# Orca Slicer 2.0+

Gambody printing recommendations for:

Howl & Sophie 3D Printer Files | Assembly

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

## Quality Tab

| | |
|---|---|
| **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** | **0.20 - 0.28** mm |

| | |
|---|---|
| *120-150% of your Layer Height* | |
| **Line width** | |
| **Default:** | **0.42** mm |
| **Initial Layer** | **0.45 - 0.50** mm |
| **Outer wall** | **0.42** mm |
| **Inner wall** | **0.45** mm |
| **Top surface** | **0.45** mm |
| **Sparse infill** | **0.45** mm |
| **Internal solid infill** | **0.45** mm |
| **Support** | **0.42** mm |
| **Seam** | |
| **Seam position** | **Aligned** |
| *But you can paint the seam if you want.* | |
| **Seam gap** | **10.0** % |
| **Role base wipe speed** | ✔ |
| **Wipe speed** | **80.0** % |
| **Precision** | |
| **Slice gap closing radius** | **0.049** mm |

| | |
|---|---|
| Resolution | 0.012 mm |
| X-Y hole compensation | 0.01 mm |
| 0.01-0.05 *You have to calibrate this parameter* | |
| X-Y contour compensation | 0.01 mm |
| 0.01-0.05 *You have to calibrate this parameter* | |
| Elephant foot compensation | 0.10 mm |
| 0.1-0.2 *You have to calibrate this parameter* | |
| Elephant foot compensation layers | 1 layers |
| Ironing | |
| Ironing Type | No ironing |
| Wall generator | |
| Wall generator | Arachne |
| Wall transitioning threshold angle (Arachne setting) | 10.0 ° |
| Wall transitioning filter margin (Arachne setting) | 25.0 % |
| Wall transition length (Arachne setting) | 100.0 % |
| Wall distribution count (Arachne setting) | 1 |
| First layer minimum wall width (Arachne setting) | 85.0 % |
| Minimum wall width (Arachne setting) | 85.0 % |

| | |
|---|---|
| Minimum feature size (Arachne setting) | 25.0 % |
| Minimum wall length (Arachne setting) | 0.50 mm |
| Walls and surfaces | |
| Walls printing order | inner/outer |
| Wall loop direction | Auto |
| Top surface flow ratio | 1.00 |
| Bottom surface flow ratio | 1.00 |
| Only one wall on top surfaces | ✔ |
| One wall threshold | 300.0 % |
| Bridging | |
| Bridge flow ratio | 0.95 - 1.05 |
| Internal bridge flow ratio | 0.95 - 1.05 |
| External bridge density | 100.0 % |
| Internal bridge density | 100.0 % |
| Thick internal bridges | ✔ |
| Extra bridge layers (beta) | Disabled |
| Filter out small internal bridges | Filter |
| Bridge counterbore holes | None |

| | |
|---|---|
| Overhangs | |
| Detect overhang walls | ✔ |

## Strength

| | |
|---|---|
| Walls | |
| Wall loops | |
| *For pins and power elements of the structure, such as the vehicle frame, use 3 loop* | |
| Detect thin walls | |
| *Disabled for vehicles and ships, enabled for characters* | |
| Top/bottom shells | |
| Top shell layers | 5 layers |
| *For 0,2 Layer Height* | |
| Top shell thickness | 1.00 mm |
| Top surface density | 100 % |
| Top surface pattern | Monotonic line |
| Bottom shell layers | 5 layers |
| *For 0,2 Layer Height* | |
| Bottom shell thickness | 1.00 mm |

| | |
|---|---|
| Bottom surface density | 100.0 % |
| Bottom surface pattern | Monotonic |
| Top/Bottom solid infill/wall overlap | 25.0 % |
| Infill | |
| Sparse infill density | 5.0 % |
| Fill Multiline | 1.00 |
| Sparse infill pattern | Triangles |
| Sparse infill direction | 45.0 ° |
| Maximum length of the infill anchor | 20.0 mm |
| Sparse infill anchor length | 400.0 % |
| Internal solid infill pattern | Monotonic |
| Solid infill direction | 45.0 ° |
| Apply gap fill | Nowhere |
| Filter out tiny gaps | 0.00 mm |
| Infill/wall overlap | 10.0 - 25.0 % |
| Advanced | |
| External bridge infill direction | 0.0 ° |
| Internal bridge infill direction | 0.0 ° |

| | |
|---|---|
| Minimum sparse infill threshold | 15.0 mm² |
| Detect narrow internal solid infill | ✔ |
| Ensure vertical shell thickness | All |

## 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 Kobra 3 Or Bambulab A1, you can select the maximum recommended values.*

| | |
|---|---|
| First layer speed | |
| First layer | 15.0 - 45.0 mm/sec |
| First layer infill | 35.0 - 75.0 mm/sec |
| Initial layer travel speed | 50.0 % |
| Number of slow layers | 2 layers |
| Other layers speed | |
| Outer wall | 30.0 - 150.0 mm/sec |
| Inner wall | 30.0 - 250.0 mm/sec |

| | |
|---|---|
| Small perimeters | 50.0 % |
| Small perimeter threshold | 0.1 mm |
| Sparse infill | 50.0 - 250.0 mm/sec |
| Internal solid infill | 50.0 - 250.0 mm/sec |
| Top surface | 30.0 - 150.0 mm/sec |
| Gap infill | 30.0 - 200.0 mm/sec |
| Slow down by height | ✘ |
| Overhang speed | |
| Slow down for overhangs | ✔ |
| Slow down for curled perimeters | ✔ |
| Overhang speed 10%, 25% | 0.0 mm/sec |
| Overhang speed 25%, 50% | 50.0 mm/sec |
| Overhang speed 50%, 75% | 30.0 mm/sec |
| Overhang speed 75%, 100% | 10.0 mm/sec |
| Bridge external | 20.0 - 40.0 mm/sec |

| | |
|---|---|
| Bridge internal | 30.0 mm/sec |
| Travel speed | |
| Travel | 100.0 - 300.0 mm/sec |
| Acceleration | |
| *Settings for advanced users, change these parameters only if you have sufficient 3D printing expertise* | |
| Normal printing | 5000.0 mm/sec² |
| Outer wall | 3000.0 mm/sec² |
| Inner wall | 3000.0 mm/sec² |
| Bridge | 50.0 % |
| Sparse infill | 100.0 % |
| Internal solid infill | 100.0 % |
| First layer | 300.0 mm/sec² |
| Top surface | 3000.0 mm/sec² |
| Travel | 5000.0 mm/sec² |
| accel to decel | 50.0 % |
| Jerk(XY) | |
| Default | 9.0 mm/sec |

| | |
|---|---|
| Outer wall | 7.0 mm/sec |
| Inner wall | 7.0 mm/sec |
| Infill | 10.0 mm/sec |
| Top surface | 7.0 mm/sec |
| First layer | 9.0 mm/sec |
| Travel | 12.0 mm/sec |

## Support

| | |
|---|---|
| Support | |
| Enable support | ✔ |
| *Enable this parameter if your model requires supports* | |
| Type | Tree (auto) |
| Style | Default |
| Threshold angle | 60.0 ° |
| *We also recommend placing and removing supports manually in some places using special button* | |
| Threshold overlap | 50.0 % |
| First layer density | 90.0 % |
| First layer expansion | 2.0 mm |

| | |
|---|---|
| Remove small overhangs | ✔ |
| **Raft** | |
| Raft layers | 0 layers |
| **Advanced** | |
| Top Z distance | 0.20 - 0.25 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* | |
| Bottom Z distance | 0.20 - 0.25 mm |
| *Bottom Z distance = 1-1.3 layer Height. If the supports are hard to remove, try increasing this setting by 0.1-0,4 mm* | |
| Support wall loops | 0 |
| Base pattern | Rectilinear |
| Base pattern spacing | 2.50 mm |
| Pattern angle | 0.0 ° |
| Top interface layers | 2 layers |
| Bottom interface layers | 2 layers |
| Interface pattern | Rectilinear |
| Top interface spacing | 0.00 mm |
| Bottom interface spacing | 0.00 mm |

| | |
|---|---|
| Normal Support expansion | 0.00 mm |
| Support/object xy distance | 0.40 mm |
| *Increase this parameter if the supports are hard to remove from walls* | |
| Support/object first layer gap | 0.35 mm |
| Independent support layer height | ✔ |
| Tree Support (only for tree supports) | |
| Tip Diameter | 0.80 mm |
| Tree support branch distance | 1.00 mm |
| Branch Density | 30.0 % |
| Tree support branch diameter | 2.0 mm |
| Branch Diameter Angle | 5.0 ° |
| Tree support branch angle | 40.0 ° |
| Preferred Branch | 25.0 ° |

## Others

| | |
|---|---|
| Skirt | |
| Skirt loops | 0 |
| Skirt type | Combined |

| | |
|---|---|
| Skirt minimum extrusion length | 0.00 mm |
| Skirt distance | 2.00 mm |
| Skirt start point | -135.0 ° |
| Skirt speed | 50.0 mm/sec |
| Skirt height | 1.0 layers |
| *For PLA and PETG filament types* | |
| Draft shield | Disabled |
| Brim | |
| Brim type | Outer and inner brim |
| Brim width | 5.0 mm |
| *5-8 mm is optional for small prints that have bad adhesion to the build plate* | |
| Brim-object gap | 0.01 - 0.12 mm |
| Special mode | |
| Slicing Mode | Regular |
| Print sequence | By layer |
| Intra-layer order | Default |
| G-code output | |
| Label objects | ✔ |

| | |
|---|---|
| Filename format | {input _filename_ base}_{print_time}.gcode |

## Filament Tab

| | |
|---|---|
| Filament | |
| Basic information | |
| Type | PLA |
| Shrinkage (XY) | 100.0 % |
| Shrinkage (Z) | 100.0 % |
| Softening temperature | 60.0 °C |
| *Read the description on your filament roll* | |
| Idle temperature | 0.0 °C |
| Recommended nozzle temperature | 190.0 - 270.0 °C |
| *Read the description on your filament roll and increase this parameter for fast printers* | |
| Flow ratio and Pressure Advance | |
| Flow ratio | 0.90 - 1.10 |
| *You have to calibrate this parameter* | |
| Print chamber temperature | |

| | |
|---|---|
| Chamber temperature | 0.0 °C |
| Print temperature | |
| Nozzle | |
| *Read the description on your filament roll and increase this parameter for fast printers* | |
| First layer | 190.0 - 270.0 °C |
| Other layers | 190.0 - 270.0 °C |
| Bed temperature | |
| *Read the description on your filament roll* | |
| Cool Plate (SuperTack) | |
| First layer | 45.0 °C |
| Other layers | 45.0 °C |
| Cool Plate | |
| First layer | 35.0 °C |
| Other layers | 35.0 °C |
| Textured Cool Plate | |
| First layer | 40.0 °C |
| Other layers | 40.0 °C |
| Engineering Plate | |

| | |
|---|---|
| First layer | 0.0 °C |
| Other layers | 0.0 °C |
| Smooth PEI Plate / High Temp Plate | |
| First layer | 55.0 °C |
| Other layers | 55.0 °C |
| Textured PEI Plate | |
| First layer | 55.0 °C |
| Other layers | 55.0 °C |
| Volumetric speed limitation | |
| Max volumetric speed | 21.0 mm/sec |
| Cooling | |
| Cooling for specific layer | |
| No cooling for the first | 1.0 layers |
| Full fan speed at layer | 3.0 layers |
| Part cooling fan | |
| Min fan speed threshold | |
| Fan speed | 70.0 % |
| Layer time | 80.0 sec |

| | |
|---|---|
| Max fan speed threshold | |
| Fan speed | 80.0 % |
| Layer time | 6.0 sec |
| Keep fan always on | ✔ |
| Slow printing down for better layer cooling | ✔ |
| Min print speed | 9 mm/sec |
| Force cooling for overhangs and bridges | ✔ |
| Overhang cooling activation threshold | 50.0 % |
| Overhangs and external bridges fan speed | 100.0 % |
| Internal bridges fan speed | -1.0 % |
| Support interface fan speed | -1.0 % |
| Ironing fan speed | -1.0 % |
| Auxiliary part cooling fan | |
| Fan speed | 70.0 % |
| Exhaust fan | |
| During print | 69.9 % |
| Complete print | 70.0 % |

## Printer Settings Tab

| | |
|---|---|
| **Basic information** | |
| *This field is filled in according to your printer specifications when you add it to the slicer.* | |
| **Machine G-code** | |
| *You can add custom G-code here for the start and end of the print. However, be careful - this is for advanced users only!* | |
| **Extruder 1** | |
| **Size** | |
| **Nozzle diameter** | **0.4** mm |
| **Layer height limits** | |
| **Min** | **0.1** mm |
| **Max** | **0.3** mm |
| **Retraction** | |
| *You have to calibrate your printer using Ge retraction test models* | |
| **Length** | **0.6** mm |
| *Retraction Length: For direct-drive setups use 0.5 mm to 2.5 mm; for Bowden extruders use 5 to 7 mm* | |
| **Extra length on restart** | **0.00** mm |
| **Retraction speed** | **40** mm/sec |

| | |
|---|---|
| Deretraction speed | 40.0 mm/sec |
| *This is how fast the filament is pulled back—40-60 mm/s for direct drive and 30-50 mm/s for Bowden setups.* | |
| Travel distance threshold | 1.00 mm |
| Retract on layer change | ✔ |
| Wipe while retracting | ✔ |
| Wipe distance | 2.00 mm |
| Retract amount before wipe | 70.0 % |
| Z-Hop | |
| On surfaces | All surfaces |
| Z-hop type | Auto |
| Z-hop height | 0.30 mm |
| *You have to calibrate this parameter: Reduce it until the printer starts to hit the parts with the nozzle during printing, then increase it by 0.2.* | |
| Traveling angle | 3.0 ° |
| Only lift Z above | 0.00 mm |
| Only lift Z below | 0.00 mm |
| Retraction when switching material | |
| Length | 1.00 mm |

| | |
|---|---|
| **Extra length on restart** | **0.00** mm |

*Best regards,*

*your Ge team*