Printing Guidelines

eFil TPU FOAM

Active-foaming flexible TPU filament — How to print, calibrate and tune

Document version: 0.2 (Working draft)

Date of issue: 30/04/2026

Living document. The values reported here reflect the current state of internal testing. They will be progressively updated as further characterization of the material is completed.

1. Product Overview

eFil TPU FOAM is a flexible TPU filament with active-foaming technology. When printed within the right temperature window, the material expands during extrusion — increasing line width, reducing part density and softening the final part. The same spool of filament can therefore deliver parts with different mechanical and tactile properties just by tuning printing parameters.

Because the material expands inside and just after the nozzle, the volumetric flow must be lowered to keep the actual line width — and therefore the geometry — under control. This guide explains how to set up your printer, how to compensate flow as a function of temperature, and how to troubleshoot the most common issues.

Key properties (unfoamed reference)

Property	Value
Base Shore hardness (unfoamed)	98 A
Filament density	1.21 g/cc
Glass transition (Tg)	−27 °C
Tensile strength	40 MPa
Elongation at break	400 %
Tear strength	170 N/mm
Maximum volumetric expansion	Up to 1.6× (≈40 % weight reduction)

2. Quick Reference

If you only read one page of this document, read this one. Recommended starting point for a 0.6 mm nozzle:

Parameter	Recommended starting value
Nozzle diameter	0.6 mm

Parameter	Recommended starting value
Layer height	0.30 mm
Nozzle temperature	230 °C (working range 210–250 °C)
Bed temperature	50–60 °C
Print speed	Up to 60 mm/s
Flow / extrusion multiplier	≈ 53 % at 230 °C (see §6)
Retraction distance	0.4 mm
Travel speed	200–300 mm/s (high to minimise stringing)
Part cooling fan	0–20 % (keep low to allow foaming)
Build surface	PEI / glass with PVA glue stick or adhesion spray
Drying before printing	4–6 h at 70 °C

3. Drying & Storage

TPU is hygroscopic — it absorbs ambient moisture. With a foaming TPU this is especially critical, because absorbed water turns to steam in the melt zone and competes with the foaming agent: too much moisture causes uncontrolled, irregular expansion, surface bubbles and stringing.

Drying procedure

1. Place the spool in a filament dryer or convection oven.
2. Dry for 4 to 6 hours at 70 °C.
3. Aim for < 15 % relative humidity inside the dryer before printing.
4. Print directly from the dry box whenever possible — do not let the spool sit on the bench for long sessions.

Storage

- Store the spool in its sealed bag with the silica gel pack provided.
- Keep below 40 °C and out of direct sunlight.
- Re-dry before any new print job if the spool has been left exposed for more than ~24 h.

4. Printer Compatibility & Setup

eFil TPU FOAM is a soft material and behaves like a regular TPU outside of the foaming window. Make sure your printer is ready to handle flexible filaments before starting:

- Use a direct-drive extruder. Bowden setups are not recommended — the elastic filament buckles in the tube and clogs.
- Use an extruder with adjustable grip pressure. The ideal grip is firm but does not crush the filament.
- Make sure the path from extruder gear to nozzle is well constrained — any gap lets the filament buckle.

- Use a hardened nozzle if you plan to print large volumes; the foaming agent is not abrasive but production runs benefit from a wear-resistant tip.
- An enclosure is not required, but it helps with consistency on long prints.

5. Print Settings — 0.6 mm Nozzle

Values below are the recommended starting point for a 0.6 mm nozzle.

Section	Parameter	Recommended value
Geometry	Nozzle diameter	0.6 mm
	Layer height	0.30 mm
	Target line width	0.62 mm
Temperature	Nozzle temperature	230 °C (range 210–250 °C)
	Bed temperature (small parts)	Room temperature or off
	Bed temperature (large parts)	50–60 °C
Speed	Print speed	Up to 60 mm/s
	Travel speed	200–300 mm/s
Retraction	Distance	0.4 mm
	Z-hop	0.2 mm (optional)
Cooling	Part cooling fan (general)	0–20 %
	Layers shorter than 10 s	Up to 40 %
	First layer	0 %
Adhesion	Recommended surfaces	PEI, glass with PVA glue stick, adhesion spray
	Brim	5 mm recommended for tall or narrow parts

6. Foaming Behavior — Temperature, Expansion & Flow

The foaming agent inside eFil TPU FOAM activates progressively. The higher the nozzle temperature, the larger the volumetric expansion of the extruded line. To keep the printed line width on target, the volumetric flow must be reduced by the same ratio.

