Why do overhangs droop in 3D printing?

Asked 3D Content Team
Published October 24, 2026 · Last verified October 24, 2026

Abstract. Each overhang layer rests partly on air; past about 45° from vertical, soft plastic sags before it cools. Fix: full fan, slower overhangs, a cooler nozzle.

Keywords: overhangs, part-cooling, slicer-settings

1 Why do overhangs droop in 3D printing?

Each layer of an overhang sits partly on air, held up by the edge of the layer below. Raise3D puts the limit at 45 degrees from vertical: past that, too little of each line is supported, and the soft edge sags before it cools. The fixes: full part-cooling fan, slower overhangs, a cooler nozzle, thinner layers, then supports.

1.1 The number

Table 1. The number

Source	Overhang that prints without support
Raise3D support guide	Up to 45° from vertical; beyond that, add support
Prusa Knowledge Base	45 to 60°, depending on nozzle size and settings; possibly under 45° with a 0.25 mm nozzle; up to 75° on Prusa's Nextruder printers with 360° cooling
Bambu Lab PLA Basic data sheet	55° maximum overhang angle

The limits come from Raise3D's support guide, Prusa's "Modeling with 3D printing in mind" article and Bambu Lab's PLA Basic data sheet V3.0, and only Raise3D states that it measures from vertical.

1.2 Why do overhangs sag and curl?

The overhanging edge of each line has nothing under it, so only its own stiffness holds it up. PLA stays soft until it cools below its glass transition (the point where it turns rigid), 60 °C on Bambu's PLA Basic data sheet. If the edge is still soft when the nozzle comes back, it sags under its own weight. An edge can also curl upward as it cools, and Prusa notes the nozzle can then crash into it.

How much of each line hangs free depends on the angle. At 45 degrees from vertical, each layer steps out by one layer height (0.2 mm on a 0.20 mm profile). Past that, it steps out further. Slicers measure the free part against the line width: PrusaSlicer counts the overlap with the layer below (100% is a plain wall, 0% a bridge), while Bambu Studio counts the share hanging past it.

Each fix works on that edge: the fan hardens it sooner, a slower overhang gives it time to set, a cooler nozzle leaves less heat to remove, and a thinner layer steps out less at the same angle.

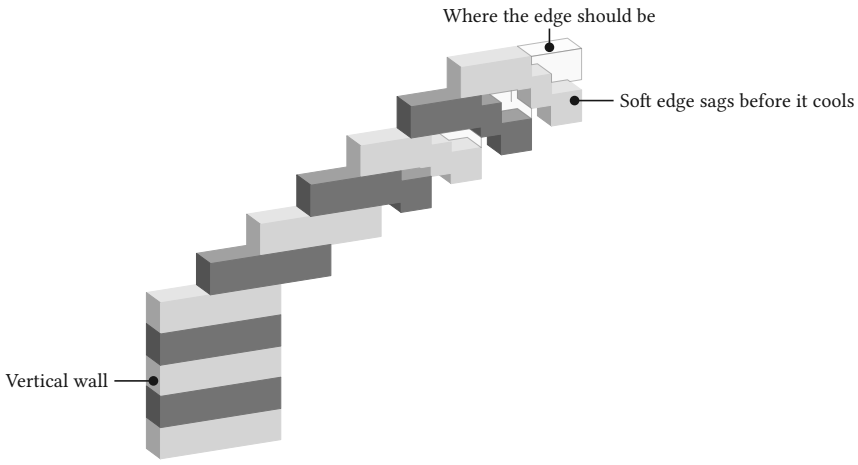


Figure 1. Each overhang layer rests only partly on the layer below, and the unsupported edge sags if it is still soft when the next line lands.

Table 2. How do you fix drooping overhangs with cooling settings?

Setting	PrusaSlicer 2.9.6, Prusament PLA and 0.20mm SPEED on the MK4S	Bambu Studio 02.08.04.61, PLA and 0.20mm Standard on the X1C	Cura 5.13.0, base definition
Fan on overhangs	Enable dynamic fan speeds: on, 100% at 0% and 25% overlap (program default: off)	Force cooling for overhang and bridge: on. Fan speed for overhang: 100%. Cooling overhang threshold: 50% for PLA, 95% in the base profile	No overhang-only fan. Bridge Fan Speed 100%, used only when Enable Bridge Settings is on (off by default)
Regular PLA fan	70 to 100%	100%	Fan Speed 100%
Overhang slowdown	Enable dynamic overhang speeds: on, 15, 25 and 50 mm/s at 0, 25 and 50% overlap, 80% of the external perimeter speed at 75% (program default: off)	Slow down for overhang: on, wall speed at 10%, then 50, 30, 10 and 10 mm/s at 25, 50, 75 and 100% overhang	Overhanging Wall Angle 90°, which means off. Overhanging Wall Speeds 100%
Support angle	Overhang threshold 35°, measured from horizontal (0, automatic, by default)	Threshold angle 30°	Support Overhang Angle 50° (0° supports every overhang, 90° none)

1.3 How do you fix drooping overhangs with cooling settings?

Run the part-cooling fan at full speed on overhang lines and slow those lines down. Each slicer has a setting for both:

The values are read from each slicer's source code and shipped profiles at the release tags named in the column headers.

