

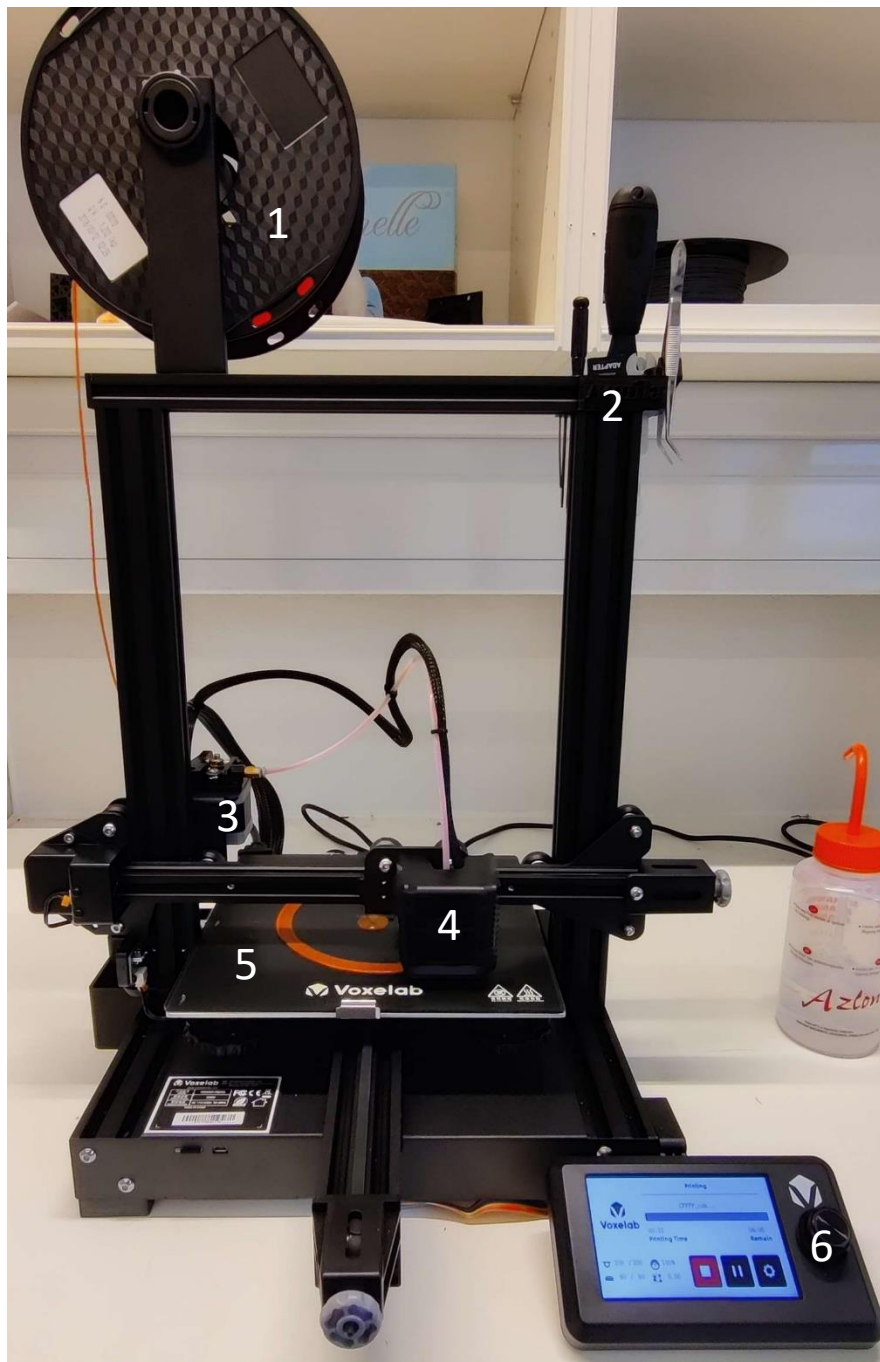
Guide to the Aquila 3D printers at Aabogade 40

Overview

Printer overview.....	2
Setting up the slicer tool.....	3
Getting started.....	6
Changing slicer settings.....	7
Micro stepping.....	7
Object placement.....	8
Filament change.....	9
Github and good 3D printer practices	10
Troubleshooting.....	11
Print not sticking to the bed:	11

Printer overview

A picture of the Voxelab Aquila v2 3D printer can be seen below.



