Why are my supports so hard to remove?

Asked 3D Content Team
Published November 7, 2026 · Last verified November 7, 2026

Abstract. The support's top is too close to the part or too solid. Profiles ship a 0.2 mm Top Z distance; Bambu Lab says 0 makes same-filament supports really hard to remove.

Keywords: supports, slicer-settings

Table 1. The number

Slicer and profile (0.4 mm nozzle, 0.2 mm layers)	Top Z distance	Top interface layers	Interface spacing or density	Interface pattern
Bambu Studio 02.09.01.70, 0.20mm Standard, Bambu Lab printers	0.2 mm	2	0.5 mm	Default
OrcaSlicer 2.5.0, 0.20mm Standard, generic Klipper, Marlin and RepRapFirmware printers	0.2 mm	2	0.5 mm	Default (Rectilinear when non-soluble)
PrusaSlicer 2.9.7, 0.20mm SPEED and STRUCTURAL, MK4S and CORE One	0.2 mm	3	0.2 mm	Default (Rectilinear when non-soluble)
UltiMaker Cura 5.14.0, Standard Quality, Creality printers	0.2 mm	4 (a 0.8 mm roof)	33.333% density	Grid
UltiMaker Cura 5.14.0, base definition (Custom FFF printer)	0.1 mm	Off (Enable Support Interface unticked)	100% once on	Concentric once on

1 Why are my supports so hard to remove?

The support's top is too close to your part or too solid, so it bonds instead of just touching. Bambu Studio, OrcaSlicer and PrusaSlicer ship a 0.2 mm Top Z distance for 0.2 mm layers; Bambu Lab says 0 makes same-filament supports "really hard to remove". Dense interface layers, stiff patterns and thick tree tips add grip.

1.1 The number

Each value comes from the profile files at a release tag, with every `inherits` chain followed: Bambu Studio v02.09.01.70, OrcaSlicer v2.5.0, PrusaSlicer 2.9.7 (`PrusaResearch.ini`) and UltiMaker Cura 5.14.0 (its base, Creality and Standard Quality files). Spacing is the gap between interface lines; the Bambu Studio, OrcaSlicer and PrusaSlicer tooltips say 0 means a solid interface.

1.2 Why does a missing gap make supports stick?

Because the gap is what lets them part. Bambu Lab's support guide says supports must come off after printing, "so there must be a gap", and Cura calls its Support Z Distance the clearance for removing supports. At 0, PrusaSlicer's tooltip says, the overhang's first layer is no longer printed with bridge flow and speed. It goes straight onto the support's dense top (the interface, or roof) and bonds there.

PrusaSlicer labels 0 "soluble" and 0.1 or 0.2 mm "detachable". With the same filament on both sides, Bambu Lab recommends about 0.2 mm and states the trade: a bigger gap removes more easily but leaves a lower-quality surface.

1.3 Why are my supports too strong?

Once the gap is right, the makers name these, in this order:

1. **Too many or too dense interface layers.** OrcaSlicer's wiki says more top interface layers are "harder to peel off"; Cura says a higher Support Roof Density makes supports "harder to remove". PrusaSlicer labels 2 layers "default" and 3 "heavy", yet its MK4S and CORE One profiles ship 3. A spacing of 0 prints the roof solid.
2. **A stiff interface pattern.** Bambu Lab says Grid "can be more difficult to detach" and Rectilinear Interlaced "may be harder to remove than standard rectilinear". Cura's Creality profiles ship Grid; OrcaSlicer says Concentric loops "peel off in one piece".
3. **A material that bonds to itself.** Prusa lists "Supports are hard to remove" among the drawbacks of PETG (glycol-modified polyethylene terephthalate) (why PETG supports are so hard to remove). For nylon and PVA (polyvinyl alcohol), Bambu Lab says to remove supports within 2 hours, before they absorb moisture and soften.