Work through the fixes in this order, one at a time:

1. **Max the fan on overhangs.** Bambu Studio already runs PLA at 100%. For a filament with a lower regular fan, set Filament > Cooling > Cooling overhang threshold to 25% or 10% so the overhang fan starts on milder overhangs. In PrusaSlicer (Expert mode), set Filament Settings > Cooling > Dynamic fan speeds to 100 for the 50% and 75% overlap fields as well. Cura has no overhang-only fan, so keep Cooling > Fan Speed at 100%, its default. Fan basics: PLA fan speed and what the part cooling fan does.
2. **Slow the overhang walls.** PrusaSlicer: Print Settings > Speed > Dynamic overhang speed; check that it is on and lower the 25% and 50% overlap speeds. Bambu Studio (Advanced): Process > Speed > Slow down for overhang; lower the 25% and 50% fields. Cura: under Experimental, set Overhanging Wall Angle to the last angle that prints clean and Overhanging Wall Speeds to a list such as `[75, 50, 25]`, Cura's own example.
3. **Drop the nozzle 5 to 10 °C**, Simplify3D's step for overheating, and stay inside the spool's range (190 to 230 °C for Bambu PLA Basic). Bambu Studio prints that PLA at 220 °C, and Prusa's MK4S profile prints Prusament PLA at 225 °C. If layers start to split, go back up: Prusa warns that too much fan can weaken layer bonds.
4. **Use thinner layers** for that part: PrusaSlicer's 0.15mm SPEED or Bambu Studio's 0.12mm Fine instead of a 0.20 mm profile.
5. **Then add support or change the model.** Turn the part, add support from the angle that fails, or use chamfers instead of fillets on edges facing the bed. Prusa notes a bed-facing fillet becomes a very steep overhang.

1.4 How to check it on your own print

- **Print an overhang test** with marked angles on your normal profile. Note whether its angles count from vertical or from horizontal, then find the first angle whose underside turns rough or whose edge droops or lifts.
- **Compare all sides.** If one side fails earlier, the airflow is uneven: check the duct and the fan.
- **Reprint the test after each fix.** The first failing angle should move further from vertical.
- **Set the support angle to the last clean angle.** Cura's Support Overhang Angle supports every overhang at 0° and none at 90°, so enter your angle from vertical. PrusaSlicer's Overhang threshold is measured from the horizontal plane (90° is vertical), so enter 90 minus your angle from vertical. Bambu Studio's tooltip calls its Threshold angle a slope angle and names no reference, so check where supports appear in the preview.
- **If tiny parts also melt,** see small prints melting. With no part fan at all, see printing without a cooling fan.

References

[1] When and How to Use 3D Printed Support Structures, Raise3D Academy. https://www.raise3d.com/academy/when-and-how-to-use-3d-printed-support-structures/. Accessed October 24, 2026.

[2] Modeling with 3D printing in mind, Prusa Knowledge Base. https://help.prusa3d.com/article/modeling-with-3d-printing-in-mind_164135. Accessed October 24, 2026.

[3] Bambu Filament Technical Data Sheet V3.0, PLA Basic (Bambu Lab). https://wiki.bambulab.com/filament-acc/abs-asa-pc/bambu_pla_basic_technical_data_sheet.pdf. Accessed October 24, 2026.

[4] src/libslic3r/PrintConfig.cpp at version_2.9.6, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/src/libslic3r/PrintConfig.cpp. Accessed October 24, 2026.

[5] resources/profiles/PrusaResearch.ini at version_2.9.6, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/resources/profiles/PrusaResearch.ini. Accessed October 24, 2026.

[6] src/libslic3r/PrintConfig.cpp at v02.08.04.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.04.61/src/libslic3r/PrintConfig.cpp. Accessed October 24, 2026.

[7] src/slic3r/GUI/Tab.cpp at version_2.9.6, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/src/slic3r/GUI/Tab.cpp. Accessed October 24, 2026.

[8] src/slic3r/GUI/Tab.cpp at v02.08.04.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.04.61/src/slic3r/GUI/Tab.cpp. Accessed October 24, 2026.
[9] BBL filament profile fdm_filament_common.json at v02.08.04.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.04.61/resources/profiles/BBL/filament/fdm_filament_common.json. Accessed October 24, 2026.
[10] BBL filament profile fdm_filament_pla.json at v02.08.04.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.04.61/resources/profiles/BBL/filament/fdm_filament_pla.json. Accessed October 24, 2026.
[11] BBL process profile 0.20mm Standard @BBL X1C.json at v02.08.04.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.04.61/resources/profiles/BBL/process/0.20mm%20Standard%20%40BBL%20X1C.json. Accessed October 24, 2026.
[12] BBL process profile fdm_process_common.json at v02.08.04.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.04.61/resources/profiles/BBL/process/fdm_process_common.json. Accessed October 24, 2026.
[13] BBL process profile 0.12mm Fine @BBL X1C.json at v02.08.04.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.04.61/resources/profiles/BBL/process/0.12mm%20Fine%20%40BBL%20X1C.json. Accessed October 24, 2026.
[14] resources/definitions/fdmprinter.def.json at 5.13.0, Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.13.0/resources/definitions/fdmprinter.def.json. Accessed October 24, 2026.
[15] Overheating, Simplify3D Print Quality Troubleshooting Guide. https://www.simplify3d.com/resources/print-quality-troubleshooting/overheating/. Accessed October 24, 2026.
[16] Warping, Prusa Knowledge Base. https://help.prusa3d.com/article/warping_2011. Accessed October 24, 2026.
[17] Layer shifting, Prusa Knowledge Base. https://help.prusa3d.com/article/layer-shifting_2020. Accessed October 24, 2026.

Cite as: Asked 3D. “Why do overhangs droop in 3D printing?.” October 24, 2026. https://asked3d.com/answers/overhangs-drooping. Last verified October 24, 2026.