# PrusaSlicer 2.0+

Gambody printing recommendations for:

Strife Darksiders 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