

Printer Settings:

Plater Print Settings Filament Settings **Printer Settings**

TEST

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

Layer height

Layer height: 0.1 mm
First layer height: 150% 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:

Plater Print Settings Filament Settings **Printer Settings**

TEST

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

Infill

Fill density: 25 %
Fill pattern: Honeycomb
Top/bottom fill pattern: Octagram Spiral

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:

Plater | Print Settings | Filament Settings | Printer Settings

TEST

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

Skirt

Loops (minimum): 1

Distance from object: 6 mm

Skirt height: 1 layers

Minimum extrusion length: 0 mm

Brim

Brim width: 0.1 mm

Plater | Print Settings | Filament Settings | Printer Settings

TEST

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

Support material

Generate support material:

Overhang threshold: 0 °

Enforce support for the first: 0 layers

Raft

Raft layers: 0 layers

Options for support material and raft

Contact Z distance: 0.2 (detachable) mm

Pattern: pillars

Pattern spacing: 2.5 mm

Pattern angle: 0 °

Interface layers: 3 layers

Interface pattern spacing: 0 mm

Don't support bridges:

Plater | Print Settings | Filament Settings | Printer Settings

TEST

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

Speed for print moves

| | | |
|-----------------------------|------|-----------|
| Perimeters: | 50 | mm/s |
| Small perimeters: | 30 | mm/s or % |
| External perimeters: | 60% | mm/s or % |
| Infill: | 60 | mm/s |
| Solid infill: | 20 | mm/s or % |
| Top solid infill: | 15 | mm/s or % |
| Support material: | 60 | mm/s |
| Support material interface: | 100% | mm/s or % |
| Bridges: | 60 | mm/s |
| Gap fill: | 20 | mm/s |

Speed for non-print moves

Travel: 150 mm/s

Modifiers

First layer speed: 50 mm/s or %

Acceleration control (advanced)

| | | |
|--------------|---|-------------------|
| Perimeters: | 0 | mm/s ² |
| Infill: | 0 | mm/s ² |
| Bridge: | 0 | mm/s ² |
| First layer: | 0 | mm/s ² |
| Default: | 0 | mm/s ² |

Filament Settings:

Plater | Print Settings | Filament Settings | Printer Settings

PLA

- Filament
- Cooling

Filament

Diameter: 2.85 mm

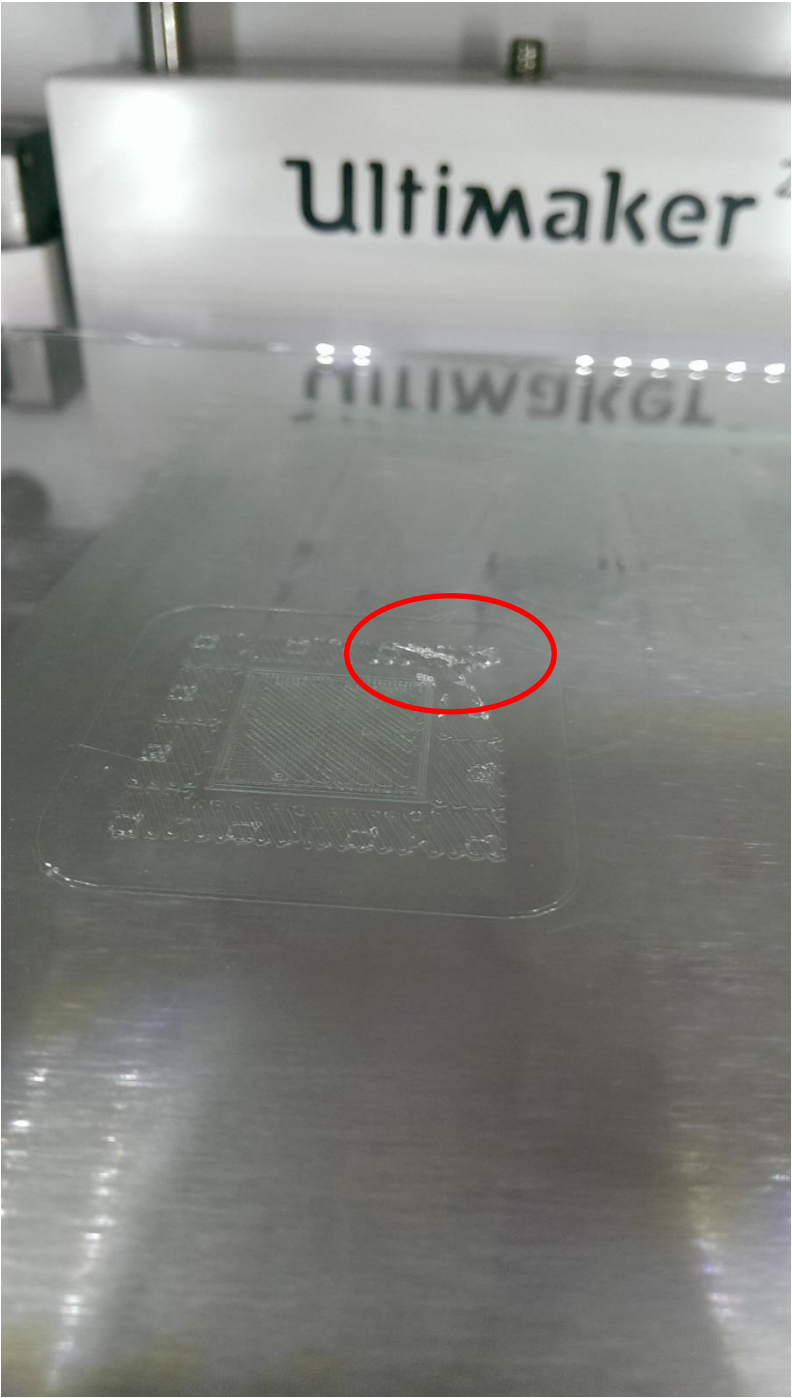
Extrusion multiplier: 1

Temperature (°C)

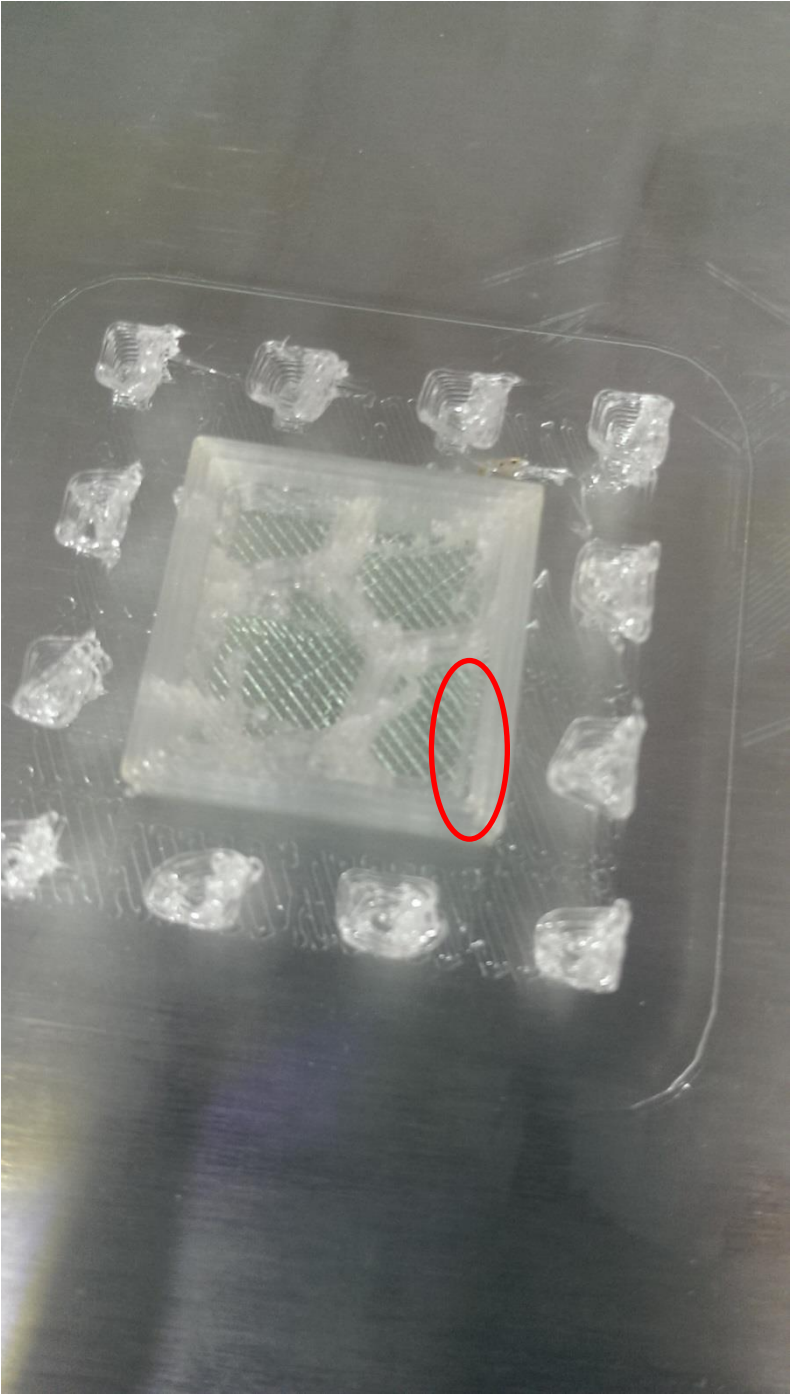
| | | |
|-----------|------------------|-------------------|
| Extruder: | First layer: 230 | Other layers: 210 |
| Bed: | First layer: 65 | Other layers: 60 |

Ergebnis:

Erster Layer haftet nicht richtig:



Hier erkennt man den Spalt und auch das Infill + Support sehen nicht sauber aus:



Hier der Bug. Den Rand hat überhaupt keine Verbindung zum Infill und der Top Layer liegt auch nur über dem Infill und nicht mit über dem Rand:

