# Orca Slicer

Gambody printing recommendations for:

Emerald Tablet of Hermes – Illuminated 3D Replica – Commercial License

Below you'll find detailed slicing settings for Orca Slicer 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!

## Quality Tab

| | |
|---|---|
| **Layer height** | |
| **Layer Height** | **0.20 - 0.20** mm |
| **First layer height** | **0.20 - 0.20** mm |
| **Line width** | |

| | |
|---|---|
| Default: | 0.42 mm |
| Initial Layer | 0.50 - 0.50 mm |
| Outer wall | 0.42 mm |
| Inner wall | 0.45 mm |
| Top surface | 0.42 mm |
| Sparse infill | 0.45 mm |
| Internal solid infill | 0.42 mm |
| Support | 0.42 mm |
| Seam | |
| Seam position | Aligned |
| Precision | |
| Slice gap closing radius | 0.490 mm |
| Resolution | 0.120 mm |
| Arc fitting | ✔ |
| Ironing | |
| Ironing Type | No ironing |
| Wall generator | |
| Wall generator | Classic |

| | |
|---|---|
| Walls and surfaces | |
| Walls printing order | inner/outer |
| Print infill first | ✘ |
| Only one wall on top surfaces | ✔ |
| Bridging | |
| Bridge flow ratio | 1.00 - 1.00 |
| Internal bridge flow ratio | 1.00 - 1.00 |

## Strength

| | |
|---|---|
| Walls | |
| Wall loops | 4 - 4 |
| Top/bottom shells | |
| Top shell layers | 5 layers |
| Top shell thickness | 0.20 mm |
| Top surface pattern | Monotonic |
| Bottom shell layers | 3 layers |
| Bottom shell thickness | 0.20 mm |
| Infill | |

| | |
|---|---|
| Sparse infill density | 100.0 % |

## Speed

| | |
|---|---|
| First layer speed | |
| First layer | 0.5 - 0.5 mm/sec |
| First layer infill | 105.0 - 105.0 mm/sec |
| Other layers speed | |
| Outer wall | 200.0 - 200.0 mm/sec |
| Inner wall | 300.0 - 300.0 mm/sec |
| Small perimeters | 50.0 % |
| Small perimeter threshold | 0.0 mm |
| Sparse infill | 270.0 - 270.0 mm/sec |
| Internal solid infill | 250.0 - 250.0 mm/sec |
| Top surface | 200.0 - 200.0 mm/sec |

| | |
|---|---|
| Bridge | 0.0 mm/sec |
| Gap infill | 50.0 mm/sec |
| Support | 30.0 mm/sec |
| Support interface | 10.0 mm/sec |
| Overhang speed | |
| Bridge external | 50.0 - 50.0 mm/sec |
| Bridge internal | 50.0 mm/sec |
| Travel speed | |
| Travel | 700.0 - 700.0 mm/sec |
| Acceleration | |
| Normal printing | 6000.0 mm/sec² |
| Outer wall | 5000.0 mm/sec² |
| Inner wall | 0.0 mm/sec² |
| Sparse infill | 100.0 % |
| First layer | 500.0 mm/sec² |
| Top surface | 2000.0 mm/sec² |
| Travel | 700.0 mm/sec² |

## Support

| | |
|---|---|
| Support | |
| Enable support | ✔ |
| Type | Tree (auto) |
| Style | Default |
| Threshold angle | 40.0 ° |
| On build plate only | ✘ |
| Support critical regions only (only for tree supports) | ✘ |
| Remove small overhangs | ✘ |
| Filament for Supports | |
| Support/raft base | default |
| Support/raft interface | default |
| Support ironing | |
| Ironing Support Interface | ✘ |
| Advanced | |
| Top Z distance | 0.28 - 0.28 mm |
| Base pattern spacing | 2.50 mm |

| | |
|---|---|
| Pattern angle | 0.0 ° |
| Top interface layers | 3 layers |
| Bottom interface layers | 2 layers |
| Top interface spacing | 0.10 mm |
| Normal Support expansion | 0.00 mm |
| Support/object xy distance | 0.35 mm |

Best regards,

your Ge team