

Slicing / Druckergebnis / Probleme

- [*Cura](#)
- [*Druckbetthaftung](#)
- [*Druckbild ist schlecht](#)
- [*Super Slicer](#)
- [Orca / Bambu Slicer 0.2 + 0.8er Nozzle Einstellungen](#)

*Cura

Cura Timelapse

Erweiterungen -> Nachbearbeitung -> G-Code ändern -> Insert at layer change -> When to insert: Before -> G-Code to insert: TIMELAPSE_TAKE_FRAME

Cura Startcode

Cura Startcode V2.0 / worked with and without BLTouch:

```
G21                ; set units to millimeters
G90                ; Set all axes to absolute
M82                ; set extruder to absolute mode
M107               ; start with the fan off
M140 S{material_bed_temperature_layer_0}    ; Start heating the bed
G4 S60             ; wait 1 minute
M104 S{material_initial_print_temperature}  ; start heating the hot end
M190 S{material_bed_temperature_layer_0}    ; wait for bed
M109 S{material_initial_print_temperature}  ; wait for hotend
M300 S1000 P500    ; BEEP heating done
G28                ; move X/Y to min endstops
M117 Bis JETZT ist alles gut!!!
G1 X0 Y-5 F2200    ; Go to front
G0 Z0.15           ; lift nozzle a bit
G92 E0             ; zero the extruded length
G1 X50 E25 F500    ; Extrude 25mm of filament in a 5cm line.
G92 E0             ; zero the extruded length again
G1 E-2 F500        ; Retract a little
G1 X120 F4000      ; Quickly wipe away from the filament line
```

End Code:

```
END_PRINT
```

*Druckbetthaftung

Ablauf vor dem Druckstart:

1. Homing aller Achsen "G28"
2. Aufheizen (Nozzle kann bei 170 bleiben)
3. Z-Tilt
4. Homing Z "G28 Z"
5. Bedmesh vor jedem Druck
6. Primeline / Start Print

Lässt man einen Punkt weg oder verdreht die Reihenfolge, kann es nicht mehr funktionieren

Vorlage Startcode:

Cura Startcode

Startcode im Slicer:

```
START_PRINT BED={material_bed_temperature_layer_0} EXTRUDER={material_initial_print_temperature}
```

Einfügen in Klipper:

```
[gcode_macro START_PRINT]
gcode:
    ##### set defaults #####
    {% set extruder = params.EXTRUDER|default(0) %}
    {% set bed = params.BED|default(0) %}
    ##### end off definition #####
    #SET_TEMPERATURE_FAN_TARGET temperature_fan=Chamber ; VORON or enclosure
    #probe_reset[] ; VYPER Set on if you use strain gauge on vyper
    M83 ; Extruder realtive mode
    #STATUS_HEATING ; NEOPIXEL
    {action_respond_info("Heat Bed")}
    M190 S{bed} ; Bed heat up
    {action_respond_info("Heat Extruder")}
    M109 S{extruder} ; Extruder heat up standby temp 170
```

```

#STATUS_HOMING[] ; NEOPIXEL
G28
#_CG28 ; Homing Check
#STATUS_CALIBRATING_Z
# G34[] ; VYPER ztilt
#_CG34[] ; VYPER zTilt
#G32[] ; VORON QGL
#_CG32 ; VORON QGL check
#G28 Z[] ; activate if you use ztilt ord qgl
#STATUS_MESHING[] ; NEOPIXEL
BED_MESH_CLEAR
BED_MESH_CALIBRATE
[]#BED_MESH_PROFILE LOAD="Name"
#BED_MESH_PRINT_AREA AREA_START_X={params.AREA_START_X|float}
AREA_START_Y={params.AREA_START_Y|float} AREA_END_X={params.AREA_END_X|float}
AREA_END_Y={params.AREA_END_Y|float}
#SET_GCODE_OFFSET Z= ; Offset reset
#NOZZLE_CLEAR[] ; VORON For Nozzle Drive Mod
#NOZZLE_DRIVE ; For Nozzle Drive Mod
G92 E0.0 ; Reset extruder length
G90 ; Absolute positioning

```

Super Slicer

Startcode im Slicer:

```
START_PRINT BED={first_layer_bed_temperature} EXTRUDER={first_layer_temperature}
```

Einfügen in Klipper:

```

[gcode_macro START_PRINT]
gcode:
    ##### set defaults #####
    {% set extruder = params.EXTRUDER|default(0) %}
    {% set bed = params.BED|default(0) %}

