Should the first layer bed temperature be different from the rest?

Asked 3D Content Team
Published October 15, 2026 · Last verified October 15, 2026

Abstract. Most slicer defaults use one bed temperature for all layers. PrusaSlicer starts PETG at 85 °C then 90 °C; OrcaSlicer starts ABS at 105 °C then 100 °C.

Keywords: slicer-settings, bed-adhesion, petg, abs

1 Should the first layer bed temperature be different from the rest?

Usually not. Bambu Studio and Cura defaults keep one bed temperature for every layer of PLA, PETG and ABS. PrusaSlicer's MK4S profiles start PETG at 85 °C and raise it to 90 °C, and OrcaSlicer starts ABS at 105 °C and drops to 100 °C. To split them, set the first-layer and other-layers bed fields separately.

1.1 The number

Values are read from the profile files in each slicer's GitHub repository, with inheritance resolved: Bambu Studio's BBL filament profiles, OrcaSlicer's filament library, Prusa Research's 2.5.11 profile bundle and UltiMaker's fdm_materials files. The last two rows come from the Prusa PETG article and Bambu's HueForge guide. PLA is polylactic acid, PETG is glycol-modified polyethylene terephthalate, ABS is acrylonitrile butadiene styrene and PEI is polyetherimide.

Cura fills its initial-layer fields from the main values unless you change them. Prusa's article and its own slicer profiles disagree on the PETG nozzle: 230 °C then 240 °C in the article, 250 °C throughout in the Prusament PETG profile.

1.2 Where to set a different first-layer bed temperature

Every slicer stores the bed pair in the filament or material profile, not in the printer profile.

- **Bambu Studio:** open the filament's settings, Filament tab, Print temperature group. Each plate type has its own row (Cool Plate, Engineering Plate, Smooth PEI Plate / High Temp Plate, Textured PEI Plate, Bambu Cool Plate SuperTack) with two boxes, **Initial layer** and **Other layers**. The Nozzle row in the same group works the same way. Edit the row for your plate.
- **OrcaSlicer:** Filament tab, Bed temperature group, one row per plate, with **First layer** and **Other layers** boxes.

Table 1. The number

Slicer and default profile	Material	Bed, first layer	Bed, other layers	Nozzle, first layer	Nozzle, other layers
Bambu Studio, Bambu PLA Basic, X1C or P1S, Textured PEI	PLA	55 °C	55 °C	220 °C	220 °C
Bambu Studio, Bambu PLA Basic, P1P, Textured PEI	PLA	65 °C	65 °C	220 °C	220 °C
Bambu Studio, Bambu PETG HF, Textured PEI	PETG	70 °C	70 °C	230 °C	245 °C
Bambu Studio, Bambu ABS, Textured PEI	ABS	90 °C	90 °C	260 °C	270 °C
OrcaSlicer, Generic PLA, Textured PEI	PLA	55 °C	55 °C	220 °C	220 °C
OrcaSlicer, Generic PETG, Textured PEI	PETG	80 °C	80 °C	255 °C	255 °C
OrcaSlicer, Generic ABS, Textured PEI	ABS	105 °C	100 °C	260 °C	260 °C
PrusaSlicer, Generic PLA @MK4S	PLA	60 °C	60 °C	230 °C	220 °C
PrusaSlicer, Prusament PETG @MK4S	PETG	85 °C	90 °C	250 °C	250 °C
PrusaSlicer, Prusament PETG @COREONE	PETG	85 °C	85 °C	250 °C	250 °C
PrusaSlicer, Generic ABS @MK4S	ABS	100 °C	110 °C	260 °C	260 °C
PrusaSlicer, Generic ABS @COREONE	ABS	110 °C	110 °C	260 °C	260 °C
Cura, generic PLA	PLA	60 °C	60 °C	200 °C	200 °C
Cura, generic PETG	PETG	85 °C	85 °C	240 °C	240 °C
Cura, generic ABS	ABS	80 °C	80 °C	245 °C	245 °C
Prusa Knowledge Base, PETG article	PETG	85 °C	90 °C	230 °C	240 °C
Bambu Lab Wiki, HueForge PLA, P1P or A1 mini, Textured PEI	PLA	70 °C	60 °C	not stated	not stated

The nozzle pair sits in the Print temperature group above it.

- **PrusaSlicer:** Filament Settings, Filament page, Temperature group. The Bed row holds **First layer** and **Other layers**, and the Nozzle row above it holds the same pair.
- **Cura:** the Material section of the print settings, **Build Plate Temperature Initial Layer** next to **Build Plate Temperature**. It appears once setting visibility is Advanced or higher. Until you type a number, it copies Build Plate Temperature. The nozzle pair is **Printing Temperature Initial Layer** and **Printing Temperature**. Don't confuse it with Initial Printing Temperature, the point at which printing may start while the nozzle is still heating.