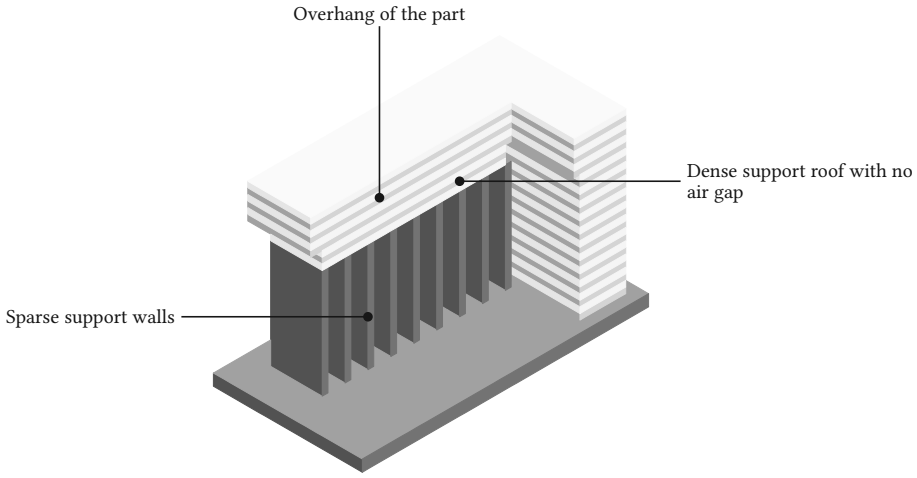


Figure 1. With no gap, the dense roof of the support is printed straight onto the overhang, and the support and part become one solid at the contact.

4. **A dense or walled base.** Cura gives Support Density the same "harder to remove" warning, and Prusa says the same of a sheath around the support.

1.4 How do you stop supports fusing to the part?

Check both contacts. On top, if Top Z distance reads 0 or 0.1 mm with the same filament for support and part, return it to the shipped 0.2 mm; values for other layer heights are in what support Z distance should be. OrcaSlicer rounds the gap to whole layers unless Independent support layer height is on.

On the sides, Prusa says a bigger XY separation shrinks the contact area and eases removal. Bambu Studio and OrcaSlicer ship 0.35 mm; PrusaSlicer's MK4S profiles use 80% of the external perimeter width. Cura uses twice the outer wall line width for Creality printers and 0.7 mm in its base definition.

For a flat underside with no gap, print the interface in another material: Bambu Lab says 0 works with a dedicated support filament (which interface material suits each filament).

1.5 Why are tree supports hard to remove?

Because tips, branch density and style add grip. Bambu Studio's tooltip says a thicker tip "sticks to the overhang better" but "is harder to remove". Tip diameter ships at 0.8 mm in Bambu Studio, OrcaSlicer and PrusaSlicer (Organic style only in the last two); Cura uses twice the support line width. Branch Density ships at 30% in OrcaSlicer, and in Cura with Support Roof on (10% with it off); both descriptions give higher values the same warning.

Bambu Lab adds that Tree Strong branches are "sometimes difficult to remove", as are larger branch diameters and smaller branch distances. How the types compare: tree vs normal supports.

1.6 How do you make supports easier to remove in Bambu Studio, OrcaSlicer, PrusaSlicer and Cura?

At 0.2 mm layers with one filament: Top Z distance 0.2 mm, 2 top interface layers, an interface spacing above 0, a Rectilinear or Concentric pattern and at least the default XY distance. Where each setting lives:

- **Bambu Studio:** Process > Support, with Advanced switched on. Advanced group: Top Z distance, Top interface layers, Interface pattern, Top interface spacing, Support/object xy distance. Tree Support group: Tip diameter, Branch diameter. Style is in the Support group.
- **OrcaSlicer:** Process > Support > Advanced: Top Z distance, Top interface layers, Interface pattern, Top interface spacing, Support/object XY distance. Tree supports group: Tip Diameter, Branch Density.
- **PrusaSlicer:** Print Settings > Support material, in Advanced or Expert mode. "Options for support material and raft": Top contact Z distance, Top interface layers, Interface pattern, Interface pattern spacing, XY separation between an object and its support. Organic supports group: Tip Diameter.
- **UltiMaker Cura:** Custom print settings > Support: Support Top Distance (under Support Z Distance), Support Roof Density, Support Roof Thickness, Support Roof Pattern, Support X/Y Distance, and for tree supports Tip Diameter and Branch Density. If a setting is hidden, type its name into the search box above the settings list.

