PrusaSlicer 2.0+

Gambody printing recommendations for:

Mustrum Ridcully 3D Printer Files | Assembly

Below you'll find detailed slicing settings for PrusaSlicer 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!

Print Settings Tab

Layers and perimeters	
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	0.18 - 0.28 mm
120-150% of your Layer Height	
Vertical shells	
Perimeters	2 - 3
Horizontal shells	
Solid layers top	4
for 0.2 Layer Height	
Solid layers bottom	4
Minimum shell thickness top	0.80 mm
Minimum shell thickness bottom	0.80 mm
Quality (slower slicing)	
Extra perimeters if needed	✔
Ensure vertical shell thickness	Enabled
Advanced	
Seam position	Aligned
But you can paint the seam if you want.	
Seam gap distance	15.0 %
Scarf joint placement	Nowhere

External perimeters first	✔
Fill gaps	✔
Perimeter generator	Arachne
Fuzzy skin (experimental)	
Fuzzy Skin	None
Fuzzy skin thickness	0.30 mm
Fuzzy skin point distance	0.80 mm
Only one perimeter	
Single perimeter on top surfaces	Disabled
Infill	
Infill	
Fill Density	6.0 %
Fill Pattern	Triangles
Length of the infill anchor	10.00 mm
Maximum length of the infill anchor	10.00 mm
Top Fill Pattern	Monotonic Lines
Bottom fill pattern	Monotonic Lines
Ironing	

Enable ironing	✘
Skirt and brim	
Skirt	
Loops (minimum)	0
Distance from brim/object	6.00 mm
Skirt height	1 layers
(for PLA and PETG)	
Minimal filament extrusion length	50.00 mm
Brim	
Brim type	Outer brim only
Brim width	5.00 - 8.00 mm
(5-8 mm is optional for small prints that have bad adhesion to the build plate)	
Brim separation gap	0.12 mm
Support material	
Support material	
Generate support material	✔
Enable this parameter if your model requires supports	

Auto generated supports	✔
Overhang threshold	60.0 °
(45-50 degree)You have to calibrate this parameter according to the capabilities of your printer and your filament, using a Gambody test models.	
Enforce support for the first	2 layers
First layer density	90.0 %
First layer expansion	3.0 mm
Raft	
Raft layers	0 layers
Options for support material and raft	
Style	Grid
Top contact Z distance	0.24 mm
Top 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	
Bottom contact Z distance	0.24 mm
Top 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	2.5 mm
Pattern angle	0.0 °
Top interface layers	2 layers
Bottom interface layers	2 layers
Interface pattern	Concentric
Intertace pattern spacing	0.00 mm
XY separation between an object and its support	1.00 mm
Increase this parameter if the supports are hard to remove from walls	
Don't support bridges	✔
Organic supports	
Maximum Branch Angle	45.0 °
Preferred Branch Angle	30.00 °
Branch Diameter	2.00 mm
Branch Diameter Angle	5.00 °
Branch Diameter with double walls	3.00 mm
Tip Diameter	0.80 mm
Branch Distance	1.00 mm
Branch Density	30.0 %

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
Small perimeters	**25.0 - 150.0** mm
External perimeters	**25.0 - 200.0** mm/sec
Infill	**50.0 - 250.0** mm/sec
Solid infill	**40.0 - 200.0** mm/sec
Top solid infill	**25.0 - 200.0** mm/sec
Support material	**50.0 - 300.0** mm/sec
Support material interface	**100.0 - 100.0** %
Bridges	**25.0 - 100.0** mm/sec

Over bridges	25.0 - 100.0 mm/sec
Gap fill	25.0 - 100.0 mm/sec
Ironing	15.0 - 80.0 mm/sec
Dynamic overhang speed	
speed for 0% overlap (bridge)	0.0 mm/sec
speed for 25% overlap	0.1 mm/sec
speed for 50% overlap	0.0 mm/sec
Speed for non-print moves	
Travel	100.0 - 150.0 mm/sec
Modifiers	
First layer speed	15.0 - 30.0 mm/sec
Speed of object first layer over raft interface	15.0 - 30.0 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.	

External perimeters	500.00 - 3000.00 mm/sec²
Perimeters	500.00 - 4000.00 mm/sec²
Top solid infill	500.00 - 1500.00 mm/sec²
Solid infill	500.00 - 4000.00 mm/sec²
Infill	500.00 - 3000.00 mm/sec²
Bridge	500.00 - 4000.00 mm/sec²
First layer	500.00 - 1000.00 mm/sec²
First object layer over raft interface	500.00 - 3000.00 mm/sec²
Wipe tower	500.00 - 4000.00 mm/sec²
Travel	500.00 - 7000.00 mm/sec²
Travel short distance acceleration	500.00 - 6000.00 mm/sec²

Default	500.00 - 4000.00 mm/sec²
Multiple Extruders	
These settings only work for 3D printers with multiple extruders	
Extruders	
Perimeter extruder:	1
Infill extruder:	1
Solid infill extruder	1
Advanced	
Extrusion width	
You can try setting all parameters in this section, except the First layer, to values between 0.75% of your nozzle diameter and 1.25% of your nozzle diameter. Adjusting them will help you work out the optimal parameters for the best quality for your print. As for the First layer, you can set it to 150% of the diameter of your nozzle for better adhesion to the build plate (for a nozzle with a diameter of 0.4 mm, the First layer extrusion width can be from 0.3 mm to 0.5 mm)	
Default extrusion width	0.44 mm
First layer	0.4 %
Perimeters	0.44 mm
External perimeters	0.42 mm

Infill	0.44 mm
Solid infill	0.44 mm
Top solid infill	0.40 mm
Support material	0.36 mm
Overlap	
Infill/perimeters overlap:	25.0 %
Flow	
Bridge flow ratio:	0.95
Arachne perimeter generator	
Perimeter transitioning threshold angle	10.0 °
Perimeter transitioning filter margin	25.0 %
Perimeter transition length	100.0 %
Perimeter distribution count	1
Minimum perimeter width	85.0 %
Minimum feature size	24.9 %

Filament Settings Tab

Filament	
Filament	PLA

Diameter	0.40 mm
Extrusion multiplier	1.0
For better printing quality you have to calibrate this parameter using Gambody test model.	
Temperature	
Check your filament manufacturer's temperature recommendations on the spool.	
Cooling	
Cooling parameters depends on the material you use for printing.	
Enable auto cooling	✔
Cooling slowdown logic	Uniform cooling
Perimeter transition distance	0.00 mm
Fan settings	
Fan speed	
*for PLA	
Min	50.0 %
Max	100.0 %
Bridges fan speed	100.0 %
Disable fan for the first	2.0 layers

Full fan speed at layer	4.0 layers
Cooling thresholds	
Enable fan if layer print time is below	600.0 sec
Slow down if layer print time is below	9.0 sec
Min print speed	20.0 mm/sec

Best regards,
your Ge team