Save the change as your own filament preset (in Cura, a custom profile). Step-by-step profile editing is on the changing bed temperature in the slicer page.

1.3 PLA bed temperature after the first layer

The same as the first layer, in every default PLA profile in the table: 55 °C in Bambu Studio on the X1C and P1S and in OrcaSlicer, 65 °C on Bambu's open-frame P1P, and 60 °C in PrusaSlicer and Cura. The only split comes from Bambu's HueForge guide. On the open-frame P1P and A1 mini with a Textured PEI plate, it runs 70 °C for the first layer, then 60 °C. On the enclosed X1C and P1S, the same guide keeps 55 °C throughout. The full PLA range by plate is on the PLA bed temperature page.

1.4 PETG and ABS initial layer settings

For PETG, the defaults split three ways. Bambu Studio holds the bed at 70 °C but starts the nozzle at 230 °C before 245 °C. Prusa's MK4S profile starts the bed cooler (85 °C) and raises it to 90 °C, while its enclosed CORE One profile keeps 85 °C. OrcaSlicer and Cura keep both values flat. More PETG numbers are on the PETG bed temperature page.

For ABS, the profiles disagree on direction. OrcaSlicer runs the first layer 5 °C hotter (105 °C, then 100 °C). PrusaSlicer's MK4S profile runs it 10 °C cooler (100 °C, then 110 °C), and its CORE One profile holds 110 °C. Bambu Studio keeps 90 °C on the bed and starts the nozzle at 260 °C before 270 °C. None of these profiles explains its split, so treat the direction as a profile choice, not a rule.

1.5 Why slicers print the first layer hotter

Heat makes the first line flow into the plate. The Slic3r manual, the documentation of the slicer PrusaSlicer grew from, says the nozzle and a heated bed "can be made hotter for the first layer, thus decreasing the viscosity of the material." Its rule of thumb is 5° extra. It adds that a hotter first layer makes no sense with the part-cooling fan still running. Prusa's own fix for a new material that won't stick is 5 to 10 °C more bed heat.

The switch happens at the start of layer 2. PrusaSlicer's start sequence waits for the first-layer bed temperature before printing. At the move to layer 2, it writes an M140 command with the other-layers value. Marlin's documentation says M140 sets the new target and continues without waiting, so layer 2 starts printing while the bed is still changing temperature. Bambu Studio's code makes the same switch at the same point.

1.6 How to check it on your own print

- **Read the G-code.** Open the exported file in a text editor and search for M140. In a PrusaSlicer file, the layer-2 line carries the comment "set bed temperature" and the other-layers value. If that line is missing, your two bed values match. PrusaSlicer users may need to turn off binary G-code in the printer settings first to get a readable text file.
- **Match the plate in Bambu Studio and OrcaSlicer.** Temperatures are stored per plate type, so the wrong plate prints the wrong numbers. Bambu Lab also notes that on a Textured PEI plate the printer lowers the nozzle by 0.04 mm after bed leveling. A wrong plate choice also loses that offset.
- **Raise the first-layer bed value when adhesion fails.** Prusa's range is 5 to 10 °C more. Put it in the first-layer field alone and the rest of the print keeps the profile's value. On an open-frame printer with PLA on textured PEI, Bambu's 70 °C then 60 °C is a published starting point.

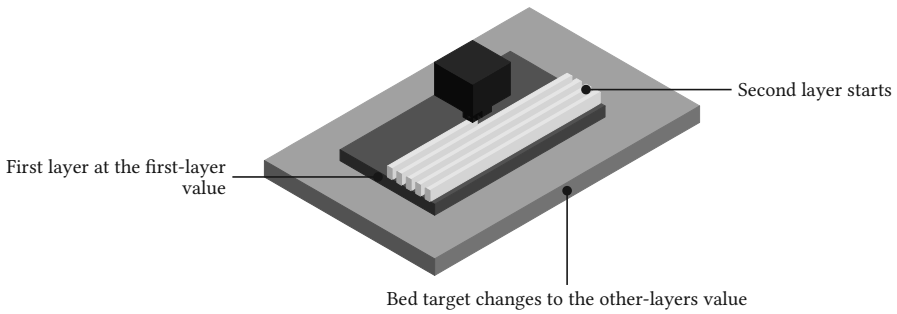


Figure 1. The bed holds the first-layer value until the second layer starts, when the slicer sends the other-layers value and printing carries on while the bed changes temperature.

