# Simplify3D 5.0+

Gambody printing recommendations for:

Spider-Punk Comics 3D Printer Files | 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