

Simplify3D v5.0.0

Settings by Gambody v 1.0



These are averaged settings which were tested in the Simplify3D v5.0.0 slicer. Test models were printed on Anycubic Vyper, FLSUN v400, Ender3 S1 with PLA filament.

Disclaimer: The following printing settings are a recommendation, not an obligation. The parameters can vary depending on the peculiarities of your 3D printer, the material you use, and especially the particular assembly part you are working with. Each part that any model comprises often needs preliminary review, and you are free to tweak the settings the way you find suitable.

Note:

- You can scale up the model (downscaling is not recommended!);
- All connectors should be printed at 100% Infill;

To avoid printing problems, we recommend the following settings:

Extruder

Nozzle Diameter*	0.4 mm
Extrusion Multiplier**	0.95
Extrusion Width:	Auto
Retraction Distance	5.00 mm
Extra retract Distance	0.00 mm
Retraction Vertical Lift	0.00 mm

Retraction Speed	35 mm/s
Coasting Distance	0.4 mm
Wipe Distance	off

** Your current nozzle diameter*

*** Calibrate your flow (from 90-100)*

Layer

Primary Layer Height*	0.16 mm
Top Solid Layers	8
Bottom Solid Layers	8
Outline/Perimeter Shells	4
Enable Adaptive Layer Heights	✓
Minimum Adaptive Layer Height **	0.12mm
Maximum Adaptive Layer Height	0.2 mm
Adaptive Smoothing Level	5
Horizontal Outer Size Compensation ***	0.00 mm
Horizontal Inner Size Compensation ****	0.00 mm
First Layer Height	150%
First Layer Width	100%
First Layer Speed	20%
Align start position at a specific XY location	X:0 Y:300
Outline Printing Order	Inside-Out
Island Printing Order	Minimize Print Time

**From 0.08-0.2 mm*

*** Experimental parameter*

**** Use this option only if your parts are too tight. but better calibrate your printer extrusion*

***** Use this option only if your parts are too tight. but better calibrate your printer extrusion*

Additions

Use Skirt/Brim	✓
Skirt Layers	1
Skirt Offset from Part*	10.00 mm
Skirt Outlines**	2
Use Prime Pillar***	
Use Ooze Shield****	

**Use 0 if you want to create brim instead skirt*

*** 1-2 for skirt and 10-20 for brim*

****Use for wipe nozzle if you need*

*****Use For ABS filament*

Infill

Internal Fill Pattern	Cyroid
Internal PatternRotation	0°
Infill Percentage	10%
Infill Extrusion Width	100%
Combined Infill Layers	1 layers
Outline Overlap	15%
Minimum Infill Length	5.00 mm
Dense infill Layer	0

Dense infill Percentage	50%
External Infill Pattern	Rectilinear
External Pattern Rotation	0°
Solid Infill Threshold Area	25 mm ²

** For small parts and all parts with connectors use 100% Infill.*

Support

Generate Support Material	✓
Support Infill Percentage:	30%
Extra Inflation Distance:	1.00 mm
Support Base Layers	2
Combine Support Every	1 layers
Dense Support Layers	3
Dense Infill Percentage	70%
Support Type	Normal
Support Pillar Resolution	5.00 mm
Max Overhang Angle*	60 °
Horizontal Offset From Part:	0.50 mm
Upper Vertical Separation Layers**	1
Lower Vertical Separation Layers:	1

**Calibrate your filament and detect optimal angle (40°-70°)*

*** Increase to 2 if supports do not separate well*

Temperature

Extruder 1 Temperature:*	190 - 210 °C**
Heated Bed:	60 °C
Wait for temperature controller to stabilize	✓

**Calibrate your filament and detect optimal temperature for it*

***Average temperature for PLA filament*

Cooling

Pre-layer Fan Controls	1 layer - 0 2 layer - 100
Bridging fan speed override:	100%

Speeds

Default Printing Speed:	60 mm/sec
Outer Perimeter Speed:	50%
Outer Perimeter Speed:	80%
Solid Infill Speed:	80%
Sparse Support Speed	80%
Dense Support Speed	70%
X/Y Axis Movement Speed:	180 mm/sec
Z Axis Movement Speed:	15 mm/sec
<i>Time estimation</i>	
XY Acceleration	8000 mm/sec ²

Z Acceleration	150 mm/sec ²
Extruder Acceleration	10000 mm/sec ²
XY Jerk	5 mm/sec
Z Jerk	0.4 mm/sec
Extruder Jerk	15 mm/sec
Reduce print speed for short perimeters	✓
Begin reducing speed for layers Below	15 sec
Minimum quick layer speed percentage	20%
Begin reducing speed for perimeters below	80 mm
Minimum short perimeter speed percentage	50 %

Other

Unsupported area threshold:	10.0 sq mm
Extra inflation distance	1.00 mm
Bridging extrusion multiplier:	95%
Bridging speed multiplier:	30%
Apply bridging settings to perimeters	✓

Advanced

External Thin Wall Type:	Perimeters Only
Internal Thin Wall Type:	Allow single extrusion fill
Allowed perimeter overlap:	15%

Minimum Extrusion Length:	1.00 mm
Minimum Printing Width:	50%
Maximum Printing Width:	200%
Endpoint Extension Distance:	0.2 mm
Force retraction between layers	✓
Require Minimum travel for retraction:	2.00 mm
Perform retraction during wipe movement:	✓
Only wipe extruder for outer-most perimeters:	✓
Non-manifold segments:	Heal

*Dear customer, if you have any questions or suggestions for the printing settings for the CURA slicer v 5.2 , you can email our **Support Team** - support@gambody.com. We will be happy to assist you*

*Best regards,
your Ge team*