# Why is my print rough under the supports?

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

**Abstract.** The underside is printed across a gap onto support interface lines and sags between them. Denser interface layers and a gap under the shipped 0.2 mm smooth it.

**Keywords:** supports, slicer-settings

**Table 1.** The number

| Setting | Bambu Studio | OrcaSlicer | PrusaSlicer | UltiMaker Cura | For a smoother underside |
|---|---|---|---|---|---|
| Top Z distance (Cura: Support Top Distance) | 0.2 mm | 0.2 mm | 0.2 mm | 0.1 mm | Smaller, but never 0 with one filament |
| Top interface layers (Cura: Enable Support Interface) | 2 | 2 | 3 | Off; a 1 mm roof once on | More |
| Interface spacing (Cura: Support Roof Density) | 0.5 mm | 0.5 mm | 0.2 mm | 100% | Tighter; 0 mm spacing is solid |
| Interface pattern | Default, which prints Rectilinear Interlaced | Default: Rectilinear | Default: Rectilinear | Concentric | Rectilinear Interlaced or Concentric |
| Support ironing | Off; shown only in Develop mode | Off | No such setting | No such setting | On, which forces a solid interface |

## 1 Why is my print rough under the supports?

The underside is printed across an air gap onto the support's interface lines, so it sags between them and keeps their ridges. A wider gap or sparser interface makes it rougher, and no gap bonds the support on. Bambu Studio, OrcaSlicer and PrusaSlicer ship a 0.2 mm gap; denser interface layers and a smaller gap smooth it.

### 1.1 The number

Values are read at release tags through each `inherits` chain: Bambu Studio v02.09.01.70 (0.20mm Standard @BBL X1C), OrcaSlicer v2.5.0 (0.20mm Standard @MyKlipper, plus field defaults), PrusaSlicer version_2.9.7 (0.20mm SPEED and STRUCTURAL @MK4S 0.4) and UltiMaker Cura 5.14.0 (the base definition). Default patterns are as each slicer resolves them for a non-soluble interface.

### 1.2 Why does the underside come out rough?

Because it is printed over air onto lines. Supports must come off, so Bambu Lab says "there must be a gap" between support and part: the Top Z distance. Unless that gap is 0, PrusaSlicer's tooltip says the part's first layer over it gets bridge flow and speed, the settings for a line printed across empty space. That bridge sags onto the top interface, the dense layers capping a support (what support interface is), and cools with one ridge per interface line.

How rough it gets depends on what sits under the bridge:

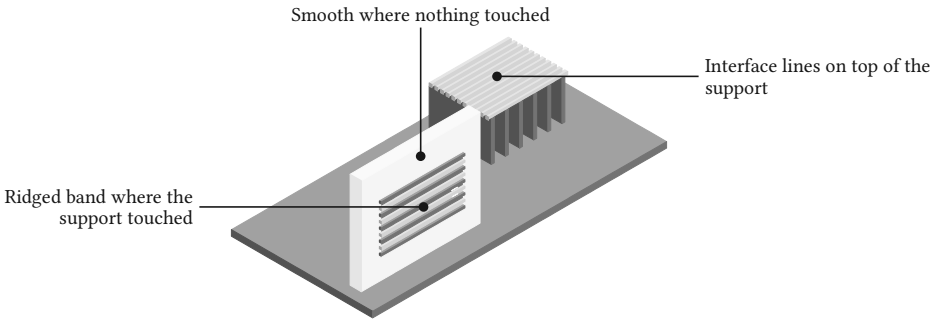



**Figure 1.** Where the support touched, the underside keeps a ridge for every interface line it sagged onto; the rest of the face stays smooth.

- **A sparse or thin interface.** Prusa says interface layers sag when the lines under them are too far apart. Bambu Lab says 0 top interface layers let the supported lines sag, while 3 build "a stable and flat platform".
- **Too wide a gap.** Bambu Lab says a bigger Top Z distance removes more easily but lowers interface quality; only a gap of 0 is "completely free from sagging".
- **Too small a gap with one filament.** The roof bonds to the part. Bambu Lab says a gap of 0 makes the support "really hard to remove", and Prusa suggests a scalpel or utility knife for pieces stuck to the model.

### 1.3 Do supports damage the print surface?

Yes. Every supported face carries marks, and with one filament for part and support you can only trade them against easy removal. Bambu Lab says that trade makes it "difficult to improve the printing quality of the support surface through parameter adjustments". Only a dedicated support filament at a gap of 0 gives a smooth face.

Supports that stand on the model mark it too. Prusa says "small imperfections may appear on the surface where supports started growing from."

The support style decides where the scars go. Prusa's knowledge base says Grid supports tend to leak out along the object's walls, Snug supports keep to the overhang, and Organic supports "do not scar the surface".

### 1.4 How do you keep support marks off the faces that matter?

1. **Orient the part** to cut overhangs, as Prusa suggests, and put a show face on the plate. Prusa's ironing guide says the bottom layer is typically even smoother than an ironed top.
2. **Block supports under a show face** with a support blocker or paint-on supports, placed where they meet the object (how to block supports in one area).
3. **Keep supports off top faces** with Support on build plate only. Where one must stand on the model, bottom interface layers protect that surface, says OrcaSlicer's wiki; PrusaSlicer's MK4S profiles ship none.
4. **Print the interface in a support filament** at a gap of 0 (which interface material suits your filament).

