Orca Slicer 2.0+

Gambody printing recommendations for:

Jurassic Park 25th Anniversary in Diorama 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