Slic3r 1.3+

Gambody printing recommendations for:

Misty Pokemon 3D Printer Files | Assembly

Below you'll find detailed slicing settings for Slic3r 1.3+ 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!

Print Settings Tab

Layers and perimeters	
Layer height	
Layer height	**0.12** 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	0.18 mm
120-150% of your Layer Height	
Vertical shells	
Perimeters	2
Horizontal shells	
Solid layers top	8 layers
For 0.12 Layer Height	
Solid layers bottom	8 layers
For 0.12 Layer Height	
Quality (slower slicing)	
Extra perimeters if needed	✔
Avoid crossing perimeters	✔
Detect thin walls	✘
Detect bridging perimeters	✘
Advanced	
Seam position	Aligned
External perimeters first	✘

Infill	
Infill	
Fill density	**5.0** %
For pins and connectors use 50% Infill	
Fill pattern top	**Rectilinear**
Fill pattern bottom	**Rectilinear**
Reducing printing time	
Combine infill every	**2** layers
Advanced	
Fill gaps	✔
Solid infill every	**0** layers
Fill angle:	**45.0** °
Solid infill threshold area	**70** mm^2
Only retract when crossing perimeters	✔
Infill before perimeters	✘
Skirt and brim	
Skirt	
Use skirt for outdated 3d printers	

Loops (minimum)	3
Distance from object	5.00 mm²
Skirt height	1 layers
Minimum extrusion length	50.0 mm
Brim	
(5-8 mm is optional for small prints that have bad adhesion to the build plate)	
Exterior brim width	8.00 mm
Interior brim width	3.00 mm
Support material	
Support material	
Generate support material	✔
Enable this parameter if your model requires supports	
Overhang threshold	60.0 %
(45-60 degree)You have to calibrate this parameter according to the capabilities of your printer and your filament, using a Gambody test models	
Options for support material and raft	
Contact Z distance:	0.15 mm

Contact Z distance = 1-1.3 layer Height. If the supports are hard to remove, try increasing this setting by 0.1-0,4 mm	
Pattern	rectilinear
Pattern spacing	2.50 mm
Pattern angle	0.0 °
Interface layers	3 layers
Interface pattern spacing	0.00 mm
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 v3, you can select the maximum recommended values	
Speed for print moves	
Perimeters	25.0 - 200.0 mm/sec
small	52.0 - 90.0 mm/sec
external	25.0 - 90.0 mm/sec
Infill	50.0 - 160.0 mm/sec
solid	25.0 - 160.0 mm/sec

top solid	25.0 - 100.0 mm/sec
gaps	25.0 - 160.0 mm/sec
Bridges	25.0 - 50.0 mm/sec
Support material	50.0 - 200.0 mm/sec
interface	100.0 %
Speed for non-print moves	
Travel:	100 - 250 mm/sec
Modifiers	
First layer speed	15.00 - 30.00 mm/sec
Acceleration control (advanced)	
Settings for advanced users, change these parameters only if you have sufficient 3D printing expertise. Use the minimum value for outdated printers without acceleration calibration, and the maximum value for modern printers if you need it.	
Perimeters:	500 - 3000 mm/sec²
Infill:	500 - 3000 mm/sec²

Bridge:	500 - 3000 mm/sec²
First layer:	500 - 1000 mm/sec²
Default:	500 - 4000 mm/sec²
Advanced	
Extrusion width	
Default extrusion width: auto	✔

Filament Settings Tab

Filament	
Filament	
Diameter	0.40 mm
Extrusion multiplier	1.00
You have to calibrate this parameter from 0.9 to 1.1 according to the capabilities of your printer and your filament, using a Gambody test models.	
Temperature	
Check your filament manufacturer's temperature recommendations on the spool.	
Extruder first layer	220.0 °C

Extruder other layers	220.0 °C
Bed first layer	60.0 °C
Bed other layers	60.0 °C
Cooling	
Cooling parameters depends on the material you use for printing.	
Enable	
Keep fan always on	✔
Fan settings	
Fan speed	80.0 - 100.0 %
Bridges fan speed	100.0 %
Disable fan for the first	1 layers
Cooling thresholds	
Enable fan if layer print time is below	60.0 sec
Slow down if layer print time is below	5.0 sec
Min print speed	10.0 mm/sec

Printer Settings Tab

General	
Size and coordinates	

Z offset	**0.00** mm
Calibrate this value if you need to reduce or improve the adhesion between the plastic and the heat bed	
Capabilities	
Extruders	**1**
Has heated bed	✔
Extruder	
Size	
Nozzle diameter:	**0.40** mm
Your current nozzle diameter	
Retraction	
You need to calibrate this parameter using Gambody test models. These values are average values for a Direct Drive extruder; for a Bowden extruder, the values should be increased.	
Length	**1.00** mm
Lift Z	**0.30** mm
Speed	**45.00** mm/sec
Extra length on restart	**0.00** mm
Minimum travel after retraction	**2.00** mm

Best regards,
your Ge team