### 1.5 Which support interface settings give a smoother underside?

Add interface layers first, then tighten their spacing, change the pattern and narrow the gap. Make one change per test print, since each makes the support harder to remove.

1. **Top interface layers to 3.** Bambu Lab says more layers improve overhang quality for slightly more material. PrusaSlicer's MK4S profiles already ship 3, a count its drop-down labels "heavy". In UltiMaker Cura, tick Enable Support Interface first.
2. **Tighter interface spacing.** Bambu Studio and OrcaSlicer leave 0.5 mm between lines, PrusaSlicer 0.2 mm; 0 is solid. In Cura, raise Support Roof Density, which its description says gives better overhangs but harder removal.
3. **Another interface pattern.** In OrcaSlicer, try Rectilinear Interlaced: Bambu Lab says it gives smoother undersides but may be harder to remove. Or try Concentric, which OrcaSlicer's wiki says leaves no line ends on the surface. Bambu Studio's Default already interlaces; PrusaSlicer offers Concentric.
4. **A smaller Top Z distance.** Prusa's knowledge base says 50 to 75% of the layer height works well: 0.10 to 0.15 mm at 0.2 mm layers. Never 0 with one filament (what support Z distance should be).
5. **Normal supports under large flat overhangs,** where Bambu Lab says they usually leave a better surface than tree supports. Bambu Studio's profiles ship Type tree(auto): pick Style Tree Hybrid, which prints normal grid supports below big flat overhangs, or set Type to normal(auto) (tree vs normal supports).

If the support then will not come off, undo the last change (why supports are hard to remove).

### 1.6 Should I iron the support interface?

Only if lines still show after those steps and you accept a solid interface and a longer print. No slicer can iron the underside itself: Bambu Lab notes ironing reaches only top surfaces, because the nozzle faces down. Support ironing instead gives the top of the interface a second, low-flow pass at the same height, so the underside lands on a smoother interface.

- **OrcaSlicer 2.5.0:** Process > Support > Support ironing > "Ironing Support Interface", off by default. It needs top interface layers or a raft, prints the top interface solid and locks its spacing. Field defaults: Rectilinear pattern, flow 10% and line spacing 0.1 mm; the speed comes from the Ironing speed setting.
- **Bambu Studio 02.09.01.70:** "Enable ironing support interface" in the same group, off, and visible only after you tick Preferences > Other > Develop mode. Bambu Lab's profiles ship flow 10%, line spacing 0.15 mm and 30 mm/s. With a sparse interface it offers to make the interface solid or switch ironing off.
- **PrusaSlicer 2.9.7 and UltiMaker Cura 5.14.0** have no such setting.

A solid interface is the densest roof, and Cura's description ties density to harder removal. Bambu Lab says ironing adds print time at a slower speed. Its low, slow flow also risks a heat-creep clog in the hotend with PLA (polylactic acid), PETG (glycol-modified polyethylene terephthalate) and TPU (thermoplastic polyurethane). No maker says how much smoother the underside gets.

### 1.7 How to check it on your own print

1. **Compare your profile with the table.** The settings sit under Process > Support in Bambu Studio and OrcaSlicer (Advanced switched on), Print Settings > Support ma-

terial in PrusaSlicer (Advanced or Expert mode) and Support in UltiMaker Cura, whose settings search box finds hidden ones.

2. **Scrub the sliced preview** to just under the overhang: dense interface lines, an empty gap, then the part. Wide spaces between interface lines mean a sparse roof.
3. **Print a small flat overhang test,** one change at a time. Keep the first setting whose underside is smooth enough and whose support still comes off.
4. **Remove supports promptly:** within 2 hours for nylon and PVA (polyvinyl alcohol), says Bambu Lab, before damp support softens and leaves residue. Prusa warns that pieces can be sharp and fly off, 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] Ironing, Bambu Lab Wiki. https://wiki.bambulab.com/en/software/bambu-studio/parameter/ironing. Accessed November 7, 2026.

[3] 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.

[4] 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.

[5] 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.

[6] 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.

[7] 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.

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

[9] src/slic3r/GUI/Preferences.cpp at v02.09.01.70, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.09.01.70/src/slic3r/GUI/Preferences.cpp. 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/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.

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

[13] 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.

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

[15] Support Ironing, OrcaSlicer Wiki. https://www.orcaslicer.com/wiki/print_settings/support/support_settings_ironing.html. Accessed November 7, 2026.

[16] 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.

[17] 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.

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

[19] Ironing, Prusa Knowledge Base. https://help.prusa3d.com/article/ironing_177488. Accessed November 7, 2026.

[20] 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.


**Cite as:** Asked 3D. "Why is my print rough under the supports?." November 7, 2026. https://asked3d.com/answers/print-rough-under-supports. Last verified November 7, 2026.