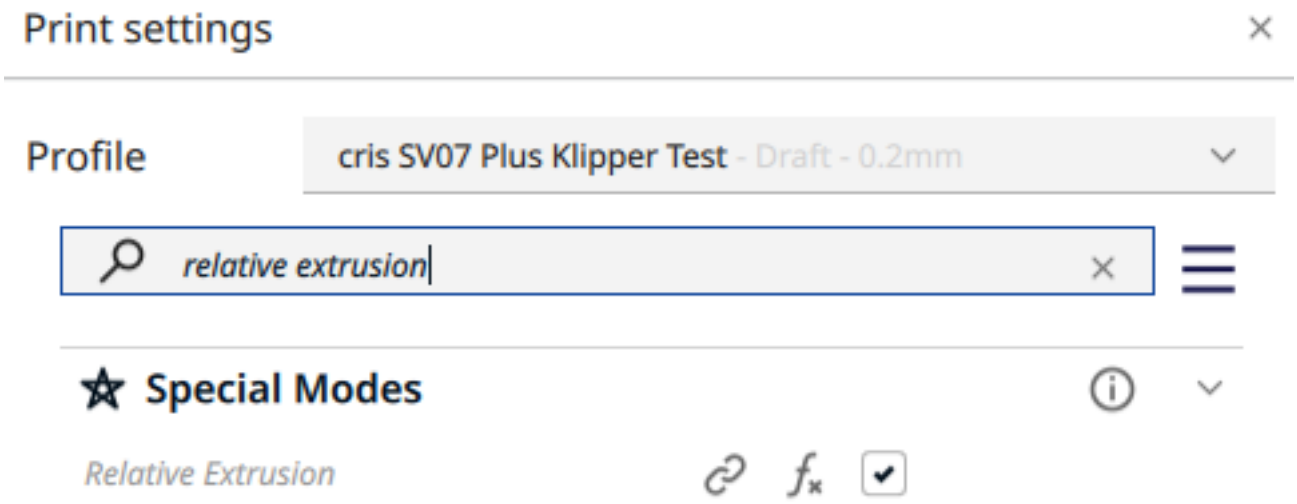
How to set up Cura for Klipper, and the Klipper Printer Additions

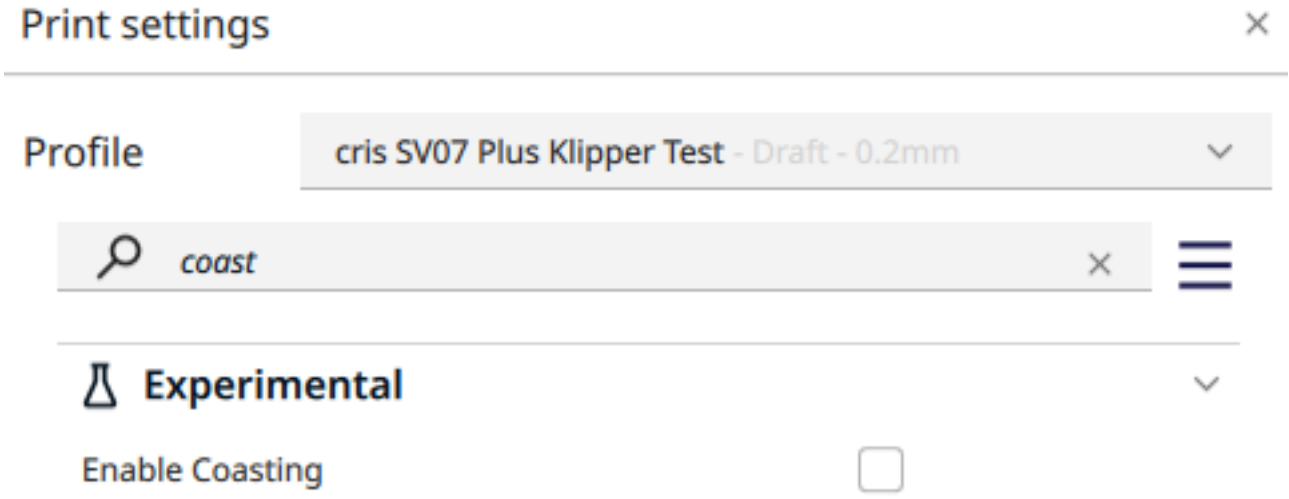
Good day! I have created this guide with **Cura 5.6**. I cannot say if it's exactly the same with older Cura versions and what minimum-version would be required.

Print Settings

• Search for "**Relative Extrusion**" and **enable** it, otherwise your extruder will spin like crazy.



• Search for "**Enable Coasting**" and **DISable** it.



Start-G-Code

- Open the menu "Preferences" --> "Printers" --> "Machine Settings".

- Set the **G-code flavor** to "Klipper", if available. Otherwise set it to "Marlin".

