My Settings (modified) ▼

Layers and perimeters
Infill
Skirt and brim
Support material
Speed
Multiple Extruders
Advanced
Output options
Notes

Layer height

Layer height:	0.2	mm
First layer height:	0.35	mm or %

Vertical shells

Perimeters:	3	(minimum)
Spiral vase:	☐	

Horizontal shells

Solid layers:	Top: 3	Bottom: 3	

Quality (slower slicing)

Extra perimeters if needed:	☑
Avoid crossing perimeters:	☐
Detect thin walls:	☑
Detect bridging perimeters:	☑

Advanced

Seam position:	Aligned ▼
External perimeters first:	☐

My Settings (modified) ▼

Layers and perimeters
Infill
Skirt and brim
Support material
Speed
Multiple Extruders
Advanced
Output options
Notes

Infill

Fill density:	20 ▼	%
Fill pattern:	Honeycomb ▼	
Top/bottom fill pattern:	Rectilinear ▼	

Reducing printing time

Combine infill every:	1	layers
Only infill where needed:	☐	

Advanced

Solid infill every:	0	layers
Fill angle:	45	°
Solid infill threshold area:	70	mm²
Only retract when crossing perimeters:	☑	
Infill before perimeters:	☐	

Slic3r

Print Settings   **Filament Settings**   **Printer Settings**

I3pro B ABS config ▾ 💾⊖

📁 Filament
⏳ Cooling

Filament

Color: [   ]

Diameter: 1.75   mm

Extrusion multiplier: 1

Temperature (°C)

Extruder: First layer: 235   Other layers: 235

Bed: First layer: 106   Other layers: 106

Print Settings   **Filament Settings**   **Printer Settings**

I3pro B ABS config ▾ 💾⊖

📁 Filament
⏳ Cooling

Enable

Keep fan always on: ☑

Enable auto cooling: ☑

If estimated layer time is below ~30s, fan will run at 100% and print speed
will be reduced so that no less than 30s are spent on that layer (however,
speed will never be reduced below 10mm/s).
If estimated layer time is greater, but still below ~60s, fan will run at a
proportionally decreasing speed between 100% and 35%.
During the other layers, fan will always run at 35% except for the first layer.

Fan settings

Fan speed: Min: 35   %Max: 100   %

Bridges fan speed: 100   %

Disable fan for the first: 1   layers

Cooling thresholds

Enable fan if layer print time is below: 60   approximate seconds

Slow down if layer print time is below: 30   approximate seconds

Min print speed: 10   mm/s

File   Window   Help

Print Settings | Filament Settings | **Printer Settings**

I3pro B ABS config ▾ 💾⛔

- 📇 General
- ⚙ Custom G-code
- 🔻 Extruder 1

Size and coordinates

Bed shape: [⚙ Set...]

Z offset: [0]   mm

Capabilities

Extruders: [1] ▲▼

OctoPrint upload

Host or IP: [   ] [🔍 Browse...] [🔧 Test]

API Key: [   ]

Firmware

G-code flavor: [RepRap (Marlin/Sprinter/Repetier) ▾]

Advanced

Use relative E distances: ☐
Use firmware retraction: ☐
Use volumetric E: ☐
Pressure advance: [0]
Vibration limit (deprecated):   [0]   Hz

Print Settings | Filament Settings | **Printer Settings**

I3pro B ABS config ▾ 💾⛔

- 📇 General
- ⚙ Custom G-code
- 🔻 Extruder 1

Size

Nozzle diameter: [0.4]   mm

Position (for multi-extruder printers)

Extruder offset: x: [0]   y: [0]   mm

Retraction

Length: [2]   mm (zero to disable)
Lift Z: [0]   mm
Speed: [40] ▲▼   mm/s
Extra length on restart: [0]   mm
Minimum travel after retraction: [2]   mm
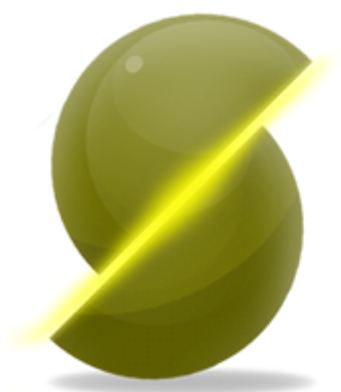Retract on layer change: ☑
Wipe while retracting: ☐

Retraction when tool is disabled (advanced settings for multi-extruder setups)

Length: [10]   mm (zero to disable)
Extra length on restart: [0]   mm

Slic3r

Version 1.2.9

Contributions by Henrik Brix Andersen, Nicolas Dandrimont,
Mark Hindess, Petr Ledvina, Y. Sapir, Mike Sheldrake and
numerous others. Manual by Gary Hodgson. Inspired by the
RepRap community.
Slic3r logo designed by Corey Daniels, Silk Icon Set designed
by Mark James.