Where **1** is the filament holder loaded with a filament spool, **2** is the tool holder, **3** is the Bowden extruder, **4** is the printer head, **5** is the printer bed and **6** is the LCD display controlled by the knob.

The printer is a copy of the Ender 3 printer. Filament is pushed through the hotend by the Bowden extruder on the bed.

The prints are done by interpreting G-code from a microSD-card.

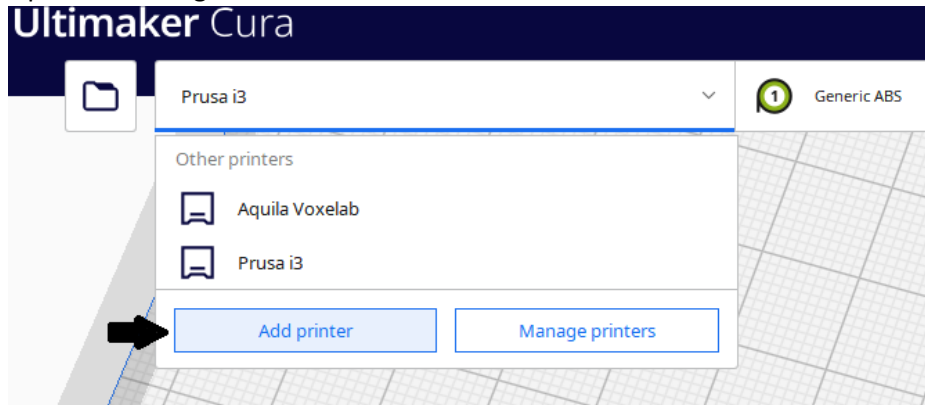
The microSD-cards need to have the FAT32 format and have a byte allocation of 4096

Setting up the slicer tool

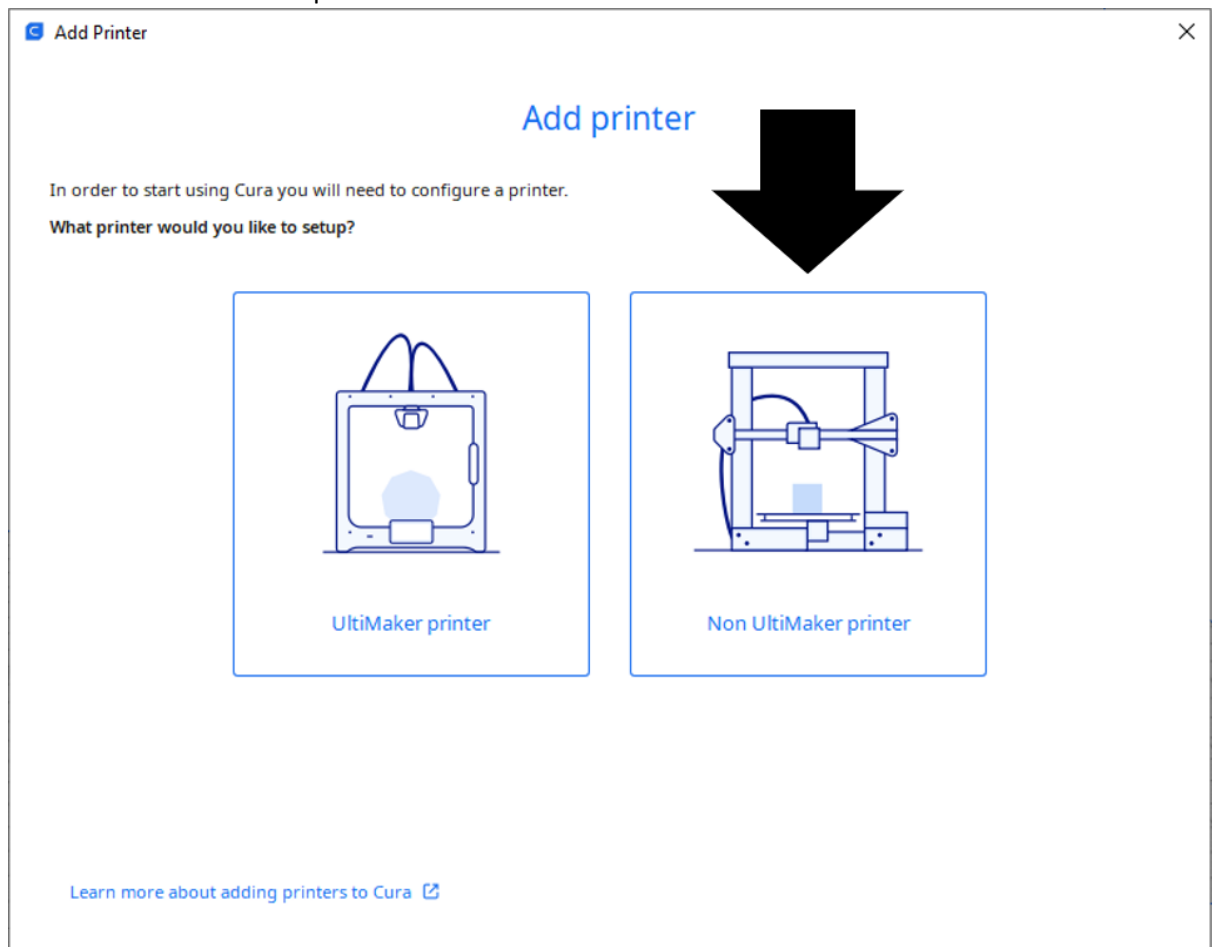
To use this 3D printer, first install the latest version of [UltiMaker Cura](#), a slicing tool for 3D drawings.