Working temperature range: 210–250 °C. Above 250 °C the cell structure collapses and foaming becomes unreliable, so this temperature is considered the upper limit of the printable window.

6.1 Material behavior by nozzle temperature

Measured with a 0.6 mm nozzle, 0.30 mm layer height, 100 % flow. The extruded line width is measured directly on a single-wall test cube.

Nozzle temp.	Line width @ flow 100 %	Line expansion vs target (0.62 mm)	Notes
210 °C	0.68 mm	+9.7 %	Minimum foaming — material behaves close to standard TPU
220 °C	0.96 mm	+55.0 %	Light foaming — firmer, denser parts
230 °C	1.17 mm	+87.9 %	Recommended starting point — balanced foaming
240 °C	1.32 mm	+113 %	High foaming — softer, lighter parts
250 °C	1.40 mm	+126 %	Maximum usable foaming — soft, low-density parts

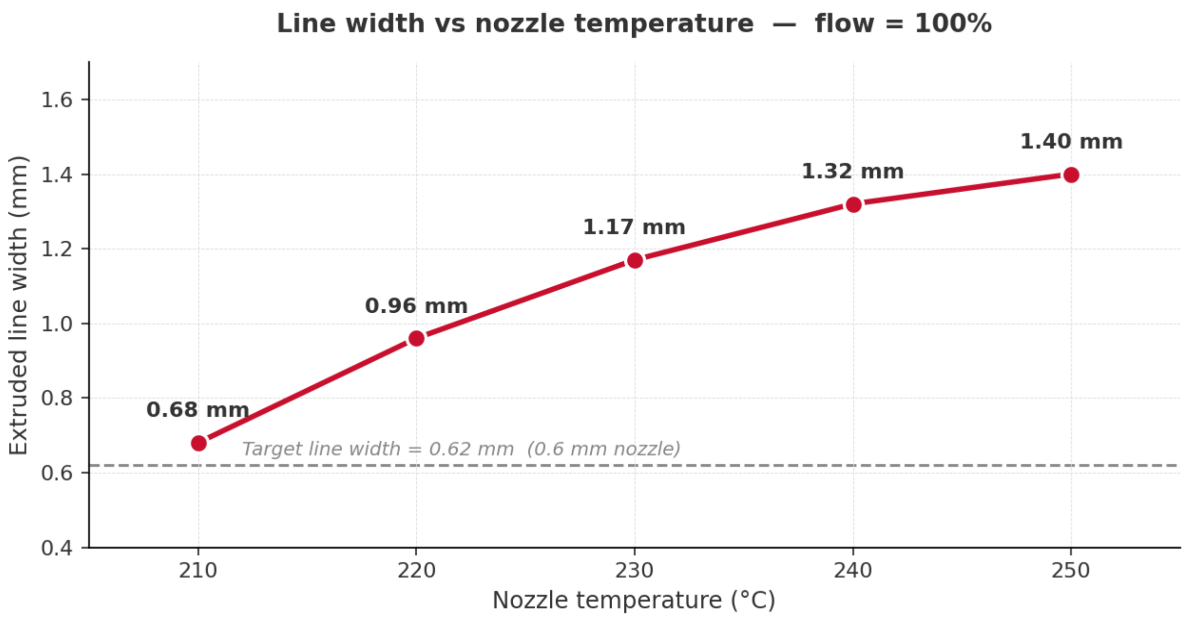

Figure 1 — Extruded line width as a function of nozzle temperature, flow set to 100 %.

