What is support interface in 3D printing?

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

Abstract. The dense layers between a support and the part. Bambu Studio ships 2 top and 2 bottom interface layers, PrusaSlicer 3 top; Cura's base profile leaves it off.

Keywords: supports, slicer-settings

1 What is support interface in 3D printing?

Support interface is the dense layers a slicer prints where a support meets the part: top (roof) layers under an overhang, bottom (floor) layers where a support stands on the model. A sparse base holds them up. Bambu Studio ships 2 top and 2 bottom layers, PrusaSlicer 3 top; UltiMaker Cura's base definition leaves it off.

1.1 The number

Every row comes from the shipped profile files at a release tag, with each `inherits` chain followed to the file that sets the value. Bambu Studio is read at v02.09.01.70, where all twelve 0.20mm Standard @BBL presets take these from `fdm_process_common.json`. OrcaSlicer is read at v2.5.0, `PrusaResearch.ini` at PrusaSlicer version_2.9.7 and UltiMaker Cura at 5.14.0. A value no profile sets falls back to the field default in the slicer's source.

1.2 Why does a support need an interface?

Because the base is kept sparse so it prints fast and breaks away: Prusa's knowledge base says to widen its line spacing "to make support removal easier". An overhang printed straight onto those lines would have to bridge from one to the next. The interface layers do that bridging instead, and Prusa warns that if the base lines are too far apart "these bridges will start to sag and have an imperfect connection with the model."

So the slicer stacks a few dense layers on the base and prints the part on that roof across a small gap, the top Z distance (how big it should be). Bambu Lab's wiki describes both ends: with 0 top interface layers, the lines over the supported area sag; with 3, the slicer builds "a stable and flat platform".

1.3 How many support interface layers should you use?

Two or three on top. Bambu Studio and OrcaSlicer's generic profiles ship 2 and PrusaSlicer 3, where its drop-down labels 2 "default" and 3 "heavy". OrcaSlicer's wiki gives the trade:

Table 1. The number

Slicer and profile (0.4 mm nozzle)	Top interface	Bottom interface	Interface spacing or density	Interface pattern
Bambu Studio 2.9.1, every 0.20mm Standard @BBL profile (X1 Carbon, P1S, A1, H2D and the rest)	2 layers	2 layers	Top interface spacing 0.5 mm	Default, which the tooltip calls Rectilinear Interlaced
OrcaSlicer 2.5.0, 0.20mm Standard for the Bambu Lab X1 Carbon and the generic Klipper printer	2 layers	2 layers	0.5 mm top and bottom	Default: Rectilinear, or Concentric on a soluble interface
OrcaSlicer 2.5.0, 0.20mm Standard for the Creality Ender-3	3 layers	Same as top	0.2 mm top, 0.5 mm bottom	Default
PrusaSlicer 2.9.7, 0.20mm SPEED and STRUCTURAL for the MK4S and CORE One	3 layers	0	Interface pattern spacing 0.2 mm	Default: Rectilinear, or Concentric on a soluble interface
UltiMaker Cura 5.14.0, base definition	Off; Support Roof Thickness 1 mm once enabled	Off; 1 mm once enabled	Support Interface Density 100%	Concentric
UltiMaker Cura 5.14.0, UltiMaker S5, AA 0.4, PLA, Normal	On, 2 × layer height	On, same	100%	Zig Zag
UltiMaker Cura 5.14.0, Creality Ender-3, Standard Quality (0.2 mm layers)	On, 4 × layer height (0.8 mm)	On, same	33.333%	Grid

