How do you know if pressure advance is too high?

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

Abstract. Rounded corners, lines that thin into a corner and gaps between perimeters mean pressure advance is too high: lower it. Over 0.200, Klipper's extruder may skip.

Keywords: pressure-advance, firmware, slicer-settings

1 How do you know if pressure advance is too high?

Pressure advance is too high when corners come out rounded, the line thins or breaks as the nozzle slows into a corner, and gaps open between perimeters at the corner. Lower the value. Bulging corners mean it is too low. Klipper's docs warn that above 0.200 the extruder may skip steps under high acceleration.

1.1 The number

Table 1. The number

What	Value	Applies to
Typical pressure advance	0.050 to 1.000, the high end usually only on Bowden extruders	Klipper
Extruder may skip at normal acceleration	Over 0.200	Klipper
No real improvement by this value: turn pressure advance off	1.000	Klipper
Typical K, Linear Advance 1.5	About 0.1 to 2.0, higher on Bowden	Marlin
Old Linear Advance 1.0 K for PLA (not valid on 1.5)	30 to 130	Marlin
K compiled in when `LIN_ADVANCE` is enabled (`ADVANCE_K`)	0.22	Marlin 2.1.2.9
Prusa's tested K, 0.4 mm nozzle	PLA 0.05, PETG 0.08, ASA/ABS 0.04	Prusa MK3 family
Best value for one filament, OrcaSlicer's worked example	0.036 at 50 mm/s and 1000 mm/s², 0.024 at 200 mm/s and 4000 mm/s²	OrcaSlicer pattern test

Values come from Klipper's pressure advance guide, Marlin's Linear Advance page and its 2.1.2.9 `Configuration_adv.h`, Prusa's Linear Advance article and the worked example in OrcaSlicer's adaptive pressure advance guide.

1.2 Should pressure advance be higher or lower?

Look at the tip of a sharp corner: if it is rounded or gapped, go lower; if it bulges, go higher. When two values look equally good, Klipper's guide says take the lower one. A slight, smooth rounding is normal even when tuned: OrcaSlicer's guide puts it down to square corner velocity.

Table 2. Should pressure advance be higher or lower?

What you see	Pressure advance is	What to do
Rounded corner tip, line thin leading into it, gaps between perimeters beside it	Too high	Lower it
Gaps at corners and between infill and walls on fast features only	Too high for those speeds	Lower it, or use adaptive pressure advance
Thin line just after a bridge	Too high for bridges	Set OrcaSlicer's bridge value to about half your usual one
Extruder clicks or skips on hard acceleration	Too high for that acceleration	Lower acceleration or pressure advance
Marlin test line thick where the fast section starts, thin where it ends	Too high	Lower K
Blob or bulge at the corner tip, line thin as it leaves	Too low	Raise it (why corners bulge)
Marlin test line thin where the fast section starts, thick where it ends	Too low	Raise K

1.3 Why does too much pressure advance leave gaps?

Because a high value pulls back more filament than the nozzle pressure needs as the head slows for a corner, so the nozzle runs short where the line turns. Pressure advance pushes extra filament in as the toolhead accelerates and, per Klipper's configuration reference, retracts an equal amount as it decelerates (what pressure advance is). Klipper calls the result "rounded corners and poor extrusion leading up to the corner."

No Klipper setting limits that extra extruder movement. At high acceleration a very high value asks for more push than the extruder motor has torque for, and it skips.

1.4 What causes gaps after sharp corners?

Usually pressure advance, and the corner tip shows which way. A rounded tip with gaps beside it means too high. A

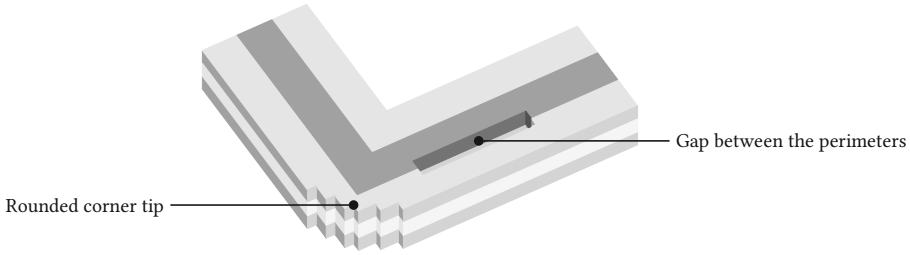


Figure 1. With pressure advance set too high, the corner tip prints rounded and a gap opens between the perimeters next to it.

bulging tip with a thin line leaving it means too low: Marlin says a low K starves the line as it accelerates. Two other cases leave the same gaps:

- **A value tuned on slow walls.** OrcaSlicer's guide says the best value drops as speed, layer height and acceleration rise, so a value that gives sharp outer-wall corners leaves "slight corner-perimeter separation" on faster features.
- **Acceleration above the input shaper's limit.** Past the maximum acceleration Klipper's input shaper calibration recommends, OrcaSlicer warns that shaper smoothing causes separations "that appear like PA issues". Lower the acceleration, not pressure advance.