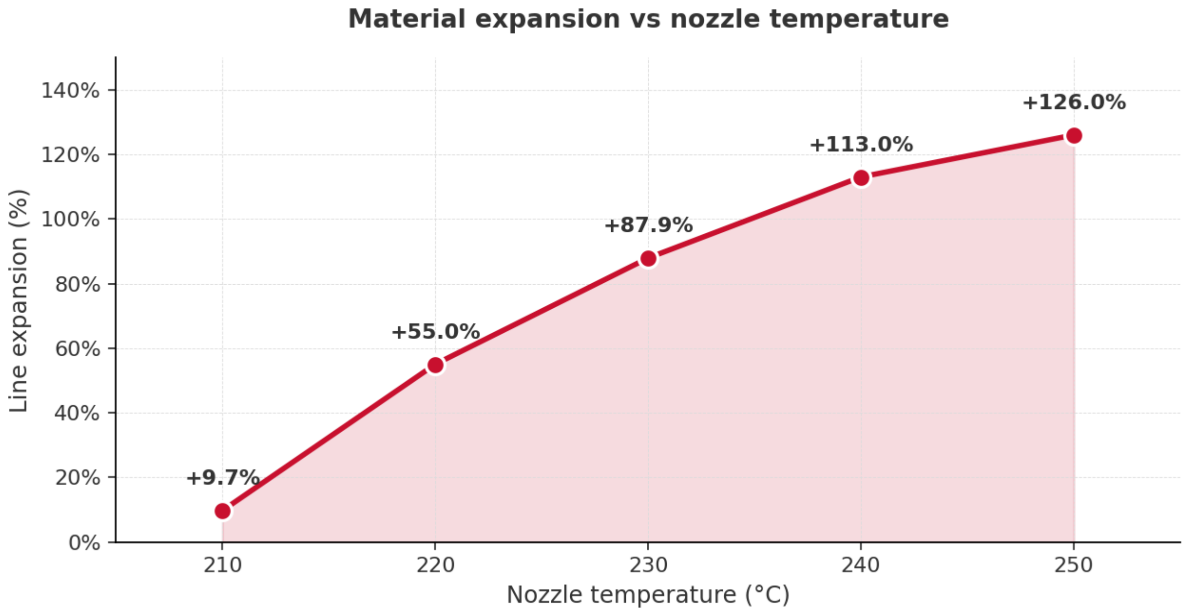

Figure 2 — Line expansion (%) versus nozzle temperature.

6.2 Recommended flow compensation

To preserve the geometric integrity of the part, the slicer flow must be reduced as the material expands. The rule of thumb is:

Recommended flow (%) ≈ *Target line width / Measured line width @ 100 % flow*

The values below are obtained directly from the measurements in §6.1 and produce a printed line width of 0.62 mm:

Nozzle temp.	Recommended flow	Effective line width	Use case
210 °C	91 %	0.62 mm	Almost no foaming — dense, firm parts
220 °C	64 %	0.62 mm	Slightly lighter parts, moderate softness
230 °C	53 %	0.62 mm	Recommended balanced setting
240 °C	46 %	0.62 mm	Lighter, softer parts
250 °C	44 %	0.62 mm	Maximum weight reduction