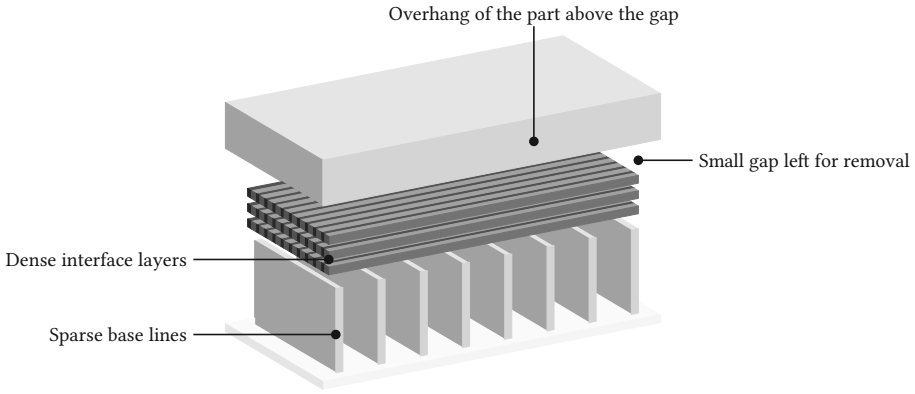


Figure 1. The sparse base carries a few dense interface layers, and the part prints on that roof across a small gap so the support can come away.

more layers make a flatter, better-supported surface but take longer to print and are harder to remove.

Bottom interface layers print only where a support stands on the model, not on the bed. Prusa ships 0 there and Bambu Lab 2. Bambu Lab warns that at 0 the support begins with a layer of bare lines, and says more bottom layers make it more stable.

1.4 Should I enable support interface?

Yes, for any supported surface you will see or that has to be flat. Bambu Studio, OrcaSlicer and PrusaSlicer have it on in every profile in the table. In UltiMaker Cura, check "Enable Support Interface": the base definition leaves it off, while the UltiMaker S5 and Creality profiles ship it enabled.

Turn it off (0 top interface layers, or the box unticked in Cura) only where the finish does not matter. OrcaSlicer's wiki says a support with no interface is the fastest and easiest to remove but leaves the roughest overhangs. If the underside is still rough with it on, see why a print is rough under supports.

1.5 What is support interface spacing, and how does it set density?

Spacing is the gap between neighboring interface lines in millimeters; 0 means a solid interface. Bambu Studio calls it "Top interface spacing", OrcaSlicer adds a separate "Bottom interface spacing", and PrusaSlicer names it "Interface pattern spacing".

The source code of all three turns it into a density the same way: line spacing ÷ (line spacing + your gap), so a gap of 0 is 100%. Bambu Studio's 0.5 mm gap is wider than its 0.42 mm support lines, so under half of each interface layer is plastic. PrusaSlicer's 0.2 mm gap is tighter.

UltiMaker Cura works the other way round: you set "Support Interface Density" as a percentage, and Cura derives the roof line distance as line width × 100 ÷ density, doubled for Grid. Creality's 33.333% Grid puts its 0.4 mm lines 2.4 mm apart.

1.6 What does the support interface pattern change?

How the lines are laid, and so how the roof peels:

- **Rectilinear:** straight lines across the base lines. Default uses it on a non-soluble interface in OrcaSlicer and PrusaSlicer.
- **Rectilinear Interlaced** (Bambu Studio, OrcaSlicer): rectilinear lines offset from layer to layer. Bambu Studio's tooltip names it as the pattern Default uses. Bambu Lab's wiki adds that it improves adhesion and smooths the underside but may be harder to remove.
- **Concentric:** loops following the outline that, per OrcaSlicer's wiki, peel off in one piece and leave no line ends. Default switches to it on a soluble interface in OrcaSlicer and PrusaSlicer; it is also Cura's base setting.
- **Grid:** crossed lines. Bambu Lab calls it very strong and stable for large flat surfaces but harder to detach. It ships in Creality's Cura profile.
- **Zig Zag** (Cura): what UltiMaker's own printer definition ships.