```

```

#### end off definition ####

#SET_TEMPERATURE_FAN_TARGET temperature_fan=Chamber ; VORON or enclosure

#probe_reset[] ; VYPER Set on if you use strain gauge on vyper

M83 ; Extruder realtive mode

#STATUS_HEATING ; NEOPIXEL

{action_respond_info("Heat Bed")}

M190 S{bed} ; Bed heat up

{action_respond_info("Heat Extruder")}

M109 S{extruder} ; Extruder heat up standby temp 170

#STATUS_HOMING[] ; NEOPIXEL

G28

#_CG28 ; Homing Check

#STATUS_CALIBRATING_Z

# G34[] ; VYPER ztilt

#_CG34[] ; VYPER zTilt

#G32[] ; VORON QGL

#_CG32 ; VORON QGL check

#G28 Z[] ; activate if you use ztilt ord qgl

#STATUS_MESHING[] ; NEOPIXEL

BED_MESH_CLEAR

BED_MESH_CALIBRATE

[]#BED_MESH_PROFILE LOAD="Name"

#BED_MESH_PRINT_AREA AREA_START_X={params.AREA_START_X|float}

AREA_START_Y={params.AREA_START_Y|float} AREA_END_X={params.AREA_END_X|float}

AREA_END_Y={params.AREA_END_Y|float}

#SET_GCODE_OFFSET Z= ; Offset reset

#NOZZLE_CLEAR[] ; VORON For Nozzle Drive Mod

#NOZZLE_DRIVE ; For Nozzle Drive Mod

G92 E0.0 ; Reset extruder length

G90 ; Absolute positioning

```

End-Code:

Cura

END_PRINT

Super Slicer

END_PRINT

[Link](#)

<https://www.youtube.com/embed/P-HjyB1EbOY>

*Druckbild ist schlecht

Wenn euer Druckbild nicht so gut ist könnt ihr diesen Einstellungsfahrplan befolgen oder unsere Videoreihe zur Filament Kalibrierung schauen. Es ist sehr wichtig das Filament ordentlich zu kalibrieren.

Einstell-Fahrplan <https://www.3d-druck-community.de/showthread.php?tid=25055#pdf>

Ellis Tunning Guide: <https://github.com/AndrewEllis93/Print-Tuning-Guide>

<https://www.youtube.com/embed/dDugYD4ayOU>

<https://www.youtube.com/embed/bw4qSSMjpak>

https://www.youtube.com/embed/YV0nX_FA2Zo

*Super Slicer

Relative Extrusion aktivieren

Druckereinstellungen/Allgemein

The screenshot shows the Super Slicer software interface. The top navigation bar has tabs for '3D view', 'Sliced preview', 'Gcode preview', 'Druckereinstellungen', 'Filamenteinstellungen', and 'Druckereinstellungen'. The 'Druckereinstellungen' tab is highlighted. The left sidebar shows a tree view with 'Allgemein' selected. The main area displays the 'Druckereinstellungen/Allgemein' settings page. The 'Erweiterte Einstellungen' section is expanded, showing the 'Relative Abstände für Extrusion benutzen:' checkbox checked. Other settings include 'Fähigkeiten', 'Hochladen zum Druckhost', 'Firmware', 'Cooling fan', and 'Thumbnails'.

Fähigkeiten

- Extruder: 1
- Einzelextruder mit Multi-Material: ☐
- Milling cutters: 0

Hochladen zum Druckhost

Note: All parameters from this group are moved to the Physical Printer settings (see changelog).

A new Physical Printer profile is created by clicking on the "cog" icon right of the Printer profiles combo box, by selecting the "Add physical printer" item in the Printer combo box. The Physical Printer profile editor opens also when clicking on the "cog" icon in the Printer settings tab. The Physical Printer profiles are being stored into SuperSlicer/physical_printer directory.

Firmware

- G-Code Typ: Klipper
- Unterstützt Stealth Modus: ☒
- Print remaining times: ☒ Method: M73
- Gcode precision: xyz decimals: 3 Extruder decimals: 5
- Processing limit: Maximum G1 per second: 1500 Minimum extrusion length: 0 mm
- Illegal characters: [<>:"'\/\|?]*äü

Cooling fan

- Speedup time: 1 s Only for overhangs: ☒
- Kickstart duration: 0 s
- Fan PWM from 0-100: ☐

