What is Z hop in 3D printing?

Asked 3D Content Team
Published October 30, 2026 · Last verified October 30, 2026

Abstract. Z hop lifts the nozzle after a retraction so travel moves clear the print. Bambu Studio ships it on at 0.4 mm; UltiMaker Cura ships it off, set to 1 mm.

Keywords: retraction, slicer-settings

1 What is Z hop in 3D printing?

Z hop is a slicer setting that lifts the nozzle a set height right after a retraction, travels at that height, then lowers it before printing resumes. That keeps the nozzle from dragging across or knocking into the print. Bambu Studio ships it on at 0.4 mm; UltiMaker Cura ships it off, with the height at 1 mm.

1.1 The number

Table 1. The number

Slicer	Setting name and where it is	As shipped
Bambu Studio 2.x	“Z hop when retract” and “Z Hop Type”: Printer settings, Extruder, Retraction	On at 0.4 mm, type Auto, in the Bambu Lab X1 Carbon 0.4 mm nozzle profile. Accepts 0 to 5 mm
OrcaSlicer 2.x	“Z-hop height” and “Z-hop type”: Printer settings, Extruder, Z-Hop	Field default 0.4 mm, type Slope; printer profiles can override both
PrusaSlicer 2.9	“Lift height”: Printers, Extruder 1, Travel lift	Field default 0 (off). Prusa’s profiles turn it on: 0.4 mm on the MK3S/MK3S+, 0.2 mm on the MINI/MINI+. The MK4S and CORE One use “Use ramping lift” instead, a 1° ramp up to a “Maximum ramping lift” of 1.5 mm, which Prusa’s PLA profiles lower to 0.6 mm
UltiMaker Cura 5.x	“Z Hop When Retracted” (a checkbox) and “Z Hop Height”: Travel	Off, height 1 mm. The UltiMaker S5 profile turns it on only when two extruders are enabled, at 2 mm

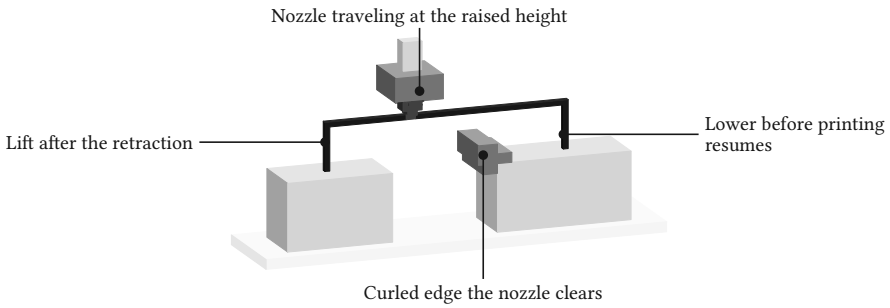


Figure 1. With Z hop on, the nozzle rises after the retraction, crosses at the raised height and drops back down, so it passes over a curled edge instead of striking it.

The Bambu row comes from Bambu Studio v02.08.02.61′s source and X1 Carbon profile, OrcaSlicer's from v2.4.2′s source, PrusaSlicer's from version_2.9.6′s source and Prusa profiles, and Cura's from the 5.13.0 printer definitions. What height to choose is its own question: Z hop height.

1.2 Why does the nozzle need to lift?

Because during a travel the nozzle tip passes just above the layer it finished, and anything standing proud of that layer is in its way. Prusa's crash detection article says layer shifts usually happen when the extruder moves quickly between objects and hits a curled-up piece of print, and that overhangs can warp upward into the nozzle. Cura's own description says the hop keeps the nozzle from hitting the print and reduces the chance of knocking it off the build plate.

1.3 What is Z hop when retracted?

It is Cura's full name for the setting, and it describes how all four slicers behave: the lift rides on a retraction.

- **Bambu Studio and OrcaSlicer** grey out the Z hop field when retraction length is 0.
- **Cura** hides "Z Hop When Retracted" while "Enable Retraction" is off.
- **PrusaSlicer** travels flat whenever a move needs no retraction.

So a travel too short to trigger a retraction (under 1 mm on the X1 Carbon profile) also skips the hop.

1.4 What is the difference between Z hop and retraction?

They are separate moves. Retraction pulls filament back into the hotend to cut the pressure that causes oozing and strings (what retraction does), while Z hop raises the nozzle to clear the part. Bambu Lab's wiki puts them in order: the nozzle lifts once the retraction is complete. Retraction works without Z hop, but Z hop needs a retraction to fire.

2 Should you use Z hop?

Use it when the nozzle hits your prints: scraping curled or warped overhangs, shifting layers or knocking tall thin parts loose. Bambu Studio and Prusa's printer profiles ship it on, Bambu at 0.4 mm. If nothing collides, leave it off, as Cura does: each hop adds a Z move up and down and can worsen stringing.

