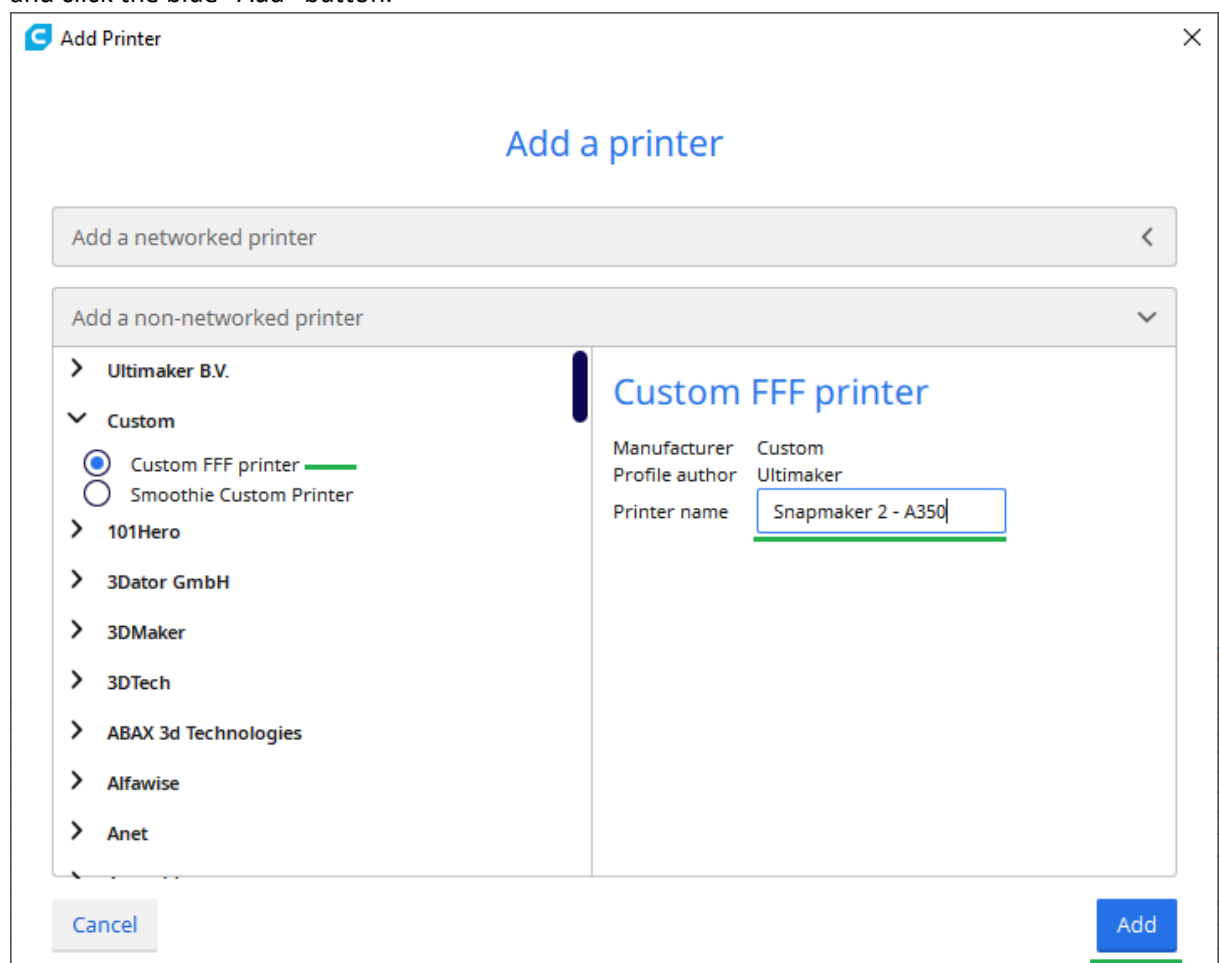


How to setup Cura for Snapmaker 2

Printer specific settings

- Open Cura
- Go to Preferences → Configure Cura...
- Go to the Printers tab
- Click the 'Add' button
- Click the dropdown next to "Add a non-networked printer"
- Scroll to "Custom" and select "Custom FFF printer"
- Give your printer a name (For example "Snapmaker 2 - A350") and click the blue "Add" button.



- In the printer tab add the following dimensions, depending on the model you have. Most settings are shared but the ones marked in orange are different depending on your model. Be sure to set them correctly !!!

○ **A150**

Machine Settings

×

Snapmaker 2 - A150

Printer

Printer Settings

X (Width)

160mm

Y (Depth)

160mm

Z (Height)

145mm

Build plate shape

Rectangular

Origin at center☐

Heated bed☒

Heated build volume☐

G-code flavor

Marlin

Start G-code

CHECK
BELOW

Extruder 1

Printhead Settings

X min

-20mm

Y min

-10mm

X max

10mm

Y max

10mm

Gantry Height

145mm

Number of Extruders

1

End G-code

CHECK
BELOW

Close

○ **A250**

Machine Settings

×

Snapmaker 2 - A250

Printer

Extruder 1

Printer Settings

X (Width)

230mm

Y (Depth)

250mm

Z (Height)

235mm

Build plate shape

Rectangular

Origin at center☐

Heated bed☒

Heated build volume☐

G-code flavor

Marlin

Start G-code

CHECK
BELOW

Printhead Settings

X min

-20mm

Y min

-10mm

X max

10mm

Y max

10mm

Gantry Height

235mm

Number of Extruders

1

End G-code

CHECK
BELOW

Close

○ **A350**

Machine Settings

Snapmaker 2 - A350

Printer

Printer Settings

X (Width) 320 mm

Y (Depth) 350 mm

Z (Height) 350 mm

Build plate shape Rectangular

Origin at center ☐

Heated bed ☒

Heated build volume ☐

G-code flavor Marlin

Printhead Settings

X min -20 mm

Y min -10 mm

X max 10 mm

Y max 10 mm

Gantry Height 330 mm

Number of Extruders 1

Start G-code

End G-code

CHECK BELOW

CHECK BELOW

Close

- In the **“Start G-code”** on the **“Printer”** tab, remove the automatically added code and replace it with the following code:

```
;Start GCode begin
G28 ;home
G90 ;absolute positioning
G1 X-10 Y-10 F3000
G1 Z0 F1800
G92 E0
G1 E20 F200
G92 E0
;Start GCode end
```

- In the **“End G-code”** on the **“Printer”** tab, remove the automatically added code and replace it with the following code:

```
;End GCode begin  
M104 S0 ;extruder heater off  
M140 S0 ;heated bed heater off (if you have it)  
G90 ;absolute positioning  
G92 E0  
G1 E-1 F300 ;retract the filament a bit before lifting the nozzle, to release some of the pressure  
G1 Z330 E-1 F3000 ;move Z up a bit and retract filament even more  
G1 X0 F3000 ;move X to min endstops, so the head is out of the way  
G1 Y350 F3000 ;so the head is out of the way and Plate is moved forward  
;End GCode end
```

- In the **“Extruder 1”** tab add the following settings:

The screenshot shows the 'Machine Settings' window for a Snapmaker 2 - A150. The 'Extruder 1' tab is selected. Under 'Nozzle Settings', the following values are entered: Nozzle size (0.4 mm), Compatible material diameter (1.75 mm), Nozzle offset X (0 mm), Nozzle offset Y (0 mm), and Cooling Fan Number (0). Below these are two empty text boxes for 'Extruder Start G-code' and 'Extruder End G-code', each marked with a large green 'X'. A 'Close' button is at the bottom right.

