# Slic3r 1.3+

Gambody printing recommendations for:

Dwarf Cleric Warrior 3D Printer Files | Static

Below you'll find detailed slicing settings for Slic3r 1.3+ 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** 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 mm |
| 120-150% of your Layer Height | |
| Vertical shells | |
| Perimeters | 2 |
| Horizontal shells | |
| Solid layers top | 8 layers |
| *For 0.12 Layer Height* | |
| Solid layers bottom | 8 layers |
| *For 0.12 Layer Height* | |
| Quality (slower slicing) | |
| Extra perimeters if needed | ✔ |
| Avoid crossing perimeters | ✔ |
| Detect thin walls | ✘ |
| Detect bridging perimeters | ✘ |
| Advanced | |
| Seam position | Aligned |
| External perimeters first | ✘ |

| | |
|---|---|
| **Infill** | |
| **Infill** | |
| **Fill density** | **5.0** % |
| *For pins and connectors use 50% Infill* | |
| **Fill pattern top** | **Rectilinear** |
| **Fill pattern bottom** | **Rectilinear** |
| **Reducing printing time** | |
| **Combine infill every** | **2** layers |
| **Advanced** | |
| **Fill gaps** | ✔ |
| **Solid infill every** | **0** layers |
| **Fill angle:** | **45.0** ° |
| **Solid infill threshold area** | **70** $mm^2$ |
| **Only retract when crossing perimeters** | ✔ |
| **Infill before perimeters** | ✘ |
| **Skirt and brim** | |
| **Skirt** | |
| *Use skirt for outdated 3d printers* | |

| | |
|---|---|
| Loops (minimum) | 3 |
| Distance from object | 5.00 mm² |
| Skirt height | 1 layers |
| Minimum extrusion length | 50.0 mm |
| Brim | |
| (5-8 mm is optional for small prints that have bad adhesion to the build plate) | |
| Exterior brim width | 8.00 mm |
| Interior brim width | 3.00 mm |
| Support material | |
| Support material | |
| Generate support material | ✔ |
| *Enable this parameter if your model requires supports* | |
| Overhang threshold | 60.0 % |
| *(45-60 degree)You have to calibrate this parameter according to the capabilities of your printer and your filament, using a Gambody test models* | |
| Options for support material and raft | |
| Contact Z distance: | 0.15 mm |

| | |
|---|---|
| *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** | **2.50** mm |
| **Pattern angle** | **0.0** ° |
| **Interface layers** | **3** layers |
| **Interface pattern spacing** | **0.00** mm |
| **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/sec |
| **small** | **52.0 - 90.0** mm/sec |
| **external** | **25.0 - 90.0** mm/sec |
| **Infill** | **50.0 - 160.0** mm/sec |
| **solid** | **25.0 - 160.0** mm/sec |

| | |
|---|---|
| top solid | 25.0 - 100.0 mm/sec |
| gaps | 25.0 - 160.0 mm/sec |
| Bridges | 25.0 - 50.0 mm/sec |
| Support material | 50.0 - 200.0 mm/sec |
| interface | 100.0 % |
| Speed for non-print moves | |
| Travel: | 100 - 250 mm/sec |
| Modifiers | |
| First layer speed | 15.00 - 30.00 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.* | |
| Perimeters: | 500 - 3000 mm/sec² |
| Infill: | 500 - 3000 mm/sec² |

| | |
|---|---|
| Bridge: | 500 - 3000 mm/sec² |
| First layer: | 500 - 1000 mm/sec² |
| Default: | 500 - 4000 mm/sec² |
| Advanced | |
| Extrusion width | |
| Default extrusion width: auto | ✔ |

### Filament Settings Tab

| | |
|---|---|
| Filament | |
| Filament | |
| Diameter | 0.40 mm |
| Extrusion multiplier | 1.00 |
| *You have to calibrate this parameter from 0.9 to 1.1 according to the capabilities of your printer and your filament, using a Gambody test models.* | |
| Temperature | |
| *Check your filament manufacturer's temperature recommendations on the spool.* | |
| Extruder first layer | 220.0 °C |

| | |
|---|---|
| Extruder other layers | 220.0 °C |
| Bed first layer | 60.0 °C |
| Bed other layers | 60.0 °C |
| Cooling | |
| *Cooling parameters depends on the material you use for printing.* | |
| Enable | |
| Keep fan always on | ✔ |
| Fan settings | |
| Fan speed | 80.0 - 100.0 % |
| Bridges fan speed | 100.0 % |
| Disable fan for the first | 1 layers |
| Cooling thresholds | |
| Enable fan if layer print time is below | 60.0 sec |
| Slow down if layer print time is below | 5.0 sec |
| Min print speed | 10.0 mm/sec |

### Printer Settings Tab

| | |
|---|---|
| General | |
| Size and coordinates | |

| | |
|---|---|
| **Z offset** | **0.00** mm |
| *Calibrate this value if you need to reduce or improve the adhesion between the plastic and the heat bed* | |
| **Capabilities** | |
| **Extruders** | **1** |
| **Has heated bed** | ✔ |
| **Extruder** | |
| **Size** | |
| **Nozzle diameter:** | **0.40** mm |
| Your current nozzle diameter | |
| **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.* | |
| **Length** | **1.00** mm |
| **Lift Z** | **0.30** mm |
| **Speed** | **45.00** mm/sec |
| **Extra length on restart** | **0.00** mm |
| **Minimum travel after retraction** | **2.00** mm |

Best regards,
your Ge team