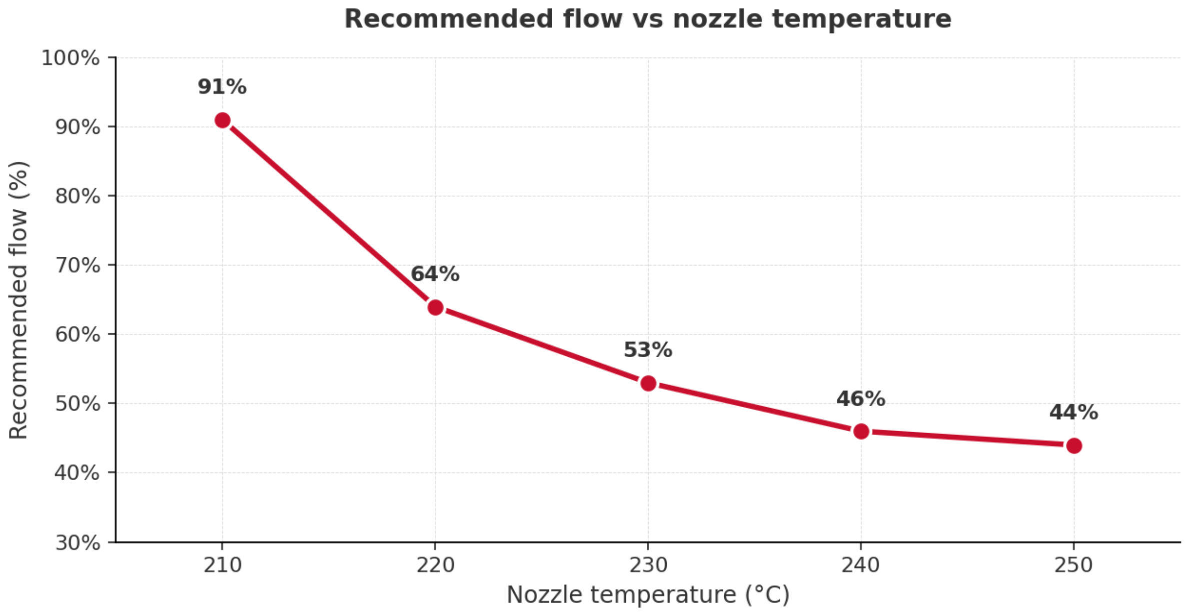

Figure 3 — Recommended flow (%) to preserve the 0.62 mm target line width across the printing range.

How to read these tables: pick the temperature that matches the desired part feel (firmer at 210–220 °C, softer and lighter at 240–250 °C) and use the matching flow value as the slicer extrusion multiplier.

7. Calibration Procedure

Each printer behaves differently. The values in §6 are a starting point — fine-tune them on your machine with the following short calibration.

Step 1 — Single-wall flow test

1. Slice a 30 × 30 × 20 mm hollow cube with a single perimeter, no top, no bottom, no infill.
2. Print at 230 °C with the flow set to 53 %.
3. Measure the actual wall thickness with a caliper at three different heights.
4. If the average is above 0.62 mm, reduce flow by 3 % and reprint. If below 0.62 mm, increase flow by 3 %.
5. Repeat until you obtain 0.60–0.64 mm — that is your reference flow at 230 °C.

Step 2 — Temperature sweep

1. Print the same hollow cube at 210, 220, 230, 240, and 250 °C — one cube per temperature.
2. For each temperature, measure wall thickness and weight.
3. Adjust flow at each temperature so the wall thickness stays at 0.62 mm. Use Table 6.2 as a starting point.

Step 3 — Retraction tuning

Foaming TPU strings more than rigid materials. Print a retraction tower with the flow already calibrated and adjust around the 0.4 mm reference value in 0.1 mm steps until stringing is acceptable. Travel speed should always be set as high as your machine allows (200–300 mm/s).

8. Troubleshooting

Symptom	Likely cause	Recommended fix
Excessive expansion / blobs / over-extrusion	Temperature too high or flow too high for the chosen temperature.	Lower nozzle temperature by 5 °C, or reduce flow by 3 %. Re-check wall thickness.
Cell collapse / surface dimples / poor foaming	Temperature above 250 °C — the foam structure breaks down.	Stay within the 210–250 °C printing window.
Under-extrusion / gaps between perimeters	Flow too low for the chosen temperature, or grip pressure too tight on a soft filament.	Increase flow by 3 % and/or release extruder grip slightly. Verify the filament feeds without buckling.
Stringing between parts	Travel speed too low, retraction too short, or moisture in the filament.	Raise travel speed to 200–300 mm/s, increase retraction in 0.1 mm steps, and dry the spool 4–6 h at 70 °C.
Bubbles, popping sounds while printing	Wet filament — water flashing to steam.	Stop the print, dry the spool 6 h at 70 °C, and resume.
Inconsistent color / patchy surface	Inconsistent foaming caused by uneven flow or temperature instability.	Verify nozzle thermistor accuracy. Lower the part-cooling fan (≤ 20 %) and keep print speed steady.
Part too soft / too rigid for the application	Wrong temperature target for the desired hardness.	Use Table 6.1 to pick a temperature: lower temp = firmer/denser part, higher temp = softer/lighter part.
Filament grinds in the extruder	Extruder grip too tight on a soft material.	Loosen the idler tension. Replace with a flexible-friendly extruder if grinding persists.
Poor first-layer adhesion	Bed too cold or unsuitable surface.	Raise bed to 55–60 °C, use PVA glue stick or adhesion spray, and increase first-layer line width by 10–20 %.
Layer separation in flexible parts	Cooling too aggressive between layers.	Lower cooling fan to 0–10 % and slightly increase nozzle temperature.

9. Disclaimer

The values provided in this guide should be considered indicative starting points. Actual results depend on printer model, slicer, environment and part geometry. The application, use and processing of the material are the responsibility of the end user. This document is updated as additional testing data becomes available.