1.5 Klipper pressure advance too high: what do you change?

Lower `pressure_advance` under `[extruder]` in printer.cfg and send `RESTART`, or try a value live with `SET_PRESSURE_ADVANCE ADVANCE=<value>`. Check the sliced G-code too: with pressure advance enabled in the filament profile, OrcaSlicer 2.4.2 writes its own `SET_PRESSURE_ADVANCE ADVANCE=` line, which replaces the printer.cfg value for that print. If the extruder skips at your normal acceleration, Klipper's fix is a lower acceleration or no pressure advance.

1.6 Linear advance too high: what does it look like in Marlin?

The same rounded, gapped corners, plus a signature on Marlin's K-factor pattern. Each test line prints 20 mm slow, 40 mm fast and 20 mm slow, and Marlin says a too-high K overshoots as the fast section starts and starves as it ends. Prusa's rule: "If you see missing filament, decrease the K value."

Linear Advance 1.5 is built not to skip steps: when a K needs more extruder speed than the extruder jerk limit allows, Marlin lowers that move's acceleration instead. Check where your K came from:

- **A Linear Advance 1.0 value.** Version 1.5 changed the unit, and Marlin says to recalibrate. Prusa firmware 3.9.0 and later converts 1.0 values in G-code automatically.
- **The compiled-in default.** The 0.22 applies until something sets another K, such as an `M900` in the G-code.

`M900` alone reports the K in use, `M900 K<value>` sets it and `M900 K0` disables Linear Advance.

1.7 How to check it on your own print

1. **Find the value in use.** Klipper: `pressure_advance` in `[extruder]`, then search the sliced G-code for `SET_PRESSURE_ADVANCE`. Marlin: send `M900` and search the G-code for `M900`. PrusaSlicer keeps the MK3 K in the filament profile's custom G-code. On a Bambu Lab printer the value is the Flow Dynamics K: where to find and change it.
2. **Compare a sharp outside corner with the table above,** but skip the corner where the layer changes (the seam): Klipper's guide says it often prints differently.
3. **Rerun the calibration from zero.** In Klipper, send `SET_VELOCITY_LIMIT SQUARE_CORNER_VELOCITY=1 ACCEL=500`, then `TUNING_TOWER COMMAND=SET_PRESSURE_ADVANCE PARAMETER=ADVANCE START=0 FACTOR=.005` (direct drive) or `FACTOR=.020` (long Bowden). Print the hollow square tower at about 100 mm/s with a layer height near 75% of the nozzle diameter. In OrcaSlicer, use the Calibration menu. To pick the new value off the print, see how to read a pressure advance test.
4. **If only fast features gap,** repeat the test at your infill speed and acceleration, or set up adaptive pressure advance in OrcaSlicer's filament settings.
5. **If the extruder clicks,** reprint with pressure advance off (`SET_PRESSURE_ADVANCE ADVANCE=0` or `M900 K0`). If the clicking stops, lower pressure advance or acceleration. If not, the cause lies elsewhere: why an extruder clicks.

References

[1] Pressure advance, Klipper documentation (same text as the v0.13.0 release). https://www.klipper3d.org/Pressure_Advance.html. Accessed October 24, 2026.
[2] Configuration reference, Klipper documentation. https://www.klipper3d.org/Config_Reference.html. Accessed October 24, 2026.
[3] G-Codes, Klipper documentation. https://www.klipper3d.org/G-Codes.html. Accessed October 24, 2026.
[4] Linear Advance, Marlin Firmware documentation. https://marlinfw.org/docs/features/lin_advance.html. Accessed October 24, 2026.

[5] M900: Linear Advance Factor, Marlin Firmware documentation. https://marlinfw.org/docs/gcode/M900.html. Accessed October 24, 2026.
[6] Marlin/Configuration_adv.h at 2.1.2.9, Marlin on GitHub. https://github.com/MarlinFirmware/Marlin/blob/2.1.2.9/Marlin/Configuration_adv.h. Accessed October 24, 2026.
[7] Linear Advance, Prusa Knowledge Base. https://help.prusa3d.com/article/linear-advance_2252. Accessed October 24, 2026.
[8] Pressure Advance calibration, OrcaSlicer Wiki. https://www.orcaslicer.com/wiki/calibration/pressure_advance_calib. Accessed October 24, 2026.
[9] Adaptive Pressure Advance calibration, OrcaSlicer Wiki. https://www.orcaslicer.com/wiki/calibration/adaptive_pressure_advance_calib. Accessed October 24, 2026.
[10] src/libslic3r/GCodeWriter.cpp at v2.4.2, OrcaSlicer on GitHub. https://github.com/OrcaSlicer/OrcaSlicer/blob/v2.4.2/src/libslic3r/GCodeWriter.cpp. Accessed October 24, 2026.

Cite as: Asked 3D. "How do you know if pressure advance is too high?" October 24, 2026. https://asked3d.com/answers/pressure-advance-too-high. Last verified October 24, 2026.