Setting	Value	Unit
Nozzle size	0.4	mm
Compatible material diameter	1.75	mm
Nozzle offset X	0	mm
Nozzle offset Y	0	mm
Cooling Fan Number	0	

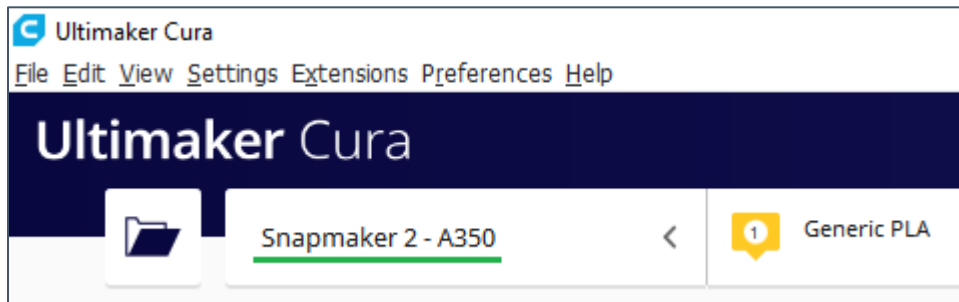
Extruder Start G-code

Extruder End G-code

Close

- Close the Cura preferences screen

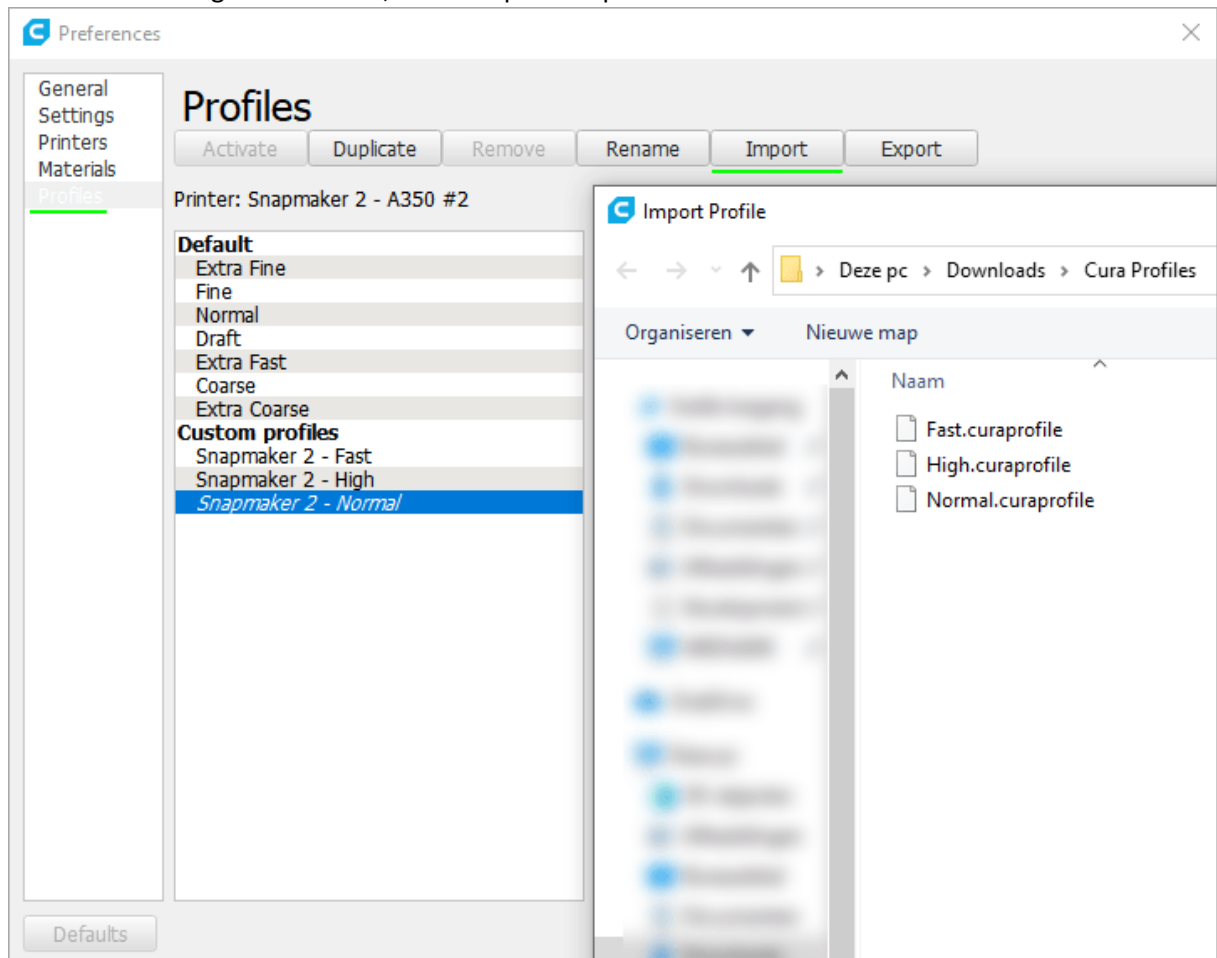
- Make sure you select your newly added printer on top of the Cura screen (if you had other printers setup before)



