# Why are my small prints melting and blobby?

Asked 3D Content Team
Published October 24, 2026 · Last verified October 24, 2026

**Abstract.** Small layers print in seconds, so the next lands before the last one cools. Set a minimum layer time (4 to 12 s in stock PLA profiles) and keep the fan at max.

**Keywords:** part-cooling, slicer-settings, pla

**Table 1.** The number

| Slicer and PLA profile (0.4 mm nozzle) | Slows any layer shorter than | Slowest speed it drops to | Part fan, long layers to short layers | Fan starts ramping up on layers shorter than |
|---|---|---|---|---|
| PrusaSlicer 2.9.6, Prusament PLA (MK3S, MINI) | 12 s | 15 mm/s | 100% | 100 s |
| PrusaSlicer 2.9.6, Prusament PLA (MK4) | 8 s | 15 mm/s | 100% | 100 s |
| PrusaSlicer 2.9.6, Prusament PLA (MK4S) | 6 s | 20 mm/s | 70 to 100% | 17 s |
| PrusaSlicer 2.9.6, Prusament PLA (CORE One) | 8 s | 20 mm/s | 85 to 100% | 40 s |
| Bambu Studio 2.08.02.61, Bambu PLA Basic (X1 Carbon, P1S) | 4 s | 20 mm/s | 100% | 100 s |
| Bambu Studio 2.08.02.61, Bambu PLA Basic (A1, A1 mini) | 6 s | 20 mm/s | 60 to 80% | 80 s |
| OrcaSlicer 2.4.2, Generic PLA | 4 s | 10 mm/s | 100% | 100 s |
| Cura 5.13.0, built-in default | 5 s | 10 mm/s | 100% | 10 s |
| Cura 5.13.0, Creality printers | 10 s | 10 mm/s | 100% | 10 s |

## 1 Why are my small prints melting and blobby?

Each layer of a small part prints in seconds, so the nozzle returns before the plastic below has set, and the top slumps. Fix it with a minimum layer time: stock PLA profiles make every layer take at least 4 to 12 seconds. Keep the part fan at maximum for PLA. Printing several copies at once also helps.

### 1.1 The number

The values are read from PrusaSlicer's PrusaResearch.ini, Bambu Studio's BBL filament profiles, OrcaSlicer's OrcaFilamentLibrary and Cura's fdmprinter.def.json and creality_base.def.json at the release tags shown, following each profile's `inherits` chain to the file that sets the value.

On layers longer than the last column the fan runs at the lower speed, and it climbs to the higher one as layer time falls to the slowdown time.

### 1.2 Why do small prints melt?

Each layer needs time away from the hot nozzle to cool and stiffen before the next one lands. A large part gets that time because the nozzle spends most of each layer elsewhere. On a pin, a thin tower or the tip of a spike, a layer is one small loop. The nozzle returns while that loop is still soft and adds heat faster than the fan removes it.

The column softens and each new layer pushes it around until the top rounds over into a blob. Simplify3D's troubleshooting guide singles out very small parts whose layers take only a few seconds, and Bambu Lab's wiki names sharp tips among the details that need the most cooling.

### 1.3 What does insufficient cooling look like on a 3D print?

Soft, rounded or curled plastic wherever a layer or line had to set fast:

- **Melted, rounded tips** on spikes, pins, horns and the tops of cones, where each layer gets smaller and faster.
- **A glossy, smeared or slumped top** on a small part that should be flat and crisp.
- **Curled-up overhang edges and corners.** Simplify3D's guide says cooling each layer fast enough to solidify before it can deform prevents curling.
- **Drooping bridges and overhangs** (why overhangs droop).

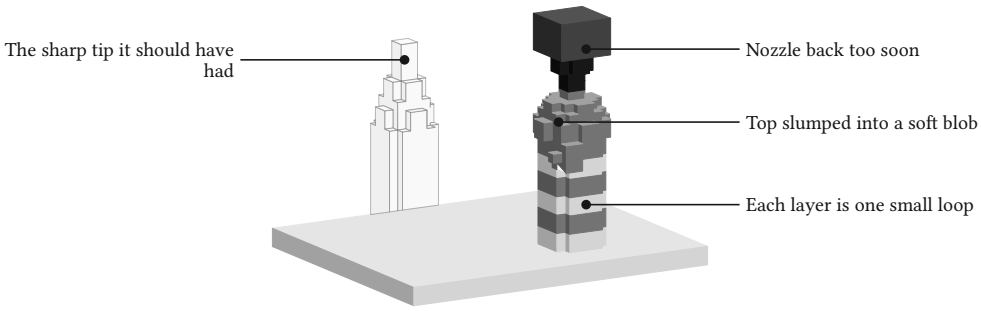



**Figure 1.** On a small spike each layer finishes in seconds, so the nozzle comes back before the plastic below has cooled and the tip slumps into a blob instead of a point.