1.7 What is the difference between support interface and support base?

Bambu Lab's wiki puts it in two sentences: "Interface layers are the layers touching the object. The rest of the support body is the base." They differ in four ways:

- **Density.** PrusaSlicer's MK4S profiles leave a 2 mm gap between base lines against 0.2 mm in the interface, OrcaSlicer's generic profile 2.5 mm against 0.5 mm. Cura's base definition fills support at 15% and the interface at 100%.
- **Pattern.** Bambu Studio and OrcaSlicer offer Rectilinear, Rectilinear grid, Honeycomb, Lightning and Hollow for the base. Default leaves tree supports Hollow, with only the branch walls under the interface.
- **Direction.** In PrusaSlicer and OrcaSlicer's Default pattern, interface lines run at 90° to the base lines.
- **Filament.** Each has its own filament field, "Support/raft base" and "Support/raft interface" in Bambu Studio, so a breakaway or soluble filament can print just the interface (which interface filament).

1.8 Where is support interface in Bambu Studio?

In the Prepare tab, open the process settings, pick the Support tab and switch on the “Advanced” toggle at the top of the panel. The Advanced group then shows “Top interface layers”, “Bottom interface layers”, “Interface pattern” and “Top interface spacing”. “Global” sets them for the whole plate, “Objects” for one object. The interface filament, “Support/raft interface”, is in the Support filament group and shows without the toggle.

1.9 How to check it on your own print

1. **Open the settings.** Bambu Studio and OrcaSlicer: the Support tab’s Advanced group, as above. PrusaSlicer: Print Settings, Support material, “Options for support material and raft”, in Advanced or Expert mode. UltiMaker Cura: custom print settings, Support. Setting visibility Advanced shows “Enable Support Interface”; Expert adds thickness, density and pattern.
2. **Tick “Enable Support Interface” in Cura** if your printer’s profile leaves it off.
3. **Slice and check the preview.** Bambu Studio shows “Support interface” as its own line type, PrusaSlicer “Support material interface”. Step through the layers under the overhang: dense layers should sit on the sparse base, with a gap between them and the part.
4. **Print and look at the underside.** Sagging lines or a rough face: add a top interface layer (2 to 3) or lower the spacing (from Bambu Studio’s 0.5 mm toward PrusaSlicer’s 0.2 mm). Support that will not come off: drop a layer or raise the spacing, then check the support Z distance and why supports are hard to remove.
5. **Save it where it belongs:** in the process profile, or per object (Objects in Bambu Studio) when only one part needs it.

References

[1] resources/profiles/BBL/process/fdm_process_common.json at v02.09.01.70 (the base every 0.20mm Standard @BBL 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.

[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] 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.

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

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

[6] 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.

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

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

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

[10] 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.

[11] src/libslic3r/Support/SupportParameters.hpp at v2.5.0, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.5.0/src/libslic3r/Support/SupportParameters.hpp. Accessed November 7, 2026.

[12] 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.

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

[14] resources/profiles/Custom/process/fdm_process_common.json at v2.5.0 (inherited by 0.20mm Standard @MyKlipper), OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.5.0/resources/profiles/Custom/process/fdm_process_common.json. Accessed November 7, 2026.

[15] resources/profiles/Creality/process/0.20mm Standard @Creality Ender3 0.4.json at v2.5.0, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.5.0/resources/profiles/Creality/process/0.20mm%20Standard%20%40Creality%20Ender3%200.4.json. Accessed November 7, 2026.

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

[17] 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.

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

[19] 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.

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

[21] 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.

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

[23] 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.

[24] resources/definitions/ultimaker.def.json at 5.14.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.14.0/resources/definitions/ultimaker.def.json. Accessed November 7, 2026.
[25] resources/quality/ultimaker_s5/um_s5_aa0.4_pla_0.15mm.inst.cfg at 5.14.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.14.0/resources/quality/ultimaker_s5/um_s5_aa0.4_pla_0.15mm.inst.cfg. Accessed November 7, 2026.
[26] 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.
[27] 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.
[28] resources/setting_visibility/advanced.cfg at 5.14.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.14.0/resources/setting_visibility/advanced.cfg. Accessed November 7, 2026.
[29] resources/setting_visibility/expert.cfg at 5.14.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.14.0/resources/setting_visibility/expert.cfg. Accessed November 7, 2026.

Cite as: Asked 3D. "What is support interface in 3D printing?." November 7, 2026. https://asked3d.com/answers/what-is-support-interface. Last verified November 7, 2026.