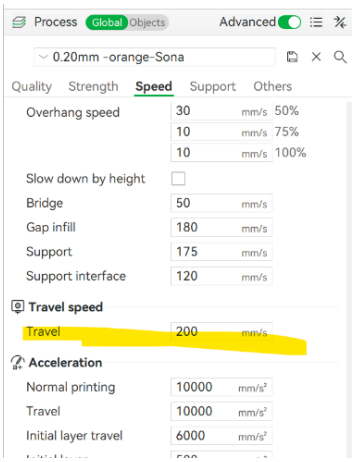


Cause 2: Retraction not set

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*

MAINTENANCE

- After printing, remove the spool from the printer and store it according to the instructions in the STORAGE section to maintain quality.
- Wash printed orthoses with lukewarm water and a mild cleaning agent, such as liquid soap, biological detergent, or toothpaste. Rinse thoroughly with clean water and dry completely. Alternatively, pre-moistened isopropyl alcohol wipes can be used.
  ⚠ If unsure about a cleaning agent, do not use it. Avoid solvents and abrasive detergents.
- 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.).

WASTE MANAGEMENT

Filament waste and printed orthoses can be safely disposed of with regular household waste without harming the environment.

INSTRUCTIONS FOR PATIENT

Provide the patient with sufficient and clear information on how to use the printed orthosis, wearing schedule, maintenance, and the possible constraints of the orthosis. Ensure that the patient fully understands the appropriate steps to take if adjustments are needed.
⚠ Do not use the printed orthosis on damaged skin or open wounds.

STORAGE

- The filament must be stored in a dark, cool, dry place at a temperature of min. 10°C (50°F) and max. 30°C (86°F). While in use, the filament can be kept on the printer's spool holder. However, when not in use, always return the filament to its storage conditions to prevent biodegradation.
- Do not lose the labels with lot numbers. They are crucial for product traceability.
- Our thermoplastic materials have a limited shelf life and can only be kept for a certain period (as indicated on the label). They must be protected as much as possible from light, heat, and humidity to prevent ageing.

GENERAL SAFETY ADVICE

⚠ This filament is not suitable for internal use. It may not be used on open wounds or in the mouth.

⚠ Never use an open flame to adjust the printed orthosis.

⚠ This filament may only be used by qualified health professionals to make 3D-printed orthoses and rehabilitation aids.

ADDITIONAL INFORMATION

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

CE
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 1
LAST UPDATE: 08/05/2026
REVISION DATE: 88/05/2028