

# Bambu Studio 2.0+

Gambody printing recommendations for:  
Death Darksiders 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](mailto: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
<i>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.</i>	
First layer height	0.20 - 0.28 mm

*120-150% of your Layer Height????????*

<b>Line width</b>	
<b>Default:</b>	0.42 mm
<b>Initial Layer</b>	0.50 - 0.60 mm
<b>Outer wall</b>	0.40 mm
<b>Inner wall</b>	0.45 mm
<b>Top surface</b>	0.45 mm
<b>Sparse infill</b>	0.45 mm
<b>Internal solid infill</b>	0.42 mm
<b>Support</b>	0.42 mm
<b>Seam</b>	
<b>Seam position</b>	Aligned
<i>But you can paint the seam if you want.</i>	
<b>Smart scarf seam application</b>	✓
<b>Scarf application angle threshold</b>	155.0 °
<b>Scarf steps</b>	10.0
<b>Scarf joint for inner walls</b>	✓
<b>Role-based wipe speed</b>	✓

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

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

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

## 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*


<b>Initial layer speed</b>	
<b>Initial layer</b>	15.0 - 45.0 mm/sec
<b>Initial layer infill</b>	35.0 - 75.0 mm/sec
<b>Other layers speed</b>	
<b>Outer wall</b>	30.0 - 150.0 mm/sec
<b>Inner wall</b>	30.0 - 250.0 mm/sec
<b>Small perimeters</b>	50.0 %
<b>Small perimeter threshold</b>	0.1 mm
<b>Sparse infill</b>	50.0 - 250.0 mm/sec
<b>Internal solid infill</b>	50.0 - 200.0 mm/sec
<b>Vertical shell speed</b>	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*

<b>Normal printing</b>	<b>2500.0 - 4000.0</b> mm/sec <sup>2</sup>
<b>Travel</b>	<b>2000.0 - 7000.0</b> mm/sec <sup>2</sup>
<b>Initial layer travel</b>	<b>700.0 - 5000.0</b> mm/sec <sup>2</sup>
<b>Initial layer</b>	<b>300.0 - 500.0</b> mm/sec <sup>2</sup>
<b>Outer wall</b>	<b>500.0 - 3000.0</b> mm/sec <sup>2</sup>
<b>Inner wall</b>	<b>500.0 - 4000.0</b> mm/sec <sup>2</sup>
<b>Top surface</b>	<b>500.0 - 2500.0</b> mm/sec <sup>2</sup>
<b>Sparse infill</b>	<b>100.0 - 100.0</b> %

## Support

<b>Support</b>	
<b>Enable support</b>	

<i>Enable this parameter if your model requires supports</i>	
Type	Tree (auto)
Style	Default
Threshold angle	10.0 - 60.0 °
<i>We also recommend placing and removing supports manually in some places using special button</i>	
Remove small overhangs	<input checked="" type="checkbox"/>
Raft	
Raft layers	0 layers
Advanced	
Initial layer density	90.0 %
Initial layer expansion	-1.0 mm
Support wall loops	-1 - 2
<i>1-2 loops for more thick support</i>	
Top Z distance	0.20 - 0.25 mm
<i>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</i>	
Bottom Z distance	
<i>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</i>	

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
<i>You have to calibrate this parameter which one is better for your filament</i>	
Normal Support expansion	0.00 mm
Support/object xy distance	0.35 - 0.80 mm
<i>Increase this parameter if the supports are hard to remove from walls</i>	
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
<i>For PLA and PETG filament types</i>	
Brim type	Outer and inner brim
Brim width	5.00 mm
<i>5-8 mm is optional for small prints that have bad adhesion to the build plate</i>	
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 <sup>3</sup>

Diameter	1.75 mm
Flow ratio	0.90 - 1.10
<i>You have to calibrate this parameter</i>	
Shrinkage	100.0 %
Velocity Adaptation Factor	1.0
Softening temperature	45.0 - 80.0 °C
<i>Read the description on your filament roll</i>	
Travel time after ramming	0.0 sec
Precooling target temperature	0.0 °C
Recommended nozzle temperature	190.0 - 270.0 °C
<i>Read the description on your filament roll and increase this parameter for fast printers</i>	
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

<b>Nozzle</b>	220.0 - 270200.0 °C
<i>Read the description on your filament roll and increase this parameter for fast printers</i>	
<b>Volumetric speed limitation</b>	
<b>Max volumetric speed</b>	12.0 mm/sec
<b>Ramming volumetric speed</b>	-1.0 mm/sec
<b>Filament scarf seam settings</b>	
<b>Scarf start height</b>	10.0 %
<b>Scarf length</b>	10.0 mm
<b>Cooling</b>	
<b>Cooling for specific layer</b>	1 layers
<b>No cooling for the first</b>	1 layers
<b>Part cooling fan</b>	
<b>Min fan speed threshold</b>	
<b>Fan speed</b>	60.0 %
<b>Layer time</b>	90.0 sec
<b>Max fan speed threshold</b>	
<b>Fan speed</b>	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 <sup>3</sup>
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*