2.1 When do slicer makers say to turn it on?

- **Curling and warped overhangs.** Prusa suggests raising the Z-hop distance in PrusaSlicer, or the print fan speed, when overhangs warp up into the nozzle. Bambu Studio's Auto type switches to a spiral lift over overhang areas, which Bambu Lab says may warp and get scraped.
- **Tall, thin parts and several objects on one plate.** OrcaSlicer's wiki names both as cases where Z hop is particularly useful.
- **Two extruders.** Cura's "Z Hop After Extruder Switch" lifts after a tool change so the nozzle does not leave oozed material on the outside of the print. The UltiMaker S5 profile turns Z hop on only for two-extruder prints.

2.2 What are the downsides of Z hop?

- **Stringing.** Prusa's stringing article says a lower lift value reduces stringing, but warns that turning lift off can let the nozzle hit the part. The causes are in does Z hop cause stringing.
- **Time.** Cura moves Z at "Z Hop Speed": 10 mm/s by default, 5 mm/s in Creality profiles. Cura sets it below print speed because the bed or gantry is harder to move, and each hop makes two of those moves.
- **More retractions in some Cura profiles.** Creality's Cura profiles switch Combing Mode off whenever Z hop is on, so travels that combing would have kept inside the part now retract and hop.

2.3 Can you use Z hop only where it is needed?

Yes. Each slicer has a way to narrow where it fires:

- **Cura:** "Z Hop Only Over Printed Parts" hops only when the travel cannot route around printed parts.
- **OrcaSlicer:** "On surfaces" limits it to top surfaces, bottom surfaces or both.
- **Bambu Studio, OrcaSlicer and PrusaSlicer:** lower and upper Z limits restrict it to a height band.

The lift can also ramp instead of going straight up: Slope and Spiral in Bambu Studio and OrcaSlicer, ramping lift in PrusaSlicer (which Z hop type).

2.4 How to check it on your own print

1. **Find the setting** in the place the table lists for your slicer. Also check the filament profile, which can set its own Z hop: Setting Overrides in Bambu Studio and OrcaSlicer, Filament Overrides in PrusaSlicer.
2. **Confirm retraction is on.** Z hop does nothing while retraction length is 0 or Cura's "Enable Retraction" is unticked.
3. **Slice and look at the travel moves** in the preview: with Z hop on, travels rise off the layer and drop back at the next start point.
4. **Print the part that failed and watch the travels** over curled edges and between parts. If you see scuffs or hear a knock with Z hop off, enable Z hop.
5. **If strings appear with it on,** limit where it fires or switch to a sloped, spiral or ramping lift before turning Z hop off.
6. **If the nozzle still hits with it on,** fix the curling too: Prusa pairs a higher Z hop with more print fan speed. Then raise the height.

References

[1] 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 30, 2026.

[2] 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 30, 2026.
[3] resources/profiles/BBL/machine/Bambu Lab X1 Carbon 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/machine/Bambu%20Lab%20X1%20Carbon%200.4%20nozzle.json. Accessed October 30, 2026.
[4] Retraction (Bambu Studio parameters), Bambu Lab Wiki. https://wiki.bambulab.com/en/software/bambu-studio/parameter/retraction. Accessed October 30, 2026.
[5] 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 30, 2026.
[6] 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 30, 2026.
[7] Z-Hop (printer_extruder_z_hop), OrcaSlicer wiki on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/wiki/printer_extruder_z_hop. Accessed October 30, 2026.
[8] 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 30, 2026.
[9] 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 30, 2026.
[10] src/libslic3r/GCode.cpp at version_2.9.6, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/src/libslic3r/GCode.cpp. Accessed October 30, 2026.
[11] src/libslic3r/GCode/Travels.cpp at version_2.9.6, PrusaSlicer on GitHub. https://github.com/prusa3d/PrusaSlicer/blob/version_2.9.6/src/libslic3r/GCode/Travels.cpp. Accessed October 30, 2026.
[12] 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 30, 2026.
[13] Stringing and oozing, Prusa Knowledge Base. https://help.prusa3d.com/article/stringing-and-oozing_1805. Accessed October 30, 2026.
[14] Crash detection, Prusa Knowledge Base. https://help.prusa3d.com/article/crash-detection_2100. Accessed October 30, 2026.
[15] 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 30, 2026.
[16] resources/definitions/ultimaker_s5.def.json at 5.13.0, UltiMaker Cura on GitHub. https://github.com/Ultimaker/Cura/blob/5.13.0/resources/definitions/ultimaker_s5.def.json. Accessed October 30, 2026.
[17] 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 30, 2026.

Cite as: Asked 3D. “What is Z hop in 3D printing?.” October 30, 2026. https://asked3d.com/answers/what-is-z-hop. Last verified October 30, 2026.