Simplify3D 5.0+

Gambody printing recommendations for:

Facehugger 3D Printer Files in Diorama | Assembly

Below you'll find detailed slicing settings for Simplify3D 5.0+ 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!

Extruder

General	
Nozzle diameter	**0.4** mm
Your current nozzle diameter	
Extrusion Multiplier	**0.98**
You have to calibrate this parameter using Gambody test models.	

Extrusion Width	
Automatic	✔
Ooze Control	
Use 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.	
Retract Distance	0.80 mm
Retract Vertical Lift	1.00 mm
Retract Speed	40.00 mm/sec

Layer

General	
Primary Extruder	Primary Extruder
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.	
Top Solid Layers	5
For 0,2 Layer Height	
Bottom Solid Layers	5

For 0,2 Layer Height	
Outline Perimeters	**2**
To increase the strength of the print parts, use Outline Perimeters: 3	
Adaptive Layer Height	
Enable Adaptive Layer Heights	✔
You can enable this parameter to print rounded or spherical models, as well as character models.	
Minimum Adaptive Layer Height	**0.08** mm
Maximum Adaptive Layer Height	**0.24** mm
Adaptive Smoothing Level	**5.0**
Dimensional Adiustments	
Horizontal Outer Size Compensation	**0.00** mm
Use this option only if your parts are too tight. but better calibrate your printer extrusion	
Horizontal Inner Size Compensation	**0.00** mm
Use this option only if your parts are too tight. but better calibrate your printer extrusion	

First Layer Settings	
First Layer Height	100.0 %
First Layer Width	150.0 %
First Layer Speed	30.0 %
Start Point Selection	
Optimize start points for fastest printing	✔
Restrict start points to preferred regions	✔
Printing Order	
Outline Printing Order	Inside-Out
Island Printing Order	Minimize Print Time

Additions

Use Skirt/Brim	
Skirt Extruder	Primary extruder
Skirt Layers	1 layers
Skirt Offset	0.00 mm
Use 2 and more if you want to create skirt instead brim	
Skirt Outlines	

1-2 for skirt and 10-20 for brim	
Use Raft	✘
Use Prime Pillar	✘
Use for wipe nozzle if you need	
Use Ooze Shield	✘
Use For ABS filament	

Infill

Sparse Internal Infill	
Internal Infill Pattern	Gyroid
Internal Pattern Rotation	0.0 °
Infill Percentage	7.0 %
For pins and connectors use 50% Infill	
Infill Extrusion Width	100.0 %
Combined Infill Layers	2 layers
Outline Overlap	15.0 %
Minimum Infill Length	5.00 mm
Dense Internal Infill	

Dense Infill Layers	0 layers
Dense Infill Percentage	50.0 %
Solid Layers	
External Infill Pattern	Rectilinear
External Pattern Rotation	0 °
Solid Infill Threshold Area	25.00 mm²
Solid Infill Extra Expansion	0.00 mm
Add Solid Diaphragms	✘

Support

General	
Support Extruder	All Extruders
Support Infill Pattern	Aligned
Support Pattern Rotation	0.0 °
Support Infill Percentage	30.0 %
Support Outlines	0.00
Base Support Layers	0 layers
Support inflation Distance	-0.01 mm

Combined Support Layers	1 layers
Dense Supports	
Upper Dense Support Layers	0 layers
Lower Dense Support Layers	0 layers
Dense Support Infill Percentage	70.0 %
Dense Support Extra Expansion	0.00 mm
Part Separation	
Support Horizontal Offset from Part	0.24 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	

Temperature

Extruder	
General	
Enable temperature controller	✔
Temperature Number	T0
Temperature Type	Extruder
Stabilize temperature controller at beginning of print	✔

Per-Layer Setpoints	
Calibrate your filament and detect optimal temperature for it	
Temperature	230.0 °C
Idle Cooldown	
Cooldown Extruder While Ide	✘
Heated Bed	
General	
Enable temperature controller	✔
Stabillze temperature controller at beginning of print	✔
Per-Layer Setpoints	
Average temperature for PLA filament	
Temperature	60.0 °C
Idle Cooldown	
Cooldown Extruder While Ide	✘

Cooling

Per-Layer Setpoints	
Fan Speed Setpoints	

Setpoint 1	
Fan Speed Percentage	0.0 %
At layer	1 layers
Setpoint 2	
Fan Speed Percentage	100.0 %
At layer	2 layers

Speeds

General	
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	
Default Printing Speed	40 - 160 mm/sec
Outer Perimeter Speed	50.0 %
Inner Perimeter Speed	80.0 %
Solid Infill Speed	80.0 %
Sparse Support Speed	80.0 %
Dense Support Speed	70.0 %
XY Travel Speed	200.0 mm/sec

Z Travel Speed	15.0 mm/sec
Time Estimation	
Settings for advanced users, change these parameters only if you have sufficient 3D printing expertise.	
XY Acceleration	500.0 - 3000.0 mm/sec²
Z Acceleration	150.0 - 600.0 mm/sec²
Extruder Acceleration	1000.0 - 4000.0 mm/sec²
XY Jerk	10.00 - 25.00 mm/sec
Z Jerk	0.30 - 0.50 mm/sec
Extruder Jerk	5.00 - 5.00 mm/sec
Speed Overrides	
Reduce print speed for excessively quick layers	✔
Begin reducing speed for layers below	15.0 sec
Minimum quick layer speed percentage	20.0 %
Reduce print speed for short perimeters	
Begin reducing speed for perimeters below	80.00 mm

Minimum short perimeter speed percentage	50.0 %

Other

Bridging	
Unsupported area threshold	50.00 mm²
Extra inflation distance	1.00 mm
Bridging extrusion multiplier	95.0 %
Bridging speed multiplier	30.0 %
Filament Properties	
Filament diameter	1.75 mm

Advanced

Thin Wall Behavior	
External Thin Wall Type	Perimeters only
Internal Thin Wall Type	Allow gap fill
Allowed Perimeter Overlap	10.0 %
Single Extrusions	
Minimum Single Extrusion Length	1.00 mm
Minimum Single Extrusion Width	50.0 %

Maximum Single Extrusion Width	200.0 %
Single Extrusion Endpoint Extension	0.20 mm
Ooze Control Behavior	
Only retract when crossing open spaces	✔
Force retraction between layers	✔

Best regards,
your Ge team