Next is to install the custom designed Aquila Voxlab 3D printing profile by following the **7** steps shown below

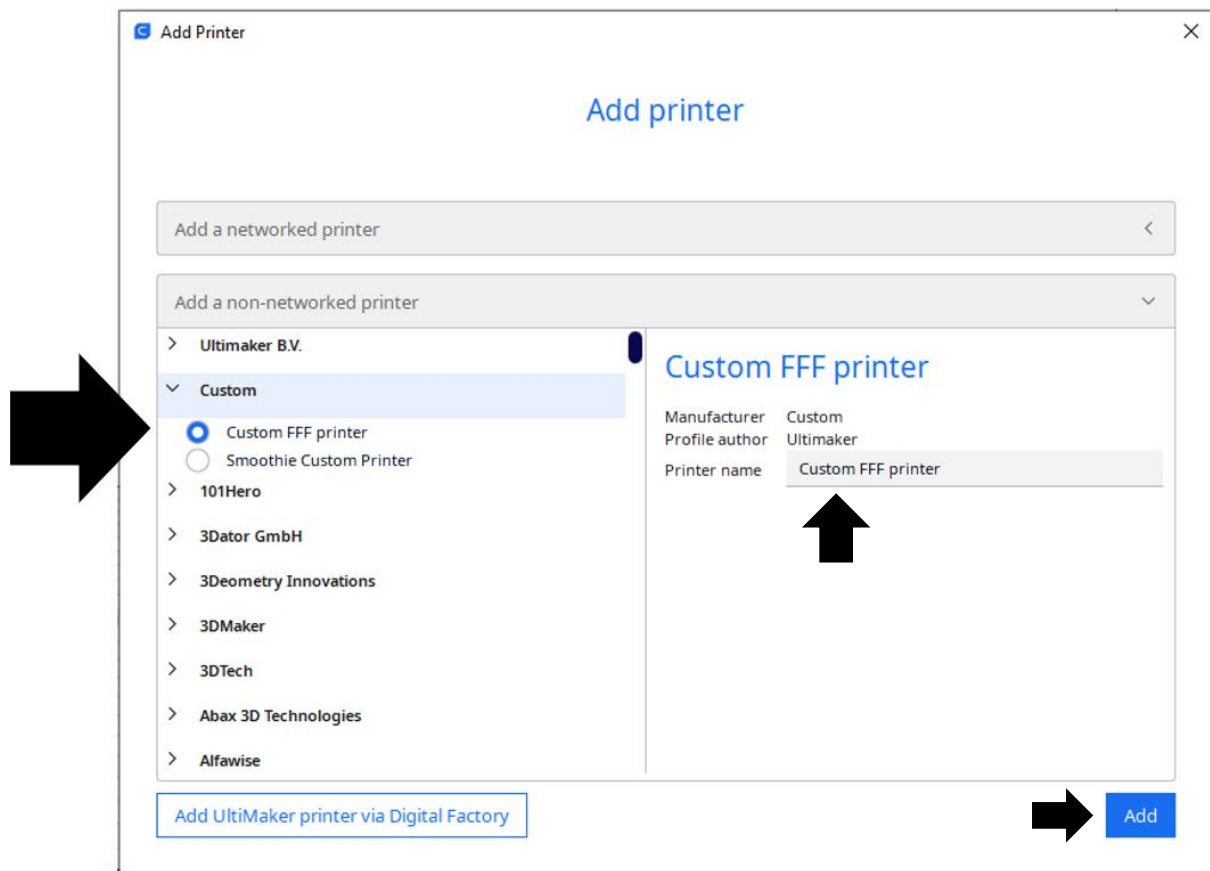
1. Get the printer profile from the micro SD-card, which is either in the toolbox or the printer
2. Open Cura and go to **Add Printer**



