How do you reduce support material?

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

Abstract. Lay the part on its largest flat face, then switch to tree supports: 1.87 g against 3.9 g for normal supports in one published Bambu Studio slice.

Keywords: supports, slicer-settings

Table 1. The number

Step	Bambu Studio	OrcaSlicer	PrusaSlicer	UltiMaker Cura
Orient automatically	“Auto orient” button in the top toolbar	“Auto orient” button, or Q	None for filament (FDM) printers	“Auto-Orientation” plugin from the Marketplace
Lay a chosen face on the bed	Lay on face (F)	Lay on Face (F)	Place on face (F)	Rotate tool (R), “Select face to align to the build plate”
Tree supports	Type “tree(auto)”, already the default once support is on	Type “Tree (auto)”; default “Normal (auto)”	Style “Organic”	Support Structure “Tree”; default “Normal”
Supports from the bed only	“On build plate only”, off	“On build plate only”, off	“Support on build plate only”, off	Support Placement “Touching Buildplate”; default “Everywhere”
Overhang threshold: shipped value, and the way that cuts support	Threshold angle 30°, from horizontal: lower it	Threshold angle 30°, from horizontal: lower it	Overhang threshold 35° (MK4S and CORE One, 0.20mm SPEED), from horizontal: lower it	Support Overhang Angle 50°, from vertical: raise it
Sparser support body	Base pattern spacing 2.5 mm: raise it	Base pattern spacing 2.5 mm: raise it	Pattern spacing 2 mm (MK4S and CORE One): raise it	Support Density 15% for Normal: lower it
Narrower normal supports	Style Snug instead of the default Grid	Style Snug instead of the default Grid	Style Snug, already shipped on MK4S and CORE One	Support Horizontal Expansion 0.8 mm: lower it

1 How do you reduce support material?

Lay the part on its largest flat face and turn it so as few surfaces as possible face down: less overhang means less support. Then switch to tree supports, which needed 1.87 g of support against 3.9 g for normal supports in one published Bambu Studio slice. Last, widen support spacing and block supports you do not need.

1.1 The number

Labels, keys and defaults are read from source and shipped profiles at release tags. Bambu Studio v02.09.01.70 values are from the base process profile that every Bambu Lab 0.20mm Standard preset inherits. OrcaSlicer v2.5.0 values are program defaults, which a printer profile can override. PrusaSlicer values are from version_2.9.7 and its bundled Prusa profiles, and Cura’s from 5.14.0 with Uranium 5.14.0. Cura’s expansion is its formula at a 0.4 mm support line; the Cura plugin is version 3.7.2.

1.2 Why does orientation change how much support a part needs?

Because the slicer supports every surface that faces down more steeply than its threshold, and orientation decides how much of the part faces down. Every support setting after that only changes how the slicer holds up that same overhang. Prusa’s knowledge base puts it first: reorienting or splitting a model can reduce overhangs “and consequently the amount of supports generated.”

Stand an L-shaped bracket on the end of its long leg and the short arm sticks out sideways at the top. Its whole underside is overhang, supported all the way from the bed. Lay the same bracket on its flat side and nothing overhangs.

Auto orientation in Bambu Studio and OrcaSlicer automates that choice. Bambu Lab’s wiki says it prefers a small overhang area, a large bottom area and hull, and little visible surface on supports, then picks the candidate with the lowest cost. The source code judges overhang by the process’s own Threshold angle. Bambu Lab warns that the tool sometimes picks a wrong orientation, mostly when a large overhang is really a bridge.