- The following text block contains **minimized text lines**, so it has no line breaks on copy & paste, which would later lead to error messages.
- **Select** the following **ENTIRE** text block. Do not copy line-per-line! Copy & Paste the **entire** text block into the **Start G-Code**:

```
; This is a text block with minimized lines.
;;; KPA Start
; M190 S{material_bed_temperature_layer_0}
; M109 S{material_print_temperature_layer_0}
_Pre_Print_Procedure BED_TEMP={material_bed_temperature_layer_0} EXTRUDER_TEMP={material_print_temperature_layer_0} AREA_START=%MINX%,%MINY% AREA_END=%MAXX%,%MAXY% MODEL_H=%MAXZ% FILA_TYPE={material_type} NOZ_DIA={machine_nozzle_size}
M140 S{material_bed_temperature_layer_0} ; Let a Power failure Resume feature pickup the temperature.
M104 S{material_print_temperature_layer_0} ; Let a Power failure Resume feature pickup the temperature.
; KPA End
```

End-G-Code

- Copy & Paste the following text block into the **End G-code**:

```
; KPA Start
_End_Print
; KPA End
```

So it looks like this:

Start G-code

```
; KPA Start
; M190 S{material_bed_temperature_layer_0}
; M109 S{material_print_temperature_layer_0}
_Pre_Print_Procedure BED_TEMP={material_bed_temperature_l
M140 S{material_bed_temperature_layer_0} ; Let a Power fa
M104 S{material_print_temperature_layer_0} ; Let a Power
; KPA End
```

End G-code

```
; KPA Start
_End_Print
; KPA End
```

If you want to use any of these KPA-Features:

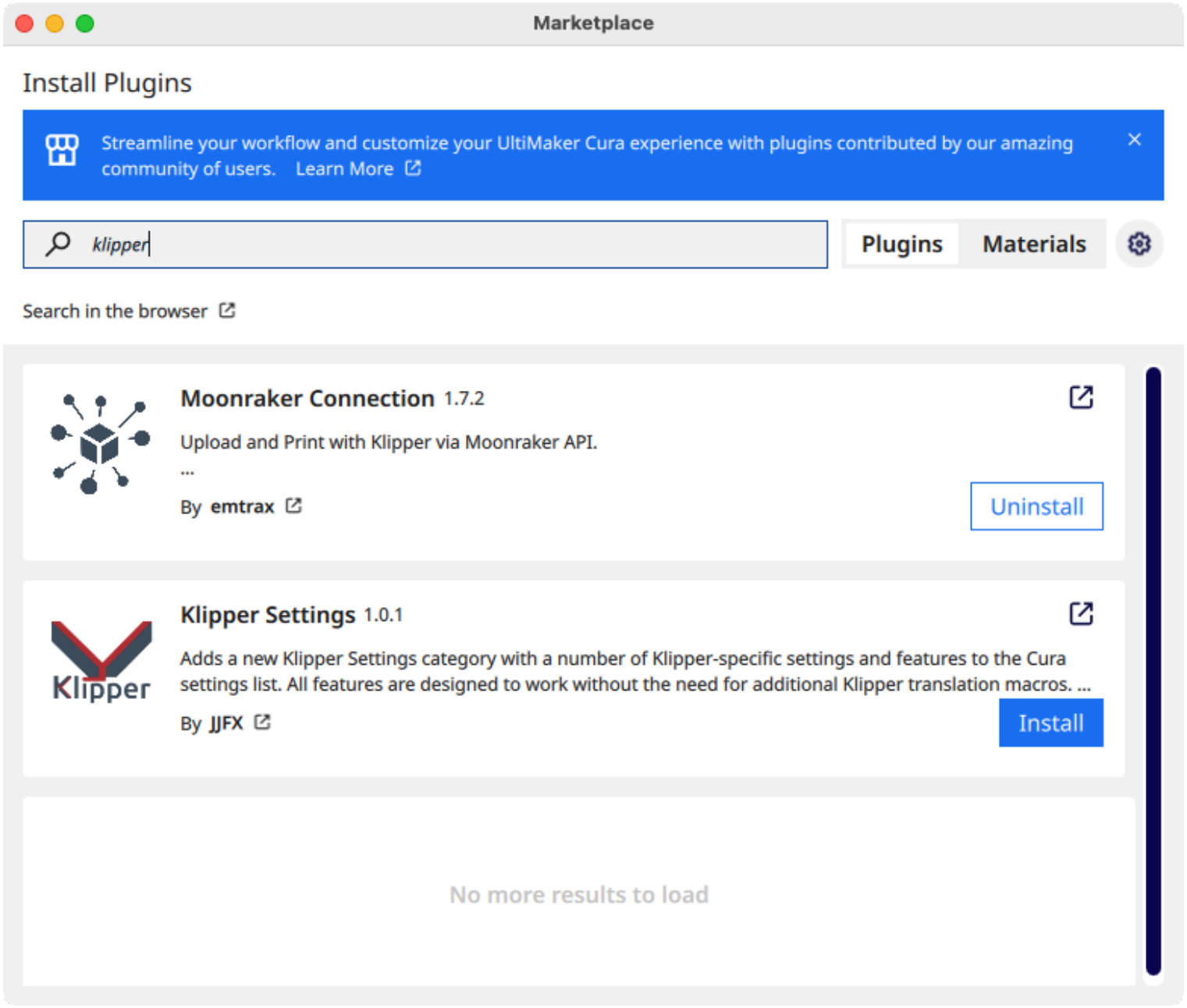
- **Adaptive Mesh**
- **Print Slow In Height**
- **Print Pause In Heights**

Cura, as of version 5.6, can not create a macro-call with it's own model-coordinates, so it requires a script which transfers these values to the actual macro-call in the gcode-output. Here is how:

