Bambu Studio 1.9.1 settings

By Gambody v 1.0

These basic 3D printing settings recommendations for beginners were tested in Bambu Studio **1.9.1**. Test models were printed on **Bambu lab A1, Bambu lab A1 mini, Creality Ender 3 s1, Anycubic Kobra 2, Anycubic Vyper** with PLA and PETG filaments.

Note:

- You can upscale your 3D printing models. Downscaling is not recommended - it can make smaller parts of the model unprintable, distort the model's level of detail and result in assembly issues.
- All connectors should be printed at 80-90% Infill.
- When printing Lock connectors, we recommend setting the "Brim width" parameter to 0 in the "Skirt and Brim" print settings. In that way, the Locks will be 3D printed with a Skirt only.

Bambu Lab

printers series settings

In our opinion, the Bambulab company offers optimal settings for its printers that require minimal changes. For better printing, change only those parameters indicated by "★"

In order to see all the parameters listed below, you need to enable the Advance mode in the Bambu Studio settings.

Quality Tab

Layer height:

Layer Height: 0.1-0.2 mm

First layer height: 0.24-0.28 mm(120-150% of your Layer Height)

Line width:

Default: 0.42 mm

Initial Layer: 0.5-0.6 mm

Outer wall: 0.4 mm
Inner wall: 0.45 mm
Top surface: 0.45 mm
Sparse infill: 0.45 mm
Internal solid infill: 0.45 mm
Support: 0.42 mm

Seam:

Aligned (But you can paint the seam  if you want.)

Precision:

Slice gap closing radius: 0.049 mm
Resolution: 0.012 mm
Arc fitting: **off**
X-Y hole compensation: 0.01-0.05 (you have to calibrate this parameter)
X-Y contour compensation: 0.01-0.05 (you have to calibrate this parameter)
Elephant foot compensation: 0.1-0.2 (you have to calibrate this parameter)

Ironing:

Ironing No ironing

Wall generator

Wall generator: classic
Wall transitioning threshold angle: 10°
Wall transitioning filter margin: 25%
Wall transition length: 100%
Wall distribution count: 1
Minimum wall width: 85%
Minimum feature size: 25%

Advanced

Order of walls: inner/outer
Print infill first: **off**
Bridge flow: 0.85
Thick bridges: **off**
Only one wall on top surfaces: Top surfaces
Only one wall on first layer: **off**
Avoid crossing wall: **off**

Strength Tab

Walls

Wall loops: 2-3
Detect thin wall: off (disabled for vehicles and enabled for characters)
Top/bottom shells:

Top/bottom shells

Top surface pattern: Monotonic line
★Top shell layers: 6 (for 0,2 Layer Height)

★Top shell thickness: 1.2
Bottom surface pattern Monotonic: Monotonic line
★Bottom shell layers: 6 (for 0,2 Layer Height)
★Bottom shell thickness: 1.2
Internal solid infill pattern: Rectilinear

Sparse infill

★Sparse infill density: 6%
★Sparse infill pattern: Triangle
Length of sparse infill anchor: 400% mm or %
Maximum length of sparse infill anchor: 20 mm or %

Advanced

Infill/Wall overlap: 15-25%
Infill direction: 45°
Bridge direction: 0°
Minimum sparse infill threshold: 10 mm2
Infill combination: **off**
Detect narrow internal solid infill: **on**
Ensure vertical shell thickness: **on**
Internal bridge support thickness: 0.8 mm

Speed Tab

Initial layer speed

The parameters in this tab vary greatly, it all depends on the quality of your printer. For example, if you have a classic Ender3, stick to the minimum parameters, but if you have a newer printer, for example Anycubic cobra 2, you can select the maximum recommended values

Initial layer: 15-45 mm/s (check your build plate adhesion)
Initial layer infill: 35-50 mm/s

Other layers speed

Outer wall: 30-150 mm/s
Inner wall: 30-150 mm/s
Small perimeters: 50% mm/s or %
Small perimeter: threshold 0 mm
Sparse infill: 50-70 mm/s
Internal solid infill: 50-150 mm/s
Top surface: 25-120 mm/s
Slow down for overhangs
Overhang speed: 0 mm/s (10%, 25%)
20 mm/s [25%, 50%)
15 mm/s [50%, 75%)
10 mm/s 175%, 100%)
Bridge: 20-40 mm/s
Gap infill: 30-100 mm/s

Travel speed

Travel: 80-150 mm/s

Acceleration

(Settings for advanced users, change these parameters only if you have sufficient 3D printing expertise)

Normal printing: 2500 mm/s
Initial layer: 2000 mm/s
Outer wall: 700 mm/s
Inner wall: 0 mm/s
Top surface: 0 mm/s
Sparse infill:

Jerk(XY)

Default: 0 mm/s
Outer wall: 9 mm/s
Inner wall: 9 mm/s
Infill: 9 mm/s
Top surface: 9 mm/s
First layer: 9 mm/s
Travel: 9 mm/s

Support Tab

Support

Enable support: **on** (enable this parameter if your model requires supports)
Type: normal/tree
Style: default
★Threshold angle: 10-60° (We also recommend placing and removing supports manually in some places using button)
On build plate only: off
Remove small overhangs: **on**

Raft

Raft layers: 0 layers

Filament for Supports

Support/raft base: default
Support/raft interface: default

Advanced

Support wall loops: 0
★Top Z distance: 0.2-0.25 mm
★Bottom Z distance: 0.2-0.25 mm
Base pattern: Default
Base pattern spacing: 2.5 mm
Pattern angle: 0°
Top interface layers: 3

Bottom interface layers: same as top layers
★Interface pattern: concentric
★Top interface spacing: 0 mm
Normal Support expansion: 0 mm
★Support/object xy distance: 1 mm
★Don't support bridges: **on**
Independent support layer height: **on**

Others Tab

Bed adhesion

Skirt loops: 0
Skirt height: 1 layer (for PLA and PETG)
★Brim type: outer and inner brim
★Brim width: 5 mm (5-8 mm is optional for small prints that have bad adhesion to the build plate)

★Brim-object gap: 0-0.12 mm

Prime tower

Enable: **off**

Flush options

Flush into objects infill: off
Flush into objects support: off

Special mode

Slicing Mode: Regular
Print sequence: By layer
Spiral vase: off
Timelapse: Traditional
Fuzzy Skin: None

Advanced

Interlocking depth of a segmented region: 0 mm

G-code output

Reduce infill retraction: **on**

Post-processing scripts

Filament Tab

Filament

Type: PLA/PetG
Vendor: Generic
Default color: -//-
Diameter: 1.75 mm
Flow ratio: 0.9-1.1 (you have to calibrate this parameter)
Enable pressure advance

Density -//-

Price -//-

Softening temperature 45°-80° (read the description on your filament roll)

Recommended nozzle temperature 190°-270° (read the description on your filament roll and increase this parameter for fast printers)

Nozzle: 190°-270° (read the description on your filament roll and increase this parameter for fast printers)

Plate: 0°-80° (read the description on your filament roll)

Cooling

No cooling for the first: 1-3 layers

Min fan speed threshold: Fan speed 60%/ Layer time 80s for PLA

Min fan speed threshold: Fan speed 40%/ Layer time 30s for PetG

Max fan speed threshold: Fan speed 80%/ Layer time 8s for PLA

Max fan speed threshold: Fan speed 90%/ Layer time 12s for PetG

Keep fan always on: Check

Slow printing down for better layer cooling Check

Min print speed: 15 mm/s

Force cooling for overhangs and bridges: Check

Cooling overhang threshold: 50% for PLA

Cooling overhang threshold: 10% for PetG

Fan speed for overhangs: 100% for PLA

Fan speed for overhangs: 90% for PetG

Auxiliary part cooling fan

Fan speed: 70% for PLA

Fan speed: 0% for PetG