Slic3r 1.3.0 settings

Ge

These basic 3D printing settings recommendations for beginners were tested in **Slic3r 1.3.0** software. Test models were printed on **Ultimaker 2, Creality Ender 3, Creality Cr-10S pro v2, Anycubic I3 Mega, Anycubic I3 MegaS, Anycubic Vyper** 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 100% 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.

Print Settings Tab

    Layers and perimeters

        Layer height:

            Layer Height: 0.1-0.2

            First layer height: 0.2-0.3

        Vertical shells:

            Perimeters: 2-3

            Spiral vase: OFF

        Horizontal shells:

            Solid layers: 8 (For 0.1 Layer Height) or 4 (For 0.2 Layer Height)

        Quality (slow slicing)

            Extra perimeters if needed: CHECK

            Avoid crossing perimeters: CHECK

            Detect thin walls: -//-

            Detect bridging perimeters: CHECK

        Advanced:

            Seam position: Aligned

            External perimeters first: -//-

    Infill

Infill

        Fill Density: 10%

        Fill Pattern: Rectilinear

        Top Fill Pattern: Rectilinear

        Bottom fill pattern: Rectilinear

Reducing printing time:

        Combine infill every: 1

        Only infill where needed: -//-

Advanced:

        Fill gaps: CHECK

        Solid Infill every: 0

        Fill angle: 45

        Solid Infill threshold area: 70

        Bridging angle: 0

        Only retract when crossing perimeters: CHECK

        Infill before perimeters: -//-

Skirt and brim

    Skirt

        Loops (minimum): 3

        Distance from Object: 5

        Skirt Height: 1 (for pla and petg)

        Minimal Filament extrusion length: 50

    Brim

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

Support material

    Support Material:

        Generate Support material: -//- (enable this parameter if your model requires supports)

        Overhang threshold: 40-60

        Enforce support for the first: 0

    Raft:

        Raft Layers: 0

    Options for Support material and Raft:

        Contact Z distance: 0.2

        Pattern: Rectilinear

        Pattern spacing: 2.5

        Pattern angle: 0

        Interface Layers: 3

        Interface pattern spacing: 0.2

        Support on build plate only: -//-

        Don't support bridges: CHECK

Speed

Speed for print movies

Perimeters: 20 mm/s

Small perimeters: 20 mm/s

External perimeters: 20 mm/s

Infill: 50 mm/s

Solid: 30 mm/s

Top solid : 20 mm/s

Gaps: 20 mm/s

Bridges: 20 mm/s

Support material: 50 mm/s

Speed for non print movies

Travel: 100 mm/s

Modifiers

First layer speed: 10 mm/s

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

Perimeters: 400 mm/s2

Infill: 400 mm/s2

Bridge: 400 mm/s2

First layer: 300 mm/s2

Default: 400 mm/s2

Auto speed (advanced)

Max print speed: 100 mm/s

Max volumetric speed: 0 mm3/s

Multiple Extruders

These settings only work for 3D printers with multiple extruders (not recommended for beginners)

Advanced

Extrusion width

You can try setting all parameters in this section, except the First layer, to values between 0.75% of your nozzle diameter and 1.25% of your nozzle diameter. Adjusting them will help you work out the optimal parameters for the best quality for your print. As for the First layer, you can set it to 150% of the diameter of your nozzle for better

Overlap

Infill/perimeters overlap: 25%

Flow

Bridge flow ratio: 0.7

Other

-//-

Output options

-//-

Notes

-//-

Shortcuts

-//-

Filament Settings Tab

Filament

Filament

Color: -//-

Diameter: 1.75

Extrusion multiplier: 1

Density: -//-

Cost: -//-

(You can set these parameters yourself if you need)

Cooling

Enable

Keep fan always on: CHECK

Enable auto cooling: -//-

Fan settings

Fan speed

Min: 100% (for PLA, PETG) or 30% (for ABS, SBS)

Max: -//-

Bridges fan speed: 100%

Disable fan for the first: 1

Cooling thresholds

Enable fan if layer print time is below: -//-

Slow down if layer print time is below: -//-

Min print Speed: -//-

Custom G-code

-//-

Notes

-//-

Printer Settings Tab

General

Size and coordinates

Bed shape: your 3D printer's bed size

Z offset: 0

Capabilities

Extruders: 1

Has heated bed: CHECK

Firmware

G-code flavor: RepRap (Marlin/Sprinter)

Advanced

Use relative E distances: CHECK

Custom G-code

Use your 3D printer's standard G-code. If you don't have it, you may use the example below.

Start G-code

M104 S(First_layer_temperature)

M140 S(First_layer_bed_temperature)

G28 S(Parking)

M109 S(waiting_for_warm-up_nozzle)

M190 S(waiting_for_warm-up_bed)

End G-code

M104 s0 ;turn off nozzle

M140 s0 ;turn off bed

G91

G1 Z1

G28 X0 Y0 ;park

M84   ;disable motors

M107 ;disable cooling

Extruder 1

Size

Nozzle diameter: 0.4 mm

Layer height limits

Min: 0.05

Max: 0.3

Position

Extruder offset: -//-

Retraction

Length: 5.5-6.5 mm (1mm for a direct extruder)

Lift Z: 0-0.3 mm

Speed: 30 - 40 mm/s

Extra length on restart: 0 mm

Minimum travel after retraction: 2 mm

Retract on layer change: -//- (you can try to enable this parameter)

Wipe while retracting: -//- (you can try to enable this parameter)