Slicer Settings Cheat Sheet

Quick Reference Guide for Cura, PrusaSlicer & Bambu Studio

3D Print Calculator | www.3dprintcalculator.co.uk

PLA Settings

Cura Settings for PLA

Setting	Value	Notes
Nozzle Temperature	200-210°C	Start at 205°C, adjust ±5°C
Bed Temperature	50-60°C	60°C for better adhesion
Print Speed	50-60 mm/s	Walls: 30-40 mm/s
Layer Height	0.2mm	0.1mm for detail, 0.3mm for speed
First Layer Height	0.3mm	Better bed adhesion
Line Width	0.4mm	Match nozzle diameter
Retraction Distance	5-6mm	Direct drive: 1-3mm
Retraction Speed	45-50 mm/s	Faster = less stringing
Cooling Fan	100%	Essential for PLA
Infill	15-25%	Grid or cubic pattern
Wall Thickness	0.8-1.2mm	2-3 walls typically
Top/Bottom Layers	4-6 layers	Ensure solid top/bottom

PrusaSlicer Settings for PLA

Setting	Value	Notes
Nozzle Temperature	200-210°C	First layer: 215°C
Bed Temperature	60°C	First layer: 60°C
Perimeters	2-3	Wall count
Top/Bottom Layers	4-5	Solid layers
Infill	15-25%	Rectilinear or grid
Layer Height	0.2mm	Quality preset
Retraction Length	2mm	Direct drive default
Cooling	100%	After first layer

Bambu Studio Settings for PLA

Setting	Value	Notes
Nozzle Temperature	220°C	Bambu PLA preset
Bed Temperature	55°C	Standard bed temp
Print Speed	200-300 mm/s	High-speed capable
Layer Height	0.2mm	Standard quality
Infill	15-25%	Grid or gyroid

Cooling	100%	Auto-cooling enabled

PETG Settings

Cura Settings for PETG

Setting	Value	Notes
Nozzle Temperature	230-250°C	Start at 240°C
Bed Temperature	70-80°C	80°C for better adhesion
Print Speed	40-50 mm/s	Slower than PLA
Layer Height	0.2-0.3mm	0.2mm recommended
Retraction Distance	6-7mm	Reduce stringing
Retraction Speed	40-45 mm/s	Slower than PLA
Cooling Fan	30-50%	Less cooling needed
Infill	20-30%	Higher for strength

> 💡 **PETG Tip:** PETG can be sticky. Use a release agent (hairspray, glue stick) on the bed to prevent over-adhesion.

ABS Settings

Cura Settings for ABS

Setting	Value	Notes
Nozzle Temperature	240-260°C	Start at 250°C
Bed Temperature	90-100°C	Essential for adhesion
Print Speed	40-60 mm/s	Moderate speed
Layer Height	0.2-0.3mm	0.2mm for quality
Retraction Distance	5-6mm	Standard retraction
Retraction Speed	40-50 mm/s	Moderate speed
Cooling Fan	0-20%	Minimal cooling
Infill	20-30%	Standard infill

⚠️ **ABS Warning:** ABS requires an enclosure to prevent warping. Ensure good ventilation as ABS emits fumes during printing.

TPU Settings

Cura Settings for TPU

Setting	Value	Notes
Nozzle Temperature	220-240°C	Start at 230°C
Bed Temperature	50-60°C	Standard bed temp
Print Speed	20-30 mm/s	Very slow - essential
Layer Height	0.2-0.3mm	0.2mm recommended
Retraction Distance	1-2mm	Minimal retraction
Retraction Speed	20-30 mm/s	Very slow
Cooling Fan	0-30%	Minimal cooling
Infill	20-30%	Standard infill

⚠️ **TPU Warning:** TPU requires a direct drive extruder. Bowden setups will struggle with flexible filament. Print very slowly!

Common Settings Explained

Layer Height

- **0.1mm:** High detail, slow printing (4x slower than 0.2mm)
- **0.2mm:** Standard quality, balanced speed/quality
- **0.3mm:** Fast printing, lower detail

Infill Patterns

- **Grid:** Strong, fast printing
- **Cubic:** Strongest, good for functional parts
- **Gyroid:** Strong, isotropic, no top layer issues
- **Triangles:** Strong, decorative
- **Lines:** Fast, weak

Infill Percentage Guide

- **10-15%:** Decorative items, prototypes
- **20-25%:** Standard functional parts
- **30-50%:** Strong parts, mechanical components
- **50-100%:** Maximum strength, heavy-duty parts

Retraction Settings

- **Direct Drive:** 1-3mm retraction distance
- **Bowden:** 5-7mm retraction distance
- **Speed:** 40-60 mm/s typically
- **Purpose:** Prevents stringing and oozing

Troubleshooting Common Issues

Stringing / Oozing

- Increase retraction distance
- Increase retraction speed
- Lower nozzle temperature by 5-10°C
- Enable "Coasting" in Cura
- Increase travel speed

Layer Adhesion Issues

- Increase nozzle temperature by 5-10°C
- Increase bed temperature
- Reduce layer height
- Slow down print speed
- Check for drafts (use enclosure)

Warping

- Increase bed temperature
- Use brim or raft
- Use enclosure (especially for ABS)
- Ensure bed is level
- Clean bed surface

Poor First Layer

- Level bed properly
- Increase first layer height to 0.3mm
- Slow down first layer speed to 20 mm/s
- Increase first layer temperature by 5-10°C
- Use proper bed adhesion (hairspray, glue stick)

Overhangs Failing

- Increase cooling fan speed
- Reduce layer height

- Slow down print speed
- Add supports for angles >45°
- Reduce nozzle temperature

Quick Reference Table

Material	Nozzle	Bed	Speed	Fan	Retraction
PLA	200-210°C	50-60°C	50-60 mm/s	100%	5-6mm
PETG	230-250°C	70-80°C	40-50 mm/s	30-50%	6-7mm
ABS	240-260°C	90-100°C	40-60 mm/s	0-20%	5-6mm
TPU	220-240°C	50-60°C	20-30 mm/s	0-30%	1-2mm

Temperature Tuning Tips

Start with manufacturer's recommended temperature

Print a temperature tower to find optimal temp

Adjust in 5°C increments

Too hot = stringing, oozing, poor overhangs

Too cold = poor layer adhesion, under-extrusion

Speed Tuning Tips

Start slow (30-40 mm/s) and increase gradually

Walls should be slower than infill

First layer should be slowest (20 mm/s)

Outer walls: 30-40 mm/s for quality

Infill: Can be faster (50-80 mm/s)

3D Print Calculator - Slicer Settings Cheat Sheet

These settings are starting points. Always test and adjust for your specific printer and filament brand.