- Technically Cura is now ready to generate files for your Snapmaker Printer.

Importing print profiles by Snapmaker

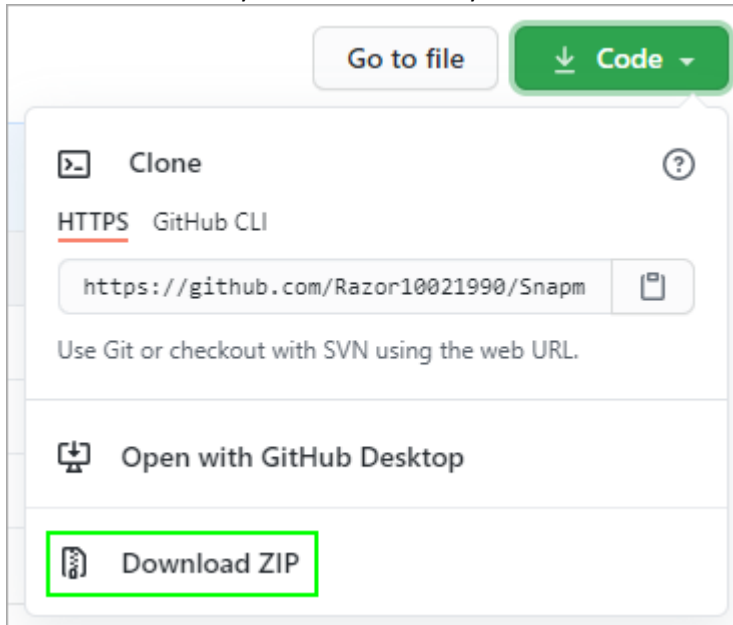
- You can download the 3 premade print profiles from the Snapmaker website here: <https://support.snapmaker.com/hc/en-us/articles/360044341034-What-is-the-recommended-3D-printing-settings-in-Cura-or-Simplify3D-for-Snapmaker-2-0->
- Go to "Preferences" → "Configure Cura..."
- In the "Profiles" tab click the "Import" button and add the 3 profiles one by one. Give them a recognizable name, for example "Snapmaker 2 - Fast".



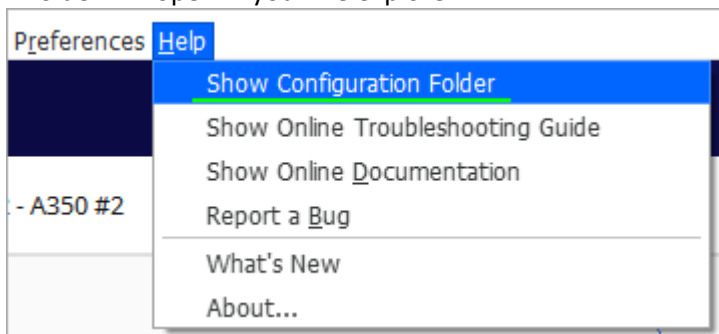
Show model preview and print settings on Snapmaker screen

By default Cura cannot generate the preview image the same way Snapmaker Luban does that. Neither can the Snapmaker machine extract the settings from the gcode to display on the screen, those settings must be added in the header of the gcode for Snapmaker to be able to display them pre-print. Snapmaker user “Razor1990” made a plugin to just that.

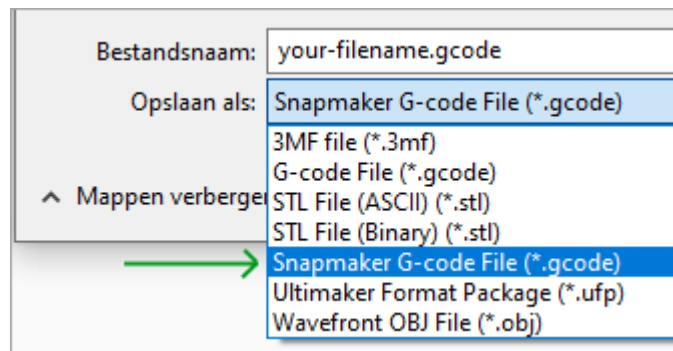
- Go to the plugin page on Github:
https://github.com/Razor10021990/SnapmakerGcodeWriter?fbclid=IwAR3_qGYVvpHAISlcluc12PGeJrr8mnkuOHhGNo9Gjz6xgSq4sNsGUoTXPTM
- Click the green “Code” button and click “Download zip”.
Save it somewhere you can find it easily.



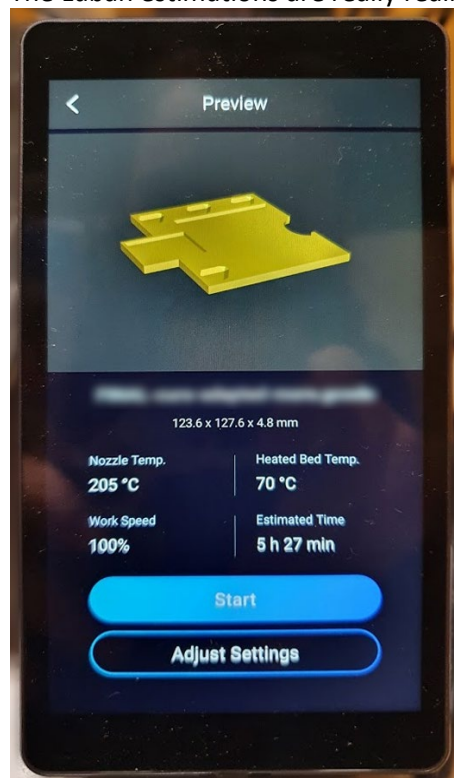
- Open Cura and click “Help” → “Show Configuration Folder”.
A folder will open in your file explorer.



- In that folder find and open the subfolder “plugins”.
This folder is probably empty.
- Past the zip folder you just downloaded into this folder and extract it here.
- Restart Cura.
- Add a model, change print settings as you desire.
- Generate Gcode
- Press the “Save to file” button.
- In the file type options you will see a new option “Snapmaker G-code File”.
Select this type. The standard “G-code File” will not contain the Snapmaker preview etc...



- Place the generated G-code on your USB stick.
 - Place the USB stick in the Snapmaker
 - Select your gcode file
 - You will see on screen a preview of your print, the settings and estimated time.
- Estimated time on Cura should be much closer to reality compared to the estimation by Luban.
The Luban estimations are really really.... * add something not positive here *



Miscellaneous

- This tutorial was based on:
<https://support.snapmaker.com/hc/en-us/articles/360044341034-What-is-the-recommended-3D-printing-settings-in-Cura-or-Simplify3D-for-Snapmaker-2-0->
But this had some mistakes
 - X min and Y min values are displayed as positive values, while Cura only allows negative values. This is correctly displayed in above tutorial.
 - Starting G-code and ending G-code were a tiny bit different compared to what Luban ads to the gcode. Above tutorial uses the same value's Luban ads, as they were in November 2020.
- Identical to what Snapmaker displays on their website:
If there is any defect or damage caused by the use of third-party materials, accessories or software, the warranty will be voided. See the details of the warranty policy at
<https://www.snapmaker.com/repair>

Me, nor Snapmaker, nor the creator(s) of the SnapmakerGcodeWriter Cura plugin can be held responsible if you brick your machine by using Cura. Even if you follow above tutorial exactly. The fact that it works for me does not guaranty that it will work for you. (But it should 😊)
- This tutorial was last updated 2020/11/21
and checked by:
 - ... (nobody yet)

GOOD LUCK
HAPPY 3D PRINTING