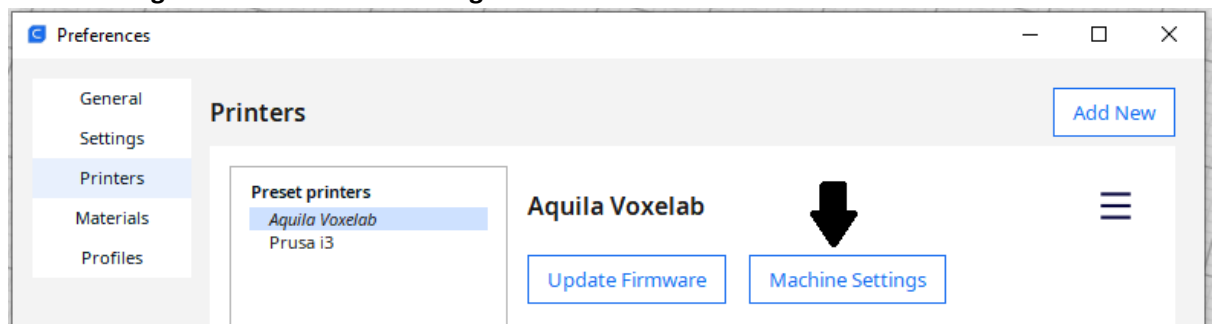
3. Chose the non UltiMaker printer



4. Select **Custom Printer FFF** (second option from the top), rename to **Voxelab Aquila**, then **Add**



5. Go to **Manage Printer>Machine Setting**



6. Apply these settings

Machine Settings

Aquila Voxelab

Printer

Printer Settings

X (Width)

220.0

mm

Y (Depth)

220.0

mm

Z (Height)

250.0

mm

Build plate shape

Rectangular

▼

Origin at center

☐

Heated bed

☒

Heated build volume

☐

G-code flavor

Marlin

▼

Start G-code

G28 ;Home
G1 Z0.0 F6000 ;Be in wipe distance
G92 E0.0 ; reset extruder distance position
G1 Y60.0 E9.0 F1000.0 ; intro line
G1 Y100.0 E21.5 F1000.0 ; intro line
G92 E0.0 ; reset extruder distance position
G1 Z5.0 ;Go slightly up

