PrusaSlicer 2.3.1 settings

These basic 3D printing settings recommendations for beginners were tested in **PrusaSlicer 2.3.1**. 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.

In order to see all the parameters listed below, you need to enable the **Expert mode** in the PrusaSlicer's settings.

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)
    - Minimum shell thickness: 0.8
- Quality (slower slicing)
    - Extra perimeters if needed: CHECK
    - Ensure vertical shell thickness: -//-
    - 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: Monotonic

    Bottom fill pattern: Monotonic

Reducing printing time:

    Combine infill every: 1

    Only infill where needed: -//-

Advanced:

    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)

    Draft Shield: -//-

    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)

    Auto generated supports: CHECK

    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

    With sheath around the support: CHECK

    Pattern spacing: 2.5

Pattern angle: 0

Interface Layers: 3

Interface pattern spacing: 0.2

Interface loops: -//-

Support on build plate only: -//-

XY separation between an object and its support: 50%

Don't support bridges: CHECK

Synchronize with object layers: -//-

Speed

Speed for print moves

(We recommend that you try printing with the parameters listed below and then adjust the print speed until the printing quality is optimal for you)

Perimeters: 20 mm/s

Small perimeters: 20 mm/s

External perimeters: 20 mm/s

Infill: 50 mm/s

Solid infill: 30 mm/s

Top solid infill: 20 mm/s

Support material: 50 mm/s

Support material interface: 20 mm/s

Bridges: 20 mm/s

Gap fill: 20 mm/s

Ironing: 15 mm/s

Speed for non-print moves

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: 400 mm/s2

Default: 400 mm/s2

Auto speed (advanced)

Max print speed: 100 mm/s

Max volumetric speed: 0 mm3/s

Multiple Extruders

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 adhesion to the build plate (for a nozzle with a diameter of 0.4 mm, the First layer extrusion width can be from 0.3 mm to 0.5 mm)

Overlap

Infill/perimeter overlap: 25%

Flow

Bridge flow ratio: 0.7

Slicing

-//-

Other

Clip multi-part objects: CHECK

Output options

-//-

Notes

-//-

Dependencies

-//-

Filament Settings Tab

Filament

Filament

Color: -//-

Diameter: 1.75

Extrusion multiplier: 1

Density: -//-

Cost: -//-

Spool weight: -//-

(You can set these parameters yourself if you need)

Temperature

Check your filament manufacturer's temperature recommendations on the spool.

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

    Full fan speed at layer: 3-4

Cooling thresholds

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

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

    Min print speed: -//-

Advanced

Print speed override

    Max volumetric speed:

        PLA: 14 MM3/S

        ABS: 10 MM3/S

        PETG: 14 MM3/S

        HIPS: 10 MM3/S

(If you use the Volcano extruder for your printer, all parameters need to be increased x2)

Filament Overrides

    -//-

Custom G-code

    -//-

Notes

    -//-

Dependencies

    -//-

Printer Settings Tab

General

Size and coordinates

    Bed shape: your 3D printer's bed size

    Max print height: your 3D printer's height

    Z offset: 0

Capabilities

    Extruders: 1

    Single Extruder Multi Material: -//-

Firmware

    G-code flavor: RepRap/Sprinter

    G-code thumbnails: -//-

Supports stealth mode: -//-

Supports remaining times: -//-

Advanced

Use relative E distances: CHECK

Use firmware retraction: -//- (needs to be enabled in the firmware of your printer)

Use volumetric E: -//-

Enable variable layer height feature: 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 (1 mm for a direct extruder)

Lift Z: 0-0.3 mm

Retraction Speed: 30 - 40 mm/s

Deretraction Speed: 0 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)
Retract amount before wipe:

Notes
-//-
Dependencies
-//-