Bambu Studio 2.0+

Gambody printing recommendations for:

Imladris 3D Printer Files | Assembly

Below you'll find detailed slicing settings for Bambu Studio 2.0+ to help you get the best results when printing this model.

These settings are optimized specifically for this 3D model and should work well in most cases. But they're not set in stone - depending on your printer, material, or even the specific part you're working with, feel free to tweak things.

Every 3D printing setup is different, so feel free to make the changes that work best for your machine. When in doubt, check your printer's manual - or reach out to our Support Team at support@gambody.com

We'll be happy to help with any questions, suggestions, or issues you may have regarding the recommended printing settings!

Quality Tab

Layer height	
Layer Height	**0.12 - 0.20** mm
For better quality use 0.12 mm layer height, for fast printing use 0.2 mm layer height. For pins and the Ge connectors, use 0.2 layer height.	
First layer height	**0.20 - 0.28** mm

120-150% of your Layer Height???????	
Line width	
Default:	**0.42** mm
Initial Layer	**0.50 - 0.60** mm
Outer wall	**0.40** mm
Inner wall	**0.45** mm
Top surface	**0.45** mm
Sparse infill	**0.45** mm
Internal solid infill	**0.42** mm
Support	**0.42** mm
Seam	
Seam position	**Aligned**
But you can paint the seam if you want.	
Smart scarf seam application	✔
Scarf application angle threshold	**155.0** °
Scarf steps	**10.0**
Scarf joint for inner walls	✔
Role-based wipe speed	✔

Precision	
Slice gap closing radius	**0.0490** mm
Resolution	**0.0120** mm
Arc fitting	✔
X-Y hole compensation	**0.010 - 0.050** mm
You have to calibrate this parameter	
X-Y contour compensation	**0.010 - 0.050** mm
You have to calibrate this parameter	
Elephant foot compensation	**0.10 - 0.20** mm
You have to calibrate this parameter	
Wall generator	
Wall generator	**Classic**
Wall transitioning threshold angle	**10.0** °
Wall transitioning filter margin	**25.0** %
Wall transition length	**100.0** %
Minimum wall width	**85.0** %
Minimum feature size	**25.0** %
Advanced	

Order of walls	inner/outer
Bridge flow	0.85
Only one wall on top surfaces	Top surfaces
Smooth speed discontinuity area	✔
Smooth coefficient	80.0

Strength

Walls	
Wall loops	2 - 3
For pins and power elements of the structure, such as the vehicle frame, use 3 loop	
Detect thin wall	
Disabled for vehicles and enabled for characters	
Top/bottom shells	
Top surface pattern	Monotonic
Top shell layers	5
For 0,2 Layer Height	
Top shell thickness	1.00 mm
Top paint penetration layers	5

Bottom surface pattern	Monotonic
Bottom shell layers	5
Bottom shell thickness	1.00 mm
Bottom paint penetration layers	5
Internal solid infill pattern	Rectilinear
Sparse infill	
Sparse infill density	6.0 %
Sparse infill pattern	Triangles
Length of sparse infill anchor	400.0 %
Maximum length of sparse infill anchor	20.0 mm
Advanced	
Infill/Wall overlap	15.0 - 25.0 %
Infill direction	45.0 °
Bridge direction	0.0 °
Minimum sparse infill threshold	10.0 mm^2
Detect narrow internal solid infill	✔
Ensure vertical shell thickness	Enabled
Detect floating vertical shells	✔

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 3 v2, you can select the maximum recommended values	
Initial layer speed	
Initial layer	**15.0 - 45.0** mm/sec
Initial layer infill	**35.0 - 75.0** mm/sec
Other layers speed	
Outer wall	**30.0 - 150.0** mm/sec
Inner wall	**30.0 - 250.0** mm/sec
Small perimeters	**50.0** %
Small perimeter threshold	**0.1** mm
Sparse infill	**50.0 - 250.0** mm/sec
Internal solid infill	**50.0 - 200.0** mm/sec
Vertical shell speed	**80.0** %

Top surface	25.0 - 150.0 mm/sec
Slow down for overhangs	✔
Overhang speed	
Overhang speed 10%	0.0 mm/sec
Overhang speed 25%	40.0 mm/sec
Overhang speed 50%	20.0 mm/sec
Overhang speed 75%	10.0 mm/sec
Overhang speed 100%	10.0 mm/sec
Bridge	20.0 - 40.0 mm/sec
Gap infill	30.0 - 150.0 mm/sec
Support	30.0 - 100.0 mm/sec
Support interface	30.0 - 60.0 mm/sec
Travel speed	
Travel	80.0 - 350.0 mm/sec
Acceleration	

Settings for advanced users, change these parameters only if you have sufficient 3D printing expertise	
Normal printing	2500.0 - 4000.0 mm/sec²
Travel	2000.0 - 7000.0 mm/sec²
Initial layer travel	700.0 - 5000.0 mm/sec²
Initial layer	300.0 - 500.0 mm/sec²
Outer wall	500.0 - 3000.0 mm/sec²
Inner wall	500.0 - 4000.0 mm/sec²
Top surface	500.0 - 2500.0 mm/sec²
Sparse infill	100.0 - 100.0 %