Extruder 1

Printhead Settings

X min

-30

mm

Y min

-30

mm

X max

30

mm

Y max

30

mm

Gantry Height

30.0

mm

Number of Extruders

1

▼

Apply Extruder offsets to GCode

☒

End G-code

M104 S0
M140 S0
;Retract the filament
G92 E1
G1 E-1 F300
G28 X0 Y0
M84

Machine Settings

Aquila Voxelab

Printer

Nozzle Settings

Nozzle size

0.4

mm

Compatible material diameter

1.75

mm

Nozzle offset X

0.0

mm

Nozzle offset Y

0.0

mm

Cooling Fan Number

0

Extruder Start G-code

Extruder End G-code

Extruder 1

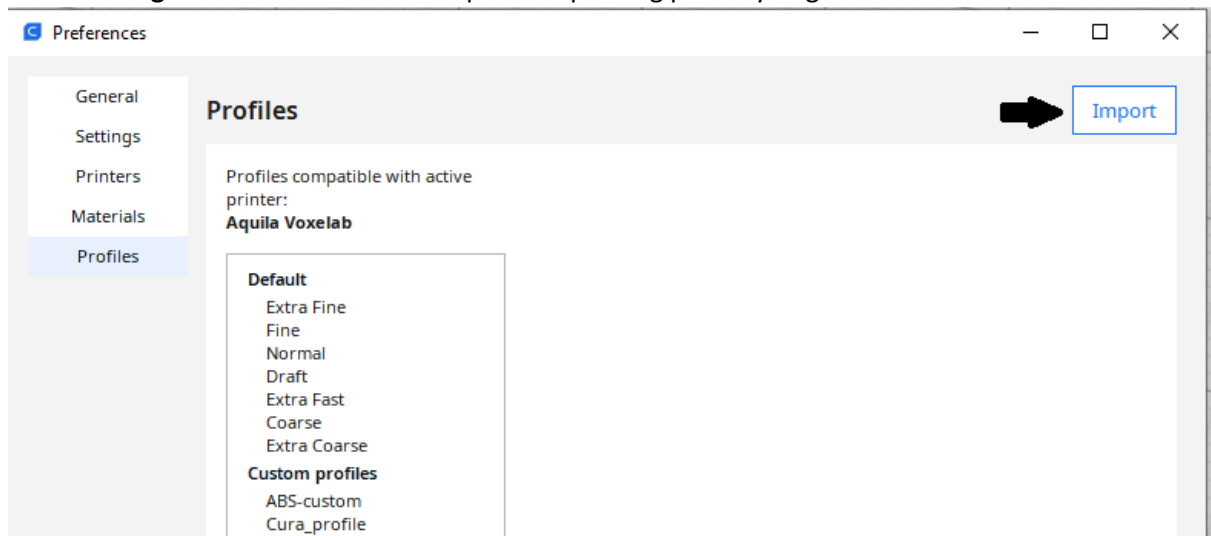
Start G-code:

```
G28 ;Home
G1 Z0.0 F6000 ;Be in wipe distance
G92 E0.0 ; reset extruder distance position
G1 Y60.0 E9.0 F1000.0 ; intro line
G1 Y100.0 E21.5 F1000.0 ; intro line
G92 E0.0 ; reset extruder distance position
G1 Z5.0 ;Go slightly up
```

End G-code:

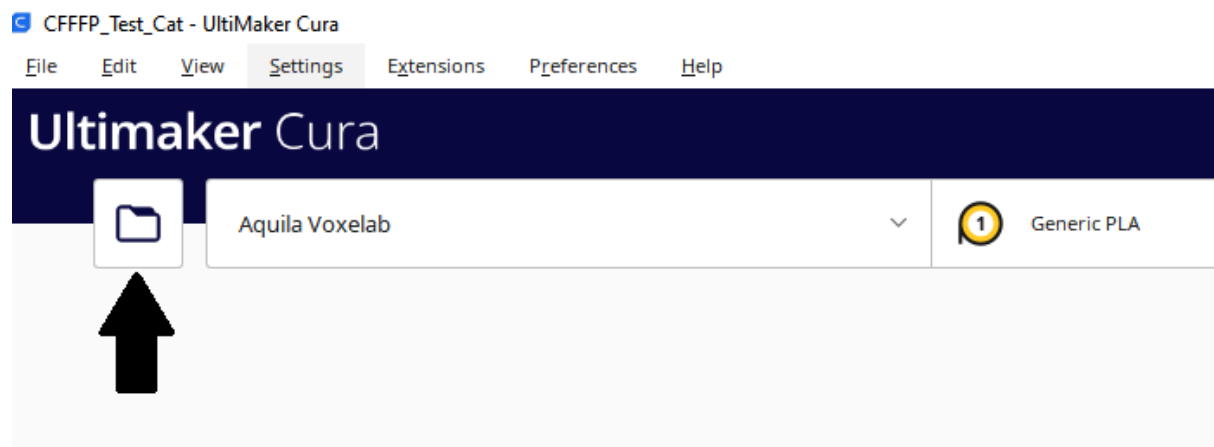
```
M104 S0
M140 S0
G92 E1 ;Retract the filament
G1 E-1 F300
G28 X0 Y0
M84
```

7. Go to **Manage Printer>Profiles** and import the printing profile you got from the SD-card



Getting started

Upload the file by pressing the folder button in the top left corner or simply drag and drop your .stl file in Cura



Change the settings as needed, see *changing settings*

Press the slice button at the bottom right corner, when done with the settings.