1.3 How do you orient a model to minimize supports?

1. **Put the largest flat face on the bed.** It needs no support, and Bambu Lab says a larger bottom area also sticks better. Bambu Studio and OrcaSlicer call the tool Lay on

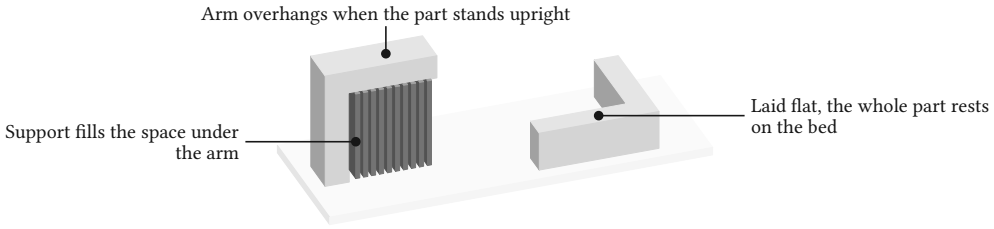


Figure 1. Stood upright, the arm of the L overhangs and needs support all the way down to the bed, while the same part laid on its largest flat face needs none.

face (F), PrusaSlicer Place on face (F). In UltiMaker Cura, open the Rotate tool (R) and use "Select face to align to the build plate", then click the face.

2. **Turn down-facing surfaces into walls or tops.** A surface facing up never needs support. One facing down needs it once it leans past your slicer's threshold, and Prusa puts the clean limit at 45 to 60° depending on nozzle and settings (what overhang angle needs support).
3. **Let short gaps bridge.** Prusa and Bambu Lab both say a short horizontal bridge prints without support, because both ends of each line rest on the part. Orient a slot or window so it spans between two walls instead of ending in mid-air.
4. **Keep the visible face off the supports.** Prusa notes a surface printed above supports looks rougher than one printed on the bed.
5. **Check strength before you commit.** Prusa compares prints to wood, stronger in one direction than another. A loaded part may need an orientation that costs more support (orientation for strength).
6. **Try the automatic tool, then check the preview.** Bambu Studio and OrcaSlicer have "Auto orient" in the top toolbar (Q in OrcaSlicer). In Cura, install the Marketplace plugin "Auto-Orientation", made "to reduce the need for support", and run "Calculate fast optimal printing orientation" from the Extensions menu. PrusaSlicer 2.9.7 has none for filament printers: its "Optimize orientation" opens only with a resin (SLA) printer selected.

1.4 Do tree supports use less material?

Yes, by the slicer makers' account and in one published slice. The University of Wisconsin-Madison ME Design Lab sliced a test part in Bambu Studio: normal supports took 3.9 g of support filament and tree supports 1.87 g, with the same print time. Bambu Studio's and OrcaSlicer's tooltips say the slim tree style (and in OrcaSlicer, organic) "save a lot of material", and Prusa calls Organic supports "cheap to print."

Keep normal supports under a large flat overhang, where Bambu Lab says they usually leave a better surface. Tree Hybrid makes that switch for you, using normal supports only there (tree vs normal supports). Where you keep normal supports, pick the Snug style over Grid: Bambu Studio, OrcaSlicer and PrusaSlicer all say snug towers "save material and reduce object scarring."

1.5 Which slicer settings print less support?

Six, each with a catch:

- **Support on build plate only.** Supports start only on the bed. Prusa calls it "a really quick way to get rid of excess supports mostly on organic shapes," but an overhang above the part itself then prints on air (what it does). With normal supports, the Wisconsin lab's test part came out "practically unusable" this way.
- **The overhang threshold.** Bambu Studio, OrcaSlicer and PrusaSlicer count from horizontal, so a lower value means less support. Cura counts from vertical, so there a higher value means less. Steeper unsupported overhangs droop if cooling cannot hold them (drooping overhangs).
- **Spacing or density.** Raise Base pattern spacing (Bambu Studio, OrcaSlicer) or Pattern spacing (PrusaSlicer), or lower Support Density (Cura). Bambu Lab's tree supports are hollow by default, so this mainly thins normal supports. Prusa warns that interface layers bridge over these lines and sag if the gaps grow too wide.
- **Cura's gradual infill and expansion.** Gradual Support Infill Steps (0 by default) halves the support density once per step further below the top. UltiMaker says lowering Support Horizontal Expansion "often means less material," though the expansion is what keeps support pillars from getting very thin.
- **Don't support bridges.** For normal supports this is off by default in Bambu Studio and OrcaSlicer and in Prusa's profiles. OrcaSlicer's tooltip says turning it on "decreases the amount of support required." Bambu Studio's adds that a bridge prints without support only if it is not very long.
- **Blockers, or enforcers only.** Block the support you do not need, or switch to enforcers only and paint support where it must go (how to block supports). Prusa's MK4S and CORE One profiles already ship that way. On one Wisconsin lab housing, automatic supports filled an internal pocket that no orientation avoided; painted tree supports left "very little wasted material."