1.7 How to check it on your own print

1. **Compare your profile with the table.** Change these first: a Top Z distance of 0 or 0.1 mm, an interface spacing of 0, 3 or more interface layers, a Grid pattern.
2. **Scrub the sliced preview.** At the top of a support there should be empty space before the overhang's first layer; if the overhang sits right on the support, the gap is 0.

3. **Print a small overhang test,** one change per print, and keep the first that snaps off cleanly. If the underside turns rough instead, see why prints are rough under supports.
4. **Remove supports promptly,** within 2 hours for nylon or PVA. Prusa warns that support pieces can be sharp and fly off in random directions, so wear eye protection (how to remove 3D print supports).

References

[1] Support, Bambu Lab Wiki. https://wiki.bambulab.com/en/software/bambu-studio/support. Accessed November 7, 2026.

[2] resources/profiles/BBL/process/0.20mm Standard @BBL X1C.json at v02.09.01.70, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.09.01.70/resources/profiles/BBL/process/0.20mm%20Standard%20%40BBL%20X1C.json. Accessed November 7, 2026.

[3] resources/profiles/BBL/process/fdm_process_common.json at v02.09.01.70 (the base every Bambu Lab 0.20mm Standard profile inherits), Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.09.01.70/resources/profiles/BBL/process/fdm_process_common.json. Accessed November 7, 2026.

[4] src/libslic3r/PrintConfig.cpp at v02.09.01.70, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.09.01.70/src/libslic3r/PrintConfig.cpp. Accessed November 7, 2026.

[5] src/slic3r/GUI/Tab.cpp at v02.09.01.70, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.09.01.70/src/slic3r/GUI/Tab.cpp. Accessed November 7, 2026.

[6] resources/profiles/Custom/process/fdm_process_common.json at v2.5.0, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.5.0/resources/profiles/Custom/process/fdm_process_common.json. Accessed November 7, 2026.

[7] resources/profiles/Custom/process/fdm_process_marlin_common.json at v2.5.0, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.5.0/resources/profiles/Custom/process/fdm_process_marlin_common.json. Accessed November 7, 2026.

[8] src/libslic3r/PrintConfig.cpp at v2.5.0, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.5.0/src/libslic3r/PrintConfig.cpp. Accessed November 7, 2026.

[9] src/slic3r/GUI/Tab.cpp at v2.5.0, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.5.0/src/slic3r/GUI/Tab.cpp. Accessed November 7, 2026.

[10] Support Advanced, OrcaSlicer Wiki. https://www.orcaslicer.com/wiki/print_settings/support/support_settings_advanced.html. Accessed November 7, 2026.

[11] Tree Support, OrcaSlicer Wiki. https://www.orcaslicer.com/wiki/print_settings/support/support_settings_tree.html. Accessed November 7, 2026.

[12] resources/profiles/PrusaResearch.ini at version_2.9.7, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.7/resources/profiles/PrusaResearch.ini. Accessed November 7, 2026.

[13] src/libslic3r/PrintConfig.cpp at version_2.9.7, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.7/src/libslic3r/PrintConfig.cpp. Accessed November 7, 2026.

[14] src/slic3r/GUI/Tab.cpp at version_2.9.7, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.7/src/slic3r/GUI/Tab.cpp. Accessed November 7, 2026.

[15] Support material, Prusa Knowledge Base. https://help.prusa3d.com/article/support-material_1698. Accessed November 7, 2026.

[16] PETG, Prusa Knowledge Base. https://help.prusa3d.com/article/petg_2059. Accessed November 7, 2026.

[17] resources/definitions/fdmprinter.def.json at 5.14.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.14.0/resources/definitions/fdmprinter.def.json. Accessed November 7, 2026.

[18] resources/definitions/creality_base.def.json at 5.14.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.14.0/resources/definitions/creality_base.def.json. Accessed November 7, 2026.

[19] resources/quality/creality/base/base_global_standard.inst.cfg at 5.14.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.14.0/resources/quality/creality/base/base_global_standard.inst.cfg. Accessed November 7, 2026.

Cite as: Asked 3D. "Why are my supports so hard to remove?." November 7, 2026. https://asked3d.com/answers/why-are-supports-hard-to-remove. Last verified November 7, 2026.