Save to the microSD-card and eject it.

Place the microSD-card into the printer in the slot found on the left side on the printer.

Turn on the printer

By using the knob, press down on the print category on the printer.

Find your print by turning the knob and then press down to start the print.

The 3D printer will start printing the build plate adhesion first either brim or skirt.

Important note: This is where you do the micro stepping, which may need to be done!

If you immediately can see that there either is a problem with the print being wrong or the 3D printer not behaving correctly press the knob down again twice and you can stop the print.

If the problem is the 3D printer please see *Troubleshooting*

Changing slicer settings

While there are a lot of settings that you can configure in Cura, please refrain from changing more than the basic settings. Usually, you won't need to change more than Layer height, infill, support and nozzle temperature to get your desired result. With infill density it is important to remember that it barely impacts the strength of the print beyond the 50% point and usually as low as 10% is enough.

The settings should be changed within the range given below:

Layer height: 0.1 – 0.3 mm

Infill Density: 5 – 50 %

Infill Pattern: Any type you want

Support: on – off

Support type: Tree - normal

Temperature: 190 – 210 °C

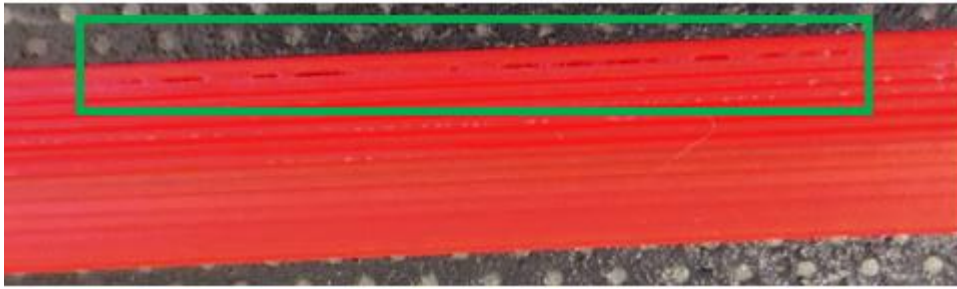
[What type of infill should I use?](#)

[How do I use support?](#)

Micro stepping

Micro stepping is when a correctional Z axis calibration is done after the print has been initialized.

While the bed leveling is continuously adjusted, errors can happen and sometimes the Z-axis needs to be corrected after the print has started. When the first layer is being put down, please look careful that the nozzle is neither too close nor too far away from the bed. An example of the nozzle being too far away can be seen as gaps in the first layer as shown below:



To correct this during the print go to the gear symbol on the printer and press the knob, then go to the bottom and find off-set Z axis and press the knob. You can then adjust the nozzle distance to the bed by either increasing (positive) or decreasing (negative) the distance in increments of 0.01 mm. Follow the [Information post about nozzle height](#) to get the best results.