- **If PETG sticks too hard, change the surface.** For PETG on smooth PEI, Prusa says to apply glue stick first, or to switch to a textured or satin sheet.

References

[1] Bambu Studio filament profile fdm_filament_pla.json (BBL), GitHub. https://github.com/bambulab/BambuStudio/blob/master/resources/profiles/BBL/filament/fdm_filament_pla.json. Accessed October 15, 2026.

[2] Bambu Studio filament profile Bambu PLA Basic @BBL P1P.json, GitHub. https://github.com/bambulab/BambuStudio/blob/master/resources/profiles/BBL/filament/P1P/Bambu%20PLA%20Basic%20%40BBL%20P1P.json. Accessed October 15, 2026.

[3] Bambu Studio filament profile Bambu PETG HF @base.json, GitHub. https://github.com/bambulab/BambuStudio/blob/master/resources/profiles/BBL/filament/Bambu%20PETG%20HF%20%40base.json. Accessed October 15, 2026.

[4] Bambu Studio filament profile fdm_filament_abs.json (BBL), GitHub. https://github.com/bambulab/BambuStudio/blob/master/resources/profiles/BBL/filament/fdm_filament_abs.json. Accessed October 15, 2026.

[5] Bambu Studio source, filament settings tab (Tab.cpp), GitHub. https://github.com/bambulab/BambuStudio/blob/master/src/slic3r/GUI/Tab.cpp. Accessed October 15, 2026.

[6] OrcaSlicer filament library base profiles (fdm_filament_pla, fdm_filament_pet, fdm_filament_abs), GitHub. https://github.com/OrcaSlicer/OrcaSlicer/tree/main/resources/profiles/OrcaFilamentLibrary/filament/base. Accessed October 15, 2026.

[7] OrcaSlicer source, filament settings tab (Tab.cpp), GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/main/src/slic3r/GUI/Tab.cpp. Accessed October 15, 2026.

[8] PrusaSlicer Prusa Research profile bundle 2.5.11 (PrusaResearch.ini), GitHub. https://github.com/prusa3d/PrusaSlicer-settings-prusa-fff/blob/main/PrusaResearch/2.5.11.ini. Accessed October 15, 2026.

[9] PrusaSlicer 2.9.6 source, filament settings tab (Tab.cpp), GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/src/slic3r/GUI/Tab.cpp. Accessed October 15, 2026.

[10] PrusaSlicer 2.9.6 source, G-code generator (GCode.cpp), GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/src/libslic3r/GCode.cpp. Accessed October 15, 2026.

[11] Cura printer setting definitions (fdmprinter.def.json), GitHub. https://github.com/Ultimaker/Cura/blob/main/resources/definitions/fdmprinter.def.json. Accessed October 15, 2026.

[12] Cura setting visibility preset, Advanced (advanced.cfg), GitHub. https://github.com/Ultimaker/Cura/blob/main/resources/setting_visibility/advanced.cfg. Accessed October 15, 2026.

[13] UltiMaker fdm_materials: generic_pla, generic_petg and generic_abs material files, GitHub. https://github.com/Ultimaker/fdm_materials. Accessed October 15, 2026.

[14] PLA, Prusa Knowledge Base. https://help.prusa3d.com/article/pla_2062. Accessed October 15, 2026.

[15] PETG, Prusa Knowledge Base. https://help.prusa3d.com/article/petg_2059. Accessed October 15, 2026.

[16] First layer issues, Prusa Knowledge Base. https://help.prusa3d.com/article/first-layer-issues_1804. Accessed October 15, 2026.

[17] HueForge printing guide, recommended parameter settings, Bambu Lab Wiki. https://wiki.bambulab.com/en/filament-acc/filament/hueforge-printing-guide. Accessed October 15, 2026.

[18] First layer not sticking: causes and solutions, Bambu Lab Wiki. https://wiki.bambulab.com/en/knowledge-sharing/first-layer-not-sticking. Accessed October 15, 2026.

[19] First Layer, Slic3r Manual. https://manual.slic3r.org/first-print/first-layer. Accessed October 15, 2026.

[20] M140: Set Bed Temperature, Marlin Firmware documentation. https://marlinfw.org/docs/gcode/M140.html. Accessed October 15, 2026.

Cite as: Asked 3D. "Should the first layer bed temperature be different from the rest?." October 15, 2026. https://asked3d.com/answers/first-layer-bed-temperature. Last verified October 15, 2026.