Support

Support	
Enable support	✔

Enable this parameter if your model requires supports	
Type	Tree (auto)
Style	Default
Threshold angle	10.0 - 60.0 °
We also recommend placing and removing supports manually in some places using special button	
Remove small overhangs	✔
Raft	
Raft layers	0 layers
Advanced	
Initial layer density	90.0 %
Initial layer expansion	-1.0 mm
Support wall loops	-1 - 2
1-2 loops for more thick support	
Top Z distance	0.20 - 0.25 mm
Top Z distance = 1-1.3 layer Height. If the supports are hard to remove, try increasing this setting by 0.1-0,4 mm	
Bottom Z distance	
Bottom Z distance = 1-1.3 layer Height. If the supports are hard to remove, try increasing this setting by 0.1-0,4 mm	

Base pattern	Rectilinear
Base pattern spacing	2.50 mm
Pattern angle	0.0 °
Top interface layers	2 - 3 layers
Interface pattern	Concentric
Top interface spacing	0.00 - 0.50 mm
You have to calibrate this parameter which one is better for your filament	
Normal Support expansion	0.00 mm
Support/object xy distance	0.35 - 0.80 mm
Increase this parameter if the supports are hard to remove from walls	
Support/object first layer gap	0.35 mm
Max bridge length (only for tree supports)	1.0 mm
Independent support layer height	✔
Tree Support (only for tree supports)	
Branch distance	5.0 mm
Branch diameter	2.0 mm
Branch angle	45.0 °
Branch diameter angle	5.0 °

Others

Bed adhesion	
Skirt loops	**0**
Skirt height	**1** layers
For PLA and PETG filament types	
Brim type	**Outer and inner brim**
Brim width	**5.00** mm
5-8 mm is optional for small prints that have bad adhesion to the build plate	
Brim-object gap	**0.01 - 0.12** mm
Prime tower	
Enable	✔
Skip points	✔
Width	**35.0** mm
Max speed	**90.0** mm/sec
Brim width	**3.0** mm
Infill gap	**150.0** %

Rib wall	✔
Rib width	8.0 mm
Fillet wall	✔
Flush options	
Flush into objects support	✔
Special mode	
Slicing Mode	Regular
Print sequence	By layer
Timelapse	Traditional
G-code output	
Reduce infill retraction	✔

Filament settings

Filament	
Type	PLA
Filament ramming length	10.0 mm
Filament prime volume	45.0 mm^3

Diameter	1.75 mm
Flow ratio	0.90 - 1.10
You have to calibrate this parameter	
Shrinkage	100.0 %
Velocity Adaptation Factor	1.0
Softening temperature	45.0 - 80.0 °C
Read the description on your filament roll	
Travel time after ramming	0.0 sec
Precooling target temperature	0.0 °C
Recommended nozzle temperature	190.0 - 270.0 °C
Read the description on your filament roll and increase this parameter for fast printers	
Print temperature	
Cool Plate SuperTack	45.0 - 45.0 °C
Cool Plate	35.0 - 35.0 °C
Engineering Plate	0.0 - 0.0 °C
Smooth PEI Plate / High	65.0 - 65.0 °C
Temp Plate	65.0 - 65.0 °C
Textured PEI Plate	65.0 - 65.0 °C

Nozzle	220.0 - 270200.0 °C
Read the description on your filament roll and increase this parameter for fast printers	
Volumetric speed limitation	
Max volumetric speed	12.0 mm/sec
Ramming volumetric speed	-1.0 mm/sec
Filament scarf seam settings	
Scarf start height	10.0 %
Scarf length	10.0 mm
Cooling	
Cooling for specific layer	1 layers
No cooling for the first	1 layers
Part cooling fan	
Min fan speed threshold	
Fan speed	60.0 %
Layer time	90.0 sec
Max fan speed threshold	
Fan speed	80.0 %

Layer time	8.0 sec
Keep fan always on	✔
Slow printing down for better layer cooling	✔
Min print speed	10.0 mm/sec
Force cooling for overhangs and bridges	✔
Cooling overhang threshold	50.0 %
Overhang threshold for participating cooling	100.0 %
Fan speed for overhangs	100.0 %
Pre start fan time	0.0 sec
Auxiliary part cooling fan	
Fan speed	70.0 %

Printer Settings Tab

Motion ability	
Jerk limitation	
Maximum jerk X	7.0 mm/sec
Maximum jerk Y	7.0 mm/sec
Maximum jerk Z	0.4 mm/sec

Maximum jerk E	5.0 mm/sec
Extruder	
Basic information	
Type	Direct drive extruder
Nozzle diameter	0.40 mm
Nozzle volume	0.0 mm^3
Layer height limits	
Min	0.08 mm
Max	0.35 mm
Retraction	
Length	0.70 mm
Z hop when retract	0.3 mm
Z hop lower boundary	0.0 mm
Z Hop Type	Normal
Retraction Speed	30.0 mm/sec
Deretraction Speed	30.0 mm/sec
Travel distance threshold	1.0 mm
Retract when change layer	✔

Wipe while retracting	✔
Wipe Distance	2.0 mm
Retract amount before wipe	70.0 %

Best regards,
your Ge team