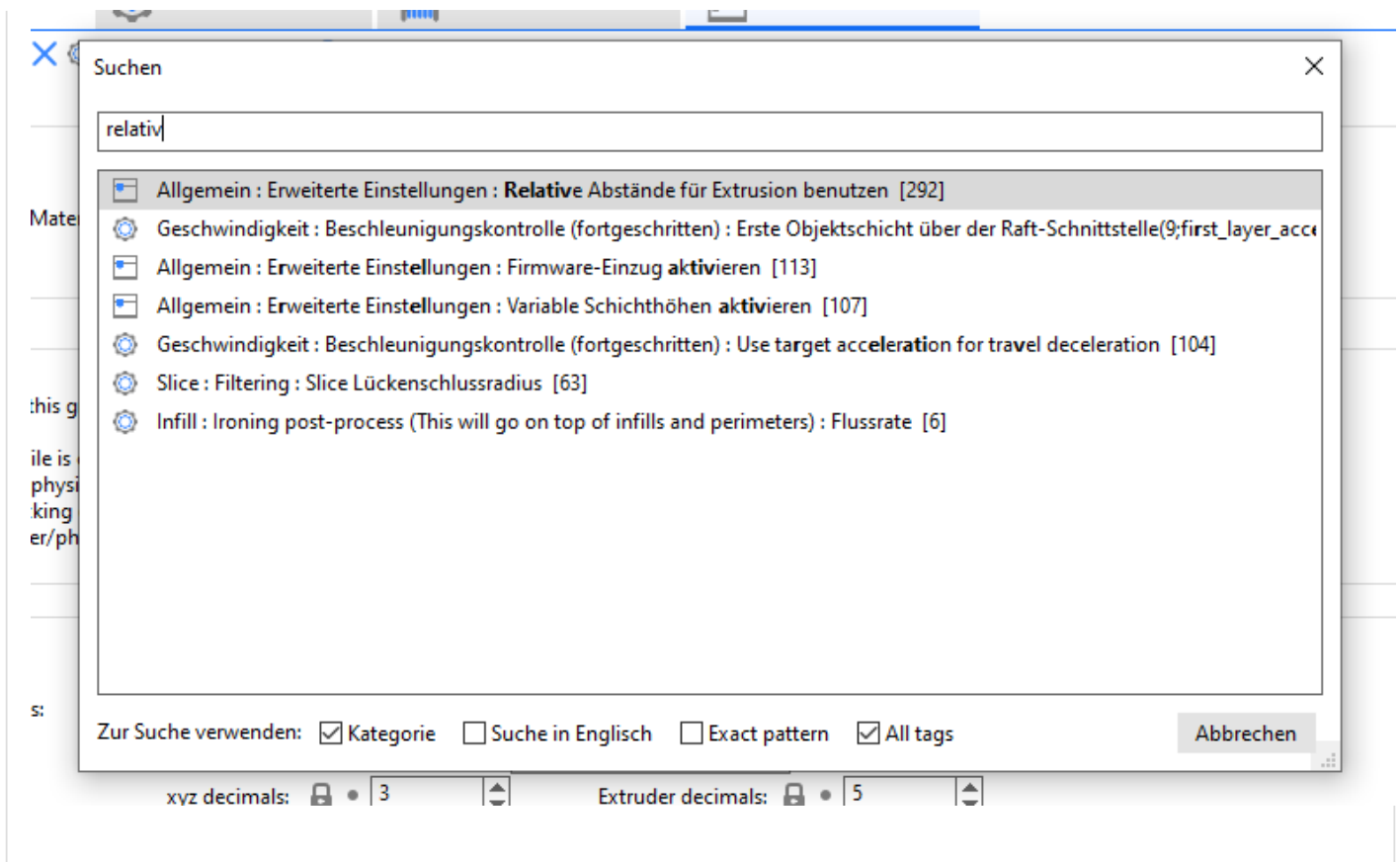
Thumbnails

- Size for Gcode: Small: x: 32 y: 32 Big: x: 400 y: 300
- Color override: ☒ ☐
- Thumbnail options: Format of G-code thumbnails: PNG Bed on thumbnail: ☒ Print at the end: ☐

Erweiterte Einstellungen

- Relative Abstände für Extrusion benutzen: ☒
- Firmware-Einzug aktivieren: ☐
- Volumetrisches E benutzen: ☐
- Variable Schichthöhen aktivieren: ☒

Ihr könnt auch oben das "Lupensymbol" benutzen und "relative" Suchen



Orca / Bambu Slicer 0.2 + 0.8er Nozzle Einstellungen

Hier hab ihr eine grobe Vorlage, wenn ihr mit einer 0.2er und 0.8er nozzle drucken wollt.

Vielen dank an Flashtu, der diese Einstellungen erarbeitet hat.

0.2 mm Nozzle Slicer Software Settings

Layer Height – 0.08 mm First Layer Height – 0.12 mm

Layer/Extrusion Width – 0.15 – 0.25 mm

Extrusion Multiplier – Can be adjusted between 0.95 – 1.05

Speed – Same as with 0.4mm nozzle

Temperature – Same as with 0.4mm nozzle Exotic Materials (Wood, PETG, TPU) and Home Made

Filaments – NOT Recommended (Increased risk of nozzle clogging even with bad quality PLA)

0.8 mm Nozzle Slicer Software Settings

Layer Height – 0.6 mm First Layer Height – 0.6 mm

Layer/Extrusion Width – 0.6 – 1 mm

Extrusion Multiplier – Can be adjusted between 0.95 – 1.05

Speed – Lower than with 0.4mm nozzle Temperature – Slightly higher than with 0.4mm nozzle

Exotic Materials (Wood, PETG, TPU) and Home Made Filaments – Recommended (reduced risk of nozzle clogging)