

FIRST PRINT: WHAT ACTUALLY MATTERS

Slicers usually come with decent presets out of the box, but knowing what the main settings do gives you much better control over your prints and helps avoid most common mistakes.

The 3 things that will make or break your first print

Bed leveling

If the nozzle is too far from the bed, nothing sticks. Too close, it clogs. Modern printers auto-level and compensate for slight misalignment in the first layers. Check that this feature is enabled and has run before starting.

Material temps

PLA, PETG, ABS all print at different temps. Wrong temp = stringing, warping, or no adhesion.

Layer height

Drives print time and surface quality. Start at 0.2mm, the universal beginner setting.

Core slicer settings and what they do

Layer height

Thickness of each printed layer. Quality vs. speed tradeoff

0.2mm ← start here

Wall count (perimeters)

Shell thickness. More walls = stronger, heavier part, but slower print

3 walls ← start here

Retraction

Pulls filament back to prevent stringing between moves

1-6mm (varies)

Print speed

How fast the nozzle moves. Slower = better quality. Faster=vibration

40-60 mm/s ← start here

Top/bottom layers

Solid layers to close the shell. Too few = pinholes

4-5 layers* ← start here

Infill density

How solid the inside is. Most prints don't need 100%

15-20% ← default

Supports

Temporary scaffolding for overhangs > 45°. Turn on if needed

Off by default

*Top/bottom layer count — do the math

Aim for 1-1.5mm of solid shell regardless of layer height. A fixed layer count doesn't scale, thinner layers need more of them.

Rule of thumb: always set thickness in mm, then convert

0.3mm layers
 $1\text{mm} \div 0.3 \approx 4 \text{ layers}$

0.2mm layers
 $1\text{mm} \div 0.2 = 5 \text{ layers}$

0.1mm layers
 $1\text{mm} \div 0.1 = 10 \text{ layers}$

Material temperature cheat sheet

Beginner

PLA

Nozzle: 190-220°C

Bed: 25-60°C

Cooling fan: on

Easiest to print, great for prototypes. Outdoor use not advised. No enclosure needed.

Intermediate

PETG

Nozzle: 230-250°C

Bed: 70-85°C

Cooling fan: partial

Stronger than PLA, better heat resistance, slightly more fussy.

Advanced

ABS / ASA

Nozzle: 230-250°C

Bed: 100-110°C

Cooling fan: off

Needs enclosure. Prone to warping. Can be used outside

Common first-print problems and fixes

Print not sticking

Re-level the bed. Increase bed temp by 5°C. Clean bed with IPA. Enable brim in slicer, apply adhesive.

Layer separation

Increase bed temp. Use a brim or raft. Ensure no drafts hit the print. ABS needs enclosure. Use adhesive.

Stringing between parts

Increase retraction distance/speed. Lower nozzle temp by 5°C. Enable "wipe on retract".

Layer separation

Increase nozzle temp. Slow down print speed. Check filament diameter matches slicer setting.