

Orca Slicer 2.3.0 settings

By Gambody v 1.0



These basic 3D printing settings recommendations for beginners were tested in Orca Slicer **2.3.0**. Test models were printed on **Bambu lab A1**, **Bambu lab A1 mini**, **Crealitty Ender 3 v3**, **Anycubic Kobra 3** with PLA and PETG filaments.

Note:

- You can upscale your 3D printing models. [Downscaling is not recommended](#) - it can make smaller parts of the model unprintable, distort the model's level of detail and result in assembly issues.
- All connectors should be printed at 80-90% Infill.
- When printing Lock connectors, we recommend setting the "Brim width" parameter to 0 in the "Skirt and Brim" print settings. In that way, the Locks will be 3D printed with a Skirt only.

*If you are using **Bambu Lab** printers series, change only those parameters indicated by*



In order to see all the parameters listed below, you need to enable the **Advanced**  mode in the Orca Slicer settings.

Quality Tab

Layer height:

Layer Height: 0.1-0.2 mm

First layer height: 0.24-0.28 mm *(120-150% of your Layer Height)*

Line width:

Default: 0.42 mm

Initial Layer: 0.45-0.5 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:

Aligned *(But you can paint the seam  if you want.)*

Staggered inner seams: **off**

Seam gap: 10%

Scarf joint seam (beta): none

Role base wipe speed: **on**

Wipe speed: 80%

Wipe on loops: **off**

Wipe before external loop: **off**

Precision:

Slice gap closing radius: 0.049 mm

Resolution: 0.012 mm

Arc fitting: **off**

X-Y hole compensation: 0.01-0.05 *(you have to calibrate this parameter)*

X-Y contour compensation: 0.01-0.05 *(you have to calibrate this*

parameter)

Elephant foot compensation: 0.1-0.2 *(you have to calibrate this*

parameter)

Elephant foot compensation layers: 1

Precise wall: **off**

Precise Z height: **off**

Convert holes to polyholes: **off**

Ironing:

Ironing: No ironing

Wall generator

Wall generator: classic

Walls and surfaces

Walls printing order: Inner/Outer

Print infill first: **off**

Wall loop direction: Auto

Flow ratio: 1

Top surface flow ratio: 1

Bottom surface flow ratio: 1

Only one wall on top surfaces: **on**

One wall threshold: 300%

Only one wall on first layer: **off**

Avoid crossing walls: **off**

Small area flow compensation (beta): **off**

Bridging

Bridge flow ratio: 0.95-1.05

Internal bridge flow ratio: 1

External bridge density: 100%
Internal bridge density: 100%
Thick external bridges: **off**
Thick internal bridges: **on**
Extra bridge layers (beta): Disabled
Filter out small internal bridges: filter
Bridge counterbore holes: none

Overhangs

Detect overhang walls: **on**
Make overhangs printable: **off**
Extra perimeters on overhangs: **off**
Reverse on even: **off**

Strength Tab

Walls

Wall loops: 2-3 (*3 loops when you need good strength*)
Alternative extra wall: **on**
Detect thin wall: **off** (*disabled for vehicles and enabled for characters*)

Top/bottom shells

★ Top shell layers: 6 (*for 0,2 Layer Height*)
Top shell thickness: 1.2
Bottom surface pattern: Monotonic line
★ Bottom shell layers: 6 (*for 0,2 Layer Height*)
★ Bottom shell thickness: 1.2
Bottom surface pattern: Monotonic
Top/Bottom solid infill/wall overlap: 25%

Infill

★ Sparse infill density: 6%
★ Sparse infill pattern: Triangle
Maximum length of the infill anchor: 20%
Sparse infill anchor length: 400%
Internal solid infill pattern: Monotonic
Apply gap fill: Nowhere
Filter out tiny gaps: 0
Infill/wall overlap: 10-25%

Advanced

Sparse infill direction: 45°
Solid infill direction: 45°
Rotate solid infill direction: **on**
External bridge infill direction: 0°
Internal bridge infill direction: 0°
Minimum sparse infill threshold: 10 mm²
Infill combination: **off**

Detect narrow internal solid infill: **on**
Ensure vertical shell thickness: all

Speed Tab

Initial layer 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, you can select the maximum recommended values.

First layer: 15-45 mm/s (check your build plate adhesion)
First layer infill: 35-50 mm/s
Initial layer travel speed: 50%
Number of slow layers: 0

Other layers speed

Outer wall: 25-150 mm/s
Inner wall: 40-150 mm/s
Small perimeters: 50% mm/s or %
Small perimeter: threshold 0 mm
Sparse infill: 50-70 mm/s
Internal solid infill: 40-150 mm/s
Top surface: 30-120 mm/s
Gap infill: 30-100 mm/s

Overhang speed

Slow down for overhangs: **on**
Slow down for curled perimeters: **on**
Overhang speed: 0 mm/s (10%, 25%)
20 mm/s [25%, 50%)
15 mm/s [50%, 75%)
10 mm/s [75%, 100%)
Bridge: 20-40 mm/s External
150% Internal

Travel speed

Travel: 100-300 mm/s

Acceleration

(Settings for advanced users, change these parameters only if you have sufficient 3D printing expertise)

Normal printing: 3000 mm/s²
Outer wall: 2000 mm/s²
Inner wall: 5000 mm/s²
Bridge: 50%
Sparse infill: 100%
Internal solid infill: 100%

First layer: 200 mm/s²
Top surface: 1000 mm/s²
Travel: 5000 mm/s²

Jerk(XY)

Default: 0 mm/s
Outer wall: 9 mm/s
Inner wall: 9 mm/s
Infill: 9 mm/s
Top surface: 9 mm/s
First layer: 9 mm/s
Travel: 12 mm/s

