

# Orca Slicer 2.0+

Gambody printing recommendations for:  
Ghostface Scream 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](mailto: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

<b>Layer height</b>	
<b>Layer Height</b>	<b>0.12 - 0.20 mm</b>
<i>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.</i>	
<b>First layer height</b>	<b>0.20 - 0.28 mm</b>

*120-150% of your Layer Height*

<b>Line width</b>	
<b>Default:</b>	0.42 mm
<b>Initial Layer</b>	0.45 - 0.50 mm
<b>Outer wall</b>	0.42 mm
<b>Inner wall</b>	0.45 mm
<b>Top surface</b>	0.45 mm
<b>Sparse infill</b>	0.45 mm
<b>Internal solid infill</b>	0.45 mm
<b>Support</b>	0.42 mm
<b>Seam</b>	
<b>Seam position</b>	Aligned
<i>But you can paint the seam if you want.</i>	
<b>Seam gap</b>	10.0 %
<b>Role base wipe speed</b>	✓
<b>Wipe speed</b>	80.0 %
<b>Precision</b>	
<b>Slice gap closing radius</b>	0.049 mm

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

Minimum feature size (Arachne setting)	25.0 %
Minimum wall length (Arachne setting)	0.50 mm
<b>Walls and surfaces</b>	
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 %
<b>Bridging</b>	
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

<b>Overhangs</b>	
<b>Detect overhang walls</b>	✓

## Strength

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

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

Minimum sparse infill threshold	15.0 mm <sup>2</sup>
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

<b>Bridge internal</b>	<b>30.0 mm/sec</b>
<b>Travel speed</b>	
<b>Travel</b>	<b>100.0 - 300.0 mm/sec</b>
<b>Acceleration</b>	
<i>Settings for advanced users, change these parameters only if you have sufficient 3D printing expertise</i>	
<b>Normal printing</b>	<b>5000.0 mm/sec<sup>2</sup></b>
<b>Outer wall</b>	<b>3000.0 mm/sec<sup>2</sup></b>
<b>Inner wall</b>	<b>3000.0 mm/sec<sup>2</sup></b>
<b>Bridge</b>	<b>50.0 %</b>
<b>Sparse infill</b>	<b>100.0 %</b>
<b>Internal solid infill</b>	<b>100.0 %</b>
<b>First layer</b>	<b>300.0 mm/sec<sup>2</sup></b>
<b>Top surface</b>	<b>3000.0 mm/sec<sup>2</sup></b>
<b>Travel</b>	<b>5000.0 mm/sec<sup>2</sup></b>
<b>accel to decel</b>	<b>50.0 %</b>
<b>Jerk(XY)</b>	
<b>Default</b>	<b>9.0 mm/sec</b>

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	<input checked="" type="checkbox"/>
<i>Enable this parameter if your model requires supports</i>	
Type	Tree (auto)
Style	Default
Threshold angle	60.0 °
<i>We also recommend placing and removing supports manually in some places using special button</i>	
Threshold overlap	50.0 %
First layer density	90.0 %
First layer expansion	2.0 mm

<b>Remove small overhangs</b>	✓
<b>Raft</b>	
<b>Raft layers</b>	0 layers
<b>Advanced</b>	
<b>Top Z distance</b>	0.20 - 0.25 mm
<i>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</i>	
<b>Bottom Z distance</b>	0.20 - 0.25 mm
<i>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</i>	
<b>Support wall loops</b>	0
<b>Base pattern</b>	Rectilinear
<b>Base pattern spacing</b>	2.50 mm
<b>Pattern angle</b>	0.0 °
<b>Top interface layers</b>	2 layers
<b>Bottom interface layers</b>	2 layers
<b>Interface pattern</b>	Rectilinear
<b>Top interface spacing</b>	0.00 mm
<b>Bottom interface spacing</b>	0.00 mm

Normal Support expansion	0.00 mm
Support/object xy distance	0.40 mm
<i>Increase this parameter if the supports are hard to remove from walls</i>	
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

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

Filename format	{input_filename_base}_{print_time}.gcode
-----------------	--

## Filament Tab

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

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

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

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

## Printer Settings Tab

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

<b>Deretraction speed</b>	<b>40.0 mm/sec</b>
<i>This is how fast the filament is pulled back—40-60 mm/s for direct drive and 30-50 mm/s for Bowden setups.</i>	
<b>Travel distance threshold</b>	<b>1.00 mm</b>
<b>Retract on layer change</b>	<input checked="" type="checkbox"/>
<b>Wipe while retracting</b>	<input checked="" type="checkbox"/>
<b>Wipe distance</b>	<b>2.00 mm</b>
<b>Retract amount before wipe</b>	<b>70.0 %</b>
<b>Z-Hop</b>	
<b>On surfaces</b>	<b>All surfaces</b>
<b>Z-hop type</b>	<b>Auto</b>
<b>Z-hop height</b>	<b>0.30 mm</b>
<i>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.</i>	
<b>Traveling angle</b>	<b>3.0 °</b>
<b>Only lift Z above</b>	<b>0.00 mm</b>
<b>Only lift Z below</b>	<b>0.00 mm</b>
<b>Retraction when switching material</b>	
<b>Length</b>	<b>1.00 mm</b>

Extra length on restart

0.00 mm

*Best regards,  
your Ge team*