# Bambu Studio 2.0+

Gambody printing recommendations for:

U-Wing Starfighter 3D Printer Files | Assembly + Action

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