1.6 When should you split the model or change the design?

When no orientation clears the overhangs. Prusa suggests splitting a model so each piece can be oriented with fewer supports, then gluing the pieces. Bambu Studio, OrcaSlicer and PrusaSlicer each have a Cut tool (C) that can add Plug, Dowel or Snap connectors to the cut faces (joining printed parts).

If you can edit the model, remove the overhang in the design. Prusa recommends a chamfer instead of a fillet on an edge facing the bed, because a fillet there makes a very steep overhang. More design moves are in printing overhangs without supports.

1.7 How to check it on your own print

1. **Slice the part as it came** and note the slicer's filament estimate.
2. **Orient it** with auto orient or lay on face, slice again and compare.
3. **Switch to tree supports,** or to Snug if you keep normal supports, and slice again.
4. **Change one setting at a time** (build plate only, threshold, spacing) and look under every overhang in the preview for any now printing on air.
5. **Block or paint** the supports that remain where the part does not need them.
6. **Print and inspect the overhangs.** If one droops or curls, put support back on that area with an enforcer instead of undoing every change.

References

[1] Auto orientation in Bambu Studio, Bambu Lab Wiki. https://wiki.bambulab.com/en/software/bambu-studio/auto-orientation. Accessed November 7, 2026.

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

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

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

[5] src/slic3r/GUI/GLCanvas3D.cpp at v02.09.01.70, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.09.01.70/src/slic3r/GUI/GLCanvas3D.cpp. Accessed November 7, 2026.

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

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

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

[9] 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.

[10] resources/profiles/BBL/process/fdm_process_common.json at v02.09.01.70 (the base every Bambu Lab process 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.

[11] 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.

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

[13] src/slic3r/GUI/GLCanvas3D.cpp at v2.5.0, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.5.0/src/slic3r/GUI/GLCanvas3D.cpp. Accessed November 7, 2026.

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

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

[16] src/slic3r/GUI/Gizmos/GLGizmoCut.cpp at v2.5.0, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.5.0/src/slic3r/GUI/Gizmos/GLGizmoCut.cpp. 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] 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.

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

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

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

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

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

[24] Modeling with 3D printing in mind, Prusa Knowledge Base. https://help.prusa3d.com/article/modeling-with-3d-printing-in-mind_164135. Accessed November 7, 2026.

[25] 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.

[26] plugins/Tools/RotateTool/RotateTool.qml at 5.14.0, Uranium (UltiMaker Cura's framework) on GitHub. https://github.com/Ultimaker/Uranium/blob/5.14.0/plugins/Tools/RotateTool/RotateTool.qml. Accessed November 7, 2026.

[27] plugins/Tools/RotateTool/RotateTool.py at 5.14.0, Uranium (UltiMaker Cura's framework) on GitHub. https://github.com/Ultimaker/Uranium/blob/5.14.0/plugins/Tools/RotateTool/RotateTool.py. Accessed November 7, 2026.
[28] UltiMaker Cura - Support settings, UltiMaker Support. https://support.ultimaker.com/articles/Knowledge/1667417606331. Accessed November 7, 2026.
[29] Auto-Orientation, UltiMaker Marketplace. https://marketplace.ultimaker.com/app/cura/plugins/nallath/OrientationPlugin. Accessed November 7, 2026.
[30] OrientationPlugin.py at v3.7.2, CuraOrientationPlugin on GitHub. https://github.com/nallath/CuraOrientationPlugin/blob/v3.7.2/OrientationPlugin.py. Accessed November 7, 2026.
[31] Supports, ME Design Lab, University of Wisconsin–Madison. https://medesignlab.me.wisc.edu/3d-printing/supports/. Accessed November 7, 2026.

Cite as: Asked 3D. "How do you reduce support material?." November 7, 2026. https://asked3d.com/answers/reduce-support-material. Last verified November 7, 2026.