Object placement

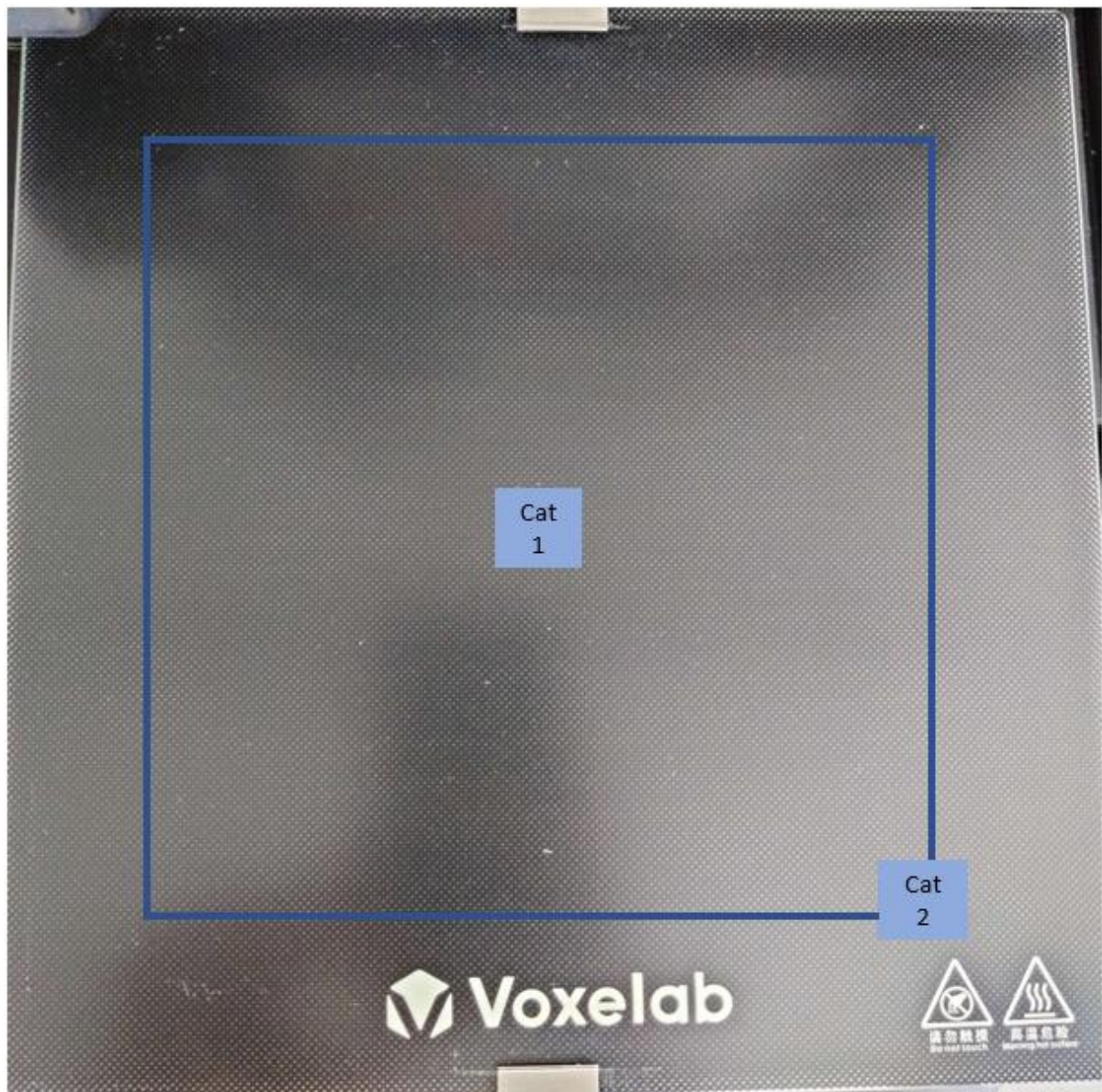
When working in the slicer it is important to understand the limits of the printer bed. At the perimeter of the bed there is worse bed adhesion when compared to the middle. The reason is a combination of the temperature gradient on the bed and the bed being uneven. This problem has actually now been mostly solved with the addition of a mesh. If there still is a problem, try and do micro stepping so that the nozzle is slightly closer to the bed, should still not touch or scratch the bed!!!. Otherwise, try and increase the bed temperature in increments of 3 C°

To show an example of what could happen a test cat has been printed both on the outskirts of the 3D printer and in the middle of the 3D printer, the results can be seen below:



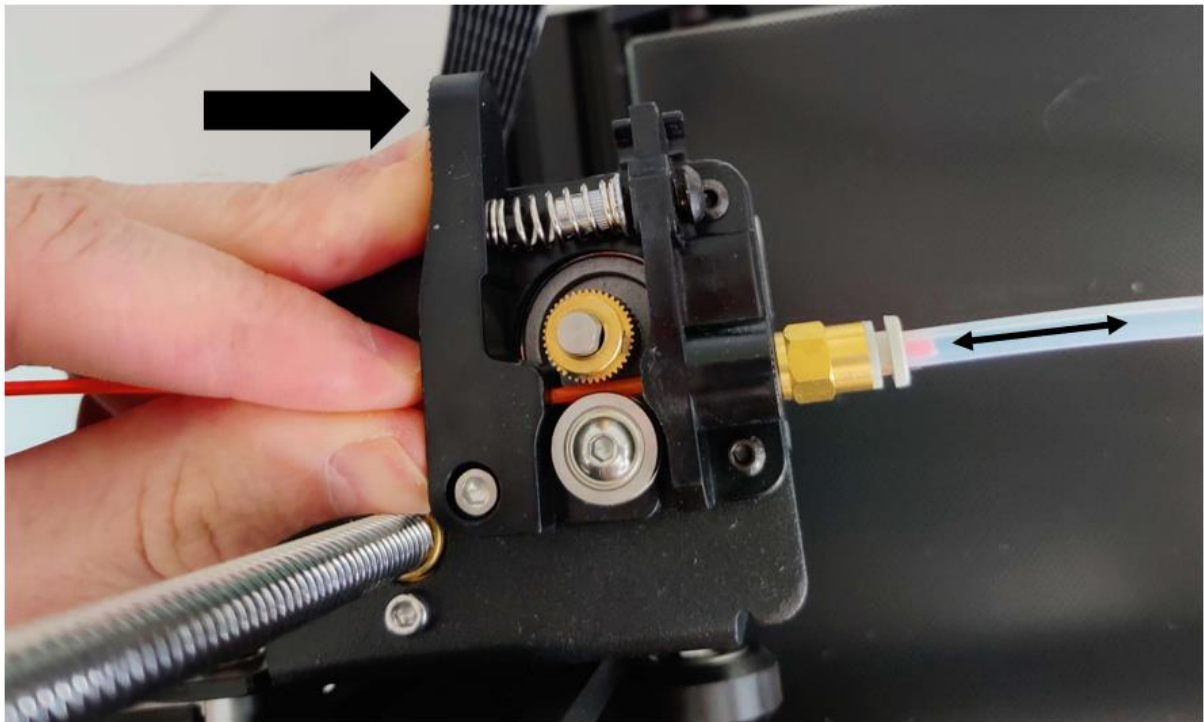
After some time, the bed completely let go of Cat 2 resulting in entanglement of the extruded filament. Therefore, it is very important to understand the bed limitations when either doing a lot of prints or doing a very large print. The placement of both cats can be seen below as well as an

estimation of where the bed adhesion becomes worse, however, the exact limits have not been examined extensively.



Filament change

To change filament first go to the control option on the printer by turning the knob. Then choose the filament unload option and wait until the filament has unloaded from the bowden extruder. Then you can exchange the filament by pulling out the filament and removing in the filament holder.



To load in a new filament on the Bowden extruder hold down the shaft and push the new filament through into the PTFE tubing. If you release the shaft it should hold the filament. You can then choose the filament load option from the settings on the printer. See pictures below for a procedure example.

Github and good 3D printer practices

You can find all the necessary files on the “3D printers Aabogade” github. When using the tools on the 3D printer please put them back and leave it as shown in the picture below:

While you can use the spatula to remove your print, please be very careful as it is easy to scratch the bed surface. Also, after you have removed your print consider quickly wiping the printer bed with a paper and a little bit of ethanol to remove fingerprints as the oil from them worsen the bed adhesion.

The microSD-card should simply be left in the printer and the printer should be turned off after the print is finished. It is a mutual responsibility to keep the 3D printer clean. Keep up to date with the printer profile as it is continuously updated and tweaked to eliminate potential errors and artefacts on the 3D prints.



Troubleshooting

To reflash the software on the Aquila: <https://www.youtube.com/watch?v=sQFsnlyJ5BM>

Print not sticking to the bed:

The reason for the 3D print not sticking could either be that the nozzle is too high above the bed or the bed simply has lost some of its gripping ability. The first thing is to try and clean the bed in isopropyl alcohol or ethanol. If that does not work and you suspect bed leveling might be the problem, please read the following link to get an understanding of how bed leveling impacts a 3D print:

[Information post about nozzle height](#)

To do bed leveling and mesh readjustments, go to the leveling option on the 3D printer and use the given options. The 3D printer will automatically take you through the leveling procedure. To test if the distance is accurate, fold a piece of A4 paper so that it has the double thickness and put it between the nozzle and the bed (do it before the nozzle goes down to the bed). The distance between the nozzle and the bed should be so that the paper is held, though it should not be impossible to remove.

[How to calibrate a 3D printer](#)