If only small parts or the narrow top of a larger print deform, the layers are too short. If bridges and overhangs sag on big parts too, check the fan and its duct first.

### 1.4 How do you set minimum layer time for small parts?

Set the layer time below which the slicer slows down, and the slowest speed it may drop to:

- **PrusaSlicer** (Expert mode): Filament Settings > Cooling. Tick "Enable auto cooling", then under Cooling thresholds set "Slow down if layer print time is below" and "Min print speed".
- **Bambu Studio and OrcaSlicer:** the filament's settings > Cooling > Part cooling fan. The minimum layer time is the "Layer time" box on the "Max fan speed threshold" line, and "Min print speed" (Advanced mode) sets the floor. Tick "Slow printing down for better layer cooling", or the slicer only raises the fan. Leave "Don't slow down outer walls" off, because a pin is mostly outer wall.
- **Cura:** Print Settings > Custom > Cooling (Advanced visibility): "Minimum Layer Time" and "Minimum Speed". "Lift Head" moves the nozzle off the part to wait out the remaining time once Minimum Speed is reached.

Start from the table and raise the time if parts still slump: PrusaSlicer's MK3S profile uses 12 s, and Simplify3D's guide gives 15 s as its example. PrusaSlicer sets longer times for bigger nozzles, up to 20 s in its 0.8 mm nozzle PLA profiles.

The slowdown stops at the minimum speed: in Bambu Lab's wiki example, a layer that still takes under 4 s at 20 mm/s prints at 20 mm/s and no slower. Lowering that floor lets a tiny part slow further, but Prusa warns that very slow printing over long stretches can cause heat creep.

### 1.5 Why does printing several copies at once help?

The slicer times the whole layer, not one object, so each copy cools while the nozzle prints the other and nothing has to slow down. Simplify3D recommends printing two objects at once for exactly this, or importing a second object to print alongside. If you need only one part, that second object can be a plain cylinder or block: a throwaway cooling tower.

It works only when:

- **The extra object is at least as tall as the small part.** Once it ends, the layers get short again.
- **The plate prints layer by layer.** Leave "Complete individual objects" off in PrusaSlicer, set "Print sequence" to "By layer" in Bambu Studio and OrcaSlicer, and set "Print Sequence" to "All at Once" in Cura. Object-by-object printing gives each copy its own short layers.

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

1. **Read the layer times in the preview.** PrusaSlicer's "Layer time (linear)" view and the "Layer Time" view in Bambu Studio and OrcaSlicer color each layer by print time. Check the layers where the part goes wrong.
2. **Confirm the part fan spins.** Send `M106 S255` from the printer's console for full speed and `M107` to stop it; Marlin and Klipper both accept these. A fan that stays still after the first layer, or a cracked or misaimed duct, causes these symptoms on every part (what the part cooling fan does, fan speed for PLA).
3. **Set or raise the minimum layer time** as above, reslice, and check that the small part's layers now show longer times.
4. **Add a copy or a cooling tower** if the part is too small to slow down enough at the minimum speed.
5. **Lower the nozzle temperature by 5 to 10 °C,** as Simplify3D suggests, but stay inside the filament's range. Bambu Studio's PLA profiles print at 220 °C and list a recommended range of 190 to 240 °C. In Cura, "Small Layer Printing Temperature" drops the temperature only on layers slowed by Minimum Layer Time. It defaults to the printing temperature, so it does nothing until you set it lower.
6. **If the tip still melts at the minimum speed,** lower "Min print speed" ("Minimum Speed" in Cura) a little, or turn on Cura's "Lift Head".

## References

[1] resources/profiles/PrusaResearch.ini at version_2.9.6, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/resources/profiles/PrusaResearch.ini. Accessed October 24, 2026.

[2] src/libslic3r/PrintConfig.cpp at version_2.9.6, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/src/libslic3r/PrintConfig.cpp. Accessed October 24, 2026.
[3] src/slic3r/GUI/Tab.cpp at version_2.9.6, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/src/slic3r/GUI/Tab.cpp. Accessed October 24, 2026.
[4] src/slic3r/GUI/GCodeViewer.cpp at version_2.9.6, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/src/slic3r/GUI/GCodeViewer.cpp. Accessed October 24, 2026.
[5] Cooling, Prusa Knowledge Base. https://help.prusa3d.com/article/cooling_127569. Accessed October 24, 2026.
[6] BBL filament profile fdm_filament_common.json at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/resources/profiles/BBL/filament/fdm_filament_common.json. Accessed October 24, 2026.
[7] BBL filament profile fdm_filament_pla.json at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/resources/profiles/BBL/filament/fdm_filament_pla.json. Accessed October 24, 2026.
[8] BBL filament profile Bambu PLA Basic @base.json at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/resources/profiles/BBL/filament/Bambu%20PLA%20Basic%20%40base.json. Accessed October 24, 2026.
[9] BBL filament profile Bambu PLA Basic @BBL X1C.json at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/resources/profiles/BBL/filament/Bambu%20PLA%20Basic%20%40BBL%20X1C.json. Accessed October 24, 2026.
[10] BBL filament profile Bambu PLA Basic @BBL P1S 0.4 nozzle.json at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/resources/profiles/BBL/filament/Bambu%20PLA%20Basic%20%40BBL%20P1S%200.4%20nozzle.json. Accessed October 24, 2026.
[11] BBL filament profile Bambu PLA Basic @BBL A1.json at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/resources/profiles/BBL/filament/Bambu%20PLA%20Basic%20%40BBL%20A1.json. Accessed October 24, 2026.
[12] BBL filament profile Bambu PLA Basic @BBL A1M.json at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/resources/profiles/BBL/filament/Bambu%20PLA%20Basic%20%40BBL%20A1M.json. Accessed October 24, 2026.
[13] src/libslic3r/PrintConfig.cpp at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/src/libslic3r/PrintConfig.cpp. Accessed October 24, 2026.
[14] src/slic3r/GUI/Tab.cpp at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/src/slic3r/GUI/Tab.cpp. Accessed October 24, 2026.
[15] src/slic3r/GUI/GCodeRenderer/BaseRenderer.cpp at v02.08.02.61, Bambu Studio on GitHub. https://github.com/bambulab/BambuStudio/blob/v02.08.02.61/src/slic3r/GUI/GCodeRenderer/BaseRenderer.cpp. Accessed October 24, 2026.
[16] Cooling Settings, Bambu Lab Wiki. https://wiki.bambulab.com/en/software/bambu-studio/auto-cooling. Accessed October 24, 2026.
[17] OrcaFilamentLibrary base fdm_filament_common.json at v2.4.2, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.4.2/resources/profiles/OrcaFilamentLibrary/filament/base/fdm_filament_common.json. Accessed October 24, 2026.
[18] OrcaFilamentLibrary base fdm_filament_pla.json at v2.4.2, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.4.2/resources/profiles/OrcaFilamentLibrary/filament/base/fdm_filament_pla.json. Accessed October 24, 2026.
[19] OrcaFilamentLibrary Generic PLA @System.json at v2.4.2, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.4.2/resources/profiles/OrcaFilamentLibrary/filament/Generic%20PLA%20%40System.json. Accessed October 24, 2026.
[20] src/libslic3r/PrintConfig.cpp at v2.4.2, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.4.2/src/libslic3r/PrintConfig.cpp. Accessed October 24, 2026.
[21] src/slic3r/GUI/Tab.cpp at v2.4.2, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.4.2/src/slic3r/GUI/Tab.cpp. Accessed October 24, 2026.
[22] src/slic3r/GUI/GCodeViewer.cpp at v2.4.2, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.4.2/src/slic3r/GUI/GCodeViewer.cpp. Accessed October 24, 2026.
[23] resources/definitions/fdmprinter.def.json at 5.13.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.13.0/resources/definitions/fdmprinter.def.json. Accessed October 24, 2026.
[24] resources/definitions/creality_base.def.json at 5.13.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.13.0/resources/definitions/creality_base.def.json. Accessed October 24, 2026.
[25] resources/setting_visibility/advanced.cfg at 5.13.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.13.0/resources/setting_visibility/advanced.cfg. Accessed October 24, 2026.
[26] UltiMaker Cura: Cooling settings, UltiMaker Support. https://support.ultimaker.com/articles/Knowledge/1667417947135. Accessed October 24, 2026.
[27] Overheating, Simplify3D Print Quality Troubleshooting Guide. https://www.simplify3d.com/resources/print-quality-troubleshooting/overheating/. Accessed October 24, 2026.
[28] Curling or Rough Corners, Simplify3D Print Quality Troubleshooting Guide. https://www.simplify3d.com/resources/print-quality-troubleshooting/curling-or-rough-corners/. Accessed October 24, 2026.
[29] M106: Set Fan Speed, Marlin Firmware documentation. https://marlinfw.org/docs/gcode/M106.html. Accessed October 24, 2026.
[30] G-Codes, Klipper documentation. https://www.klipper3d.org/G-Codes.html. Accessed October 24, 2026.

**Cite as:** Asked 3D. "Why are my small prints melting and blobby?." October 24, 2026. https://asked3d.com/answers/small-prints-melting-blobby. Last verified October 24, 2026.