Advanced

Extrusion rate smoothing: 0 mm³/s²

Support Tab

Support

Enable support: **on** *(enable this parameter if your model requires supports)*
Type: normal/tree
Style: default
★ Threshold angle: 20-60° *(We also recommend placing and removing supports manually in some places using  button)*
Threshold overlap: 50%
First layer density: 90%
First layer expansion: 2 mm
On build plate only: **off**
Remove small overhangs: **on**

Raft

Raft layers: 0 layers

Filament for Supports

Support/raft base: default
Support/raft interface: default

Advanced

★ Top Z distance: 0.2-0.25 mm
★ Bottom Z distance: 0.2-0.25 mm
Support wall loops: 0
Base pattern: Default
Base pattern spacing: 2.5 mm
Pattern angle: 0°
Top interface layers: 3
Bottom interface layers: same as top layers
★ Interface pattern: concentric
★ Top interface spacing: 0 mm
Normal Support expansion: 0 mm
★ Support/object xy distance: 1 mm

Support/object first layer gap

★ Don't support bridges: **on**

Multimaterial

Prime tower

Enable: **on**

Width: 30 mm

Prime volume: 45 mm³

Brim width: 4 mm

Ooze prevention

Enable: -//-

Flush options

Flush into objects' infill: **off**

Flush into objects' support: **on**

Advanced

Use beam interlocking: **off**

Interface shells: **off**

Maximum width of a segmented region: 0 mm

Interlocking depth of a segmented region: 0 mm

Others Tab

Skirt

Skirt loops: 0

Skirt type: combined

Skirt minimum extrusion length: 0 mm

Skirt distance: 2 mm

Skirt start point: -135°

Skirt height: 1 layer (for PLA and PETG)

Skirt speed: 50 mm/s

Draft shield: Disabled

Brim

★ Brim type: outer and inner brim

★ Brim width: 5 mm *(5-8 mm is optional for small prints that have bad adhesion to the build plate)*

★ Brim-object gap: 0-0.12 mm

Special mode

Slicing Mode: Regular

Print sequence: By layer

Intra-layer order: Default

Spiral vase: **off**

Timelapse: Traditional

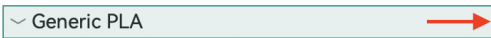
Fuzzy skin: None

G-code output

Reduce infill retraction: **on**

Verbose G-code: **off**
Label objects: **on**
Exclude objects: **off**
Filename format: {input_filename_base}_{print_time}.gcode

Filament Tab

(You can find this parameter here [1](#)  )

Basic information

Type: PLA/PetG
Vendor: Generic
Soluble material: **off**
Support material: **off**
Default color: -//-
Diameter: 1.75 mm
Density: -//-
Shrinkage (XY): 100%
Shrinkage (Z): 100%
Price: -//-
Softening temperature 45°-80° *(read the description on your filament roll)*
Idle temperature: 0°
Recommended nozzle temperature 190°-270° *(read the description on your filament roll and increase this parameter for fast printers)*

Flow ratio and Pressure Advance

Flow ratio: 0.9-1.1 *(you have to calibrate this parameter)*
Enable pressure advance: -//-
Pressure advance: -//-
Enable adaptive pressure advance (beta): -//-

Print chamber temperature

Chamber temperature: 0°
Activate temperature control: **off**

Print temperature

Nozzle: 190°-270° *(read the description on your filament roll and increase this parameter for fast printers)*

Bed temperature:

Plate: 0°-80° *(read the description on your filament roll)*

Volumetric speed limitation

Max volumetric speed: 12mm³/sec

Cooling

No cooling for the first: 1-3 layers

Full fan speed at layer: 0 layer

Min fan speed threshold: Fan speed 60%/ Layer time 80s for PLA

Min fan speed threshold: Fan speed 40%/ Layer time 30s for PetG

Max fan speed threshold: Fan speed 80%/ Layer time 8s for PLA

Max fan speed threshold: Fan speed 90%/ Layer time 12s for PetG

Keep fan always on: **on**

Slow printing down for better layer cooling: **on**

Don't slow down outer walls: **off**

Min print speed: 15 mm/s

Force cooling for overhangs and bridges: **on**

Overhang cooling threshold: 50% for PLA

Overhang cooling threshold: 10% for PetG

Overhangs and external bridges fan speed: 100% for PLA

Overhangs and external bridges fan speed: 90% for PetG

Internal bridges fan speed: -1%

Support interface fan speed: -1%

Auxiliary part cooling fan

Fan speed: 70% for PLA

Fan speed: 0% for PetG

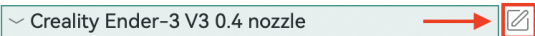
Exhaust fan

Activate air filtration: **off**

During print: 70%

Complete print: 70%

Printer Settings Tab

(You can find this parameter here: )

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!)

Multimaterial: -/-

Extruder 1:

Size

Nozzle diameter: 0.4

Layer height limits

Min: 0.1

Max: 0.35

Position

Extruder offset: x: y:

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 mm

Retraction speed: 40 mm/s

Deretraction speed: 40 mm/s *(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 mm

Retract on layer change: **on**

Retract on top layer: **on**

Wipe while retracting: **on**

Wipe distance: 2 mm

Retract amount before wipe: 70%

Z-Hop

On surfaces: All Surfaces

Z-hop type: Auto

Z-hop height: 0.2-2 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°

Only lift Z above: 0 mm

Only lift Z below: 0 mm

Retraction when switching material

Length: 1 mm

Extra length on restart: 0 mm

Motion ability: -//-

Notes: