Bambu Studio 1.9.5 settings

By Gambody v 1.0

These basic 3D printing settings recommendations for beginners were tested in Bambu Studio **1.9.5**. Test models were printed on **Bambu lab A1, Bambu lab A1 mini, Creality Ender 3 s1, Anycubic Kobra 2** 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.

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: **On**
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: 1
Thick bridges: **off**
Only one wall on top surfaces: Top surfaces
Only one wall on first layer: **off**
Smooth speed discontinuity: **On**
Smooth coefficient: 80
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: 5%
★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: 15 mm2
Infill combination: **off**
Detect narrow internal solid infill: **on**
Ensure vertical shell thickness: **on**

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%)
Over 100% wall (not bridge): 24 mm/s

Bridge: 20-40 mm/s
Gap infill: 30-100 mm/s
Support: 150 mm/s
Support interface: 80 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

Initial layer expansion: 2 mm
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: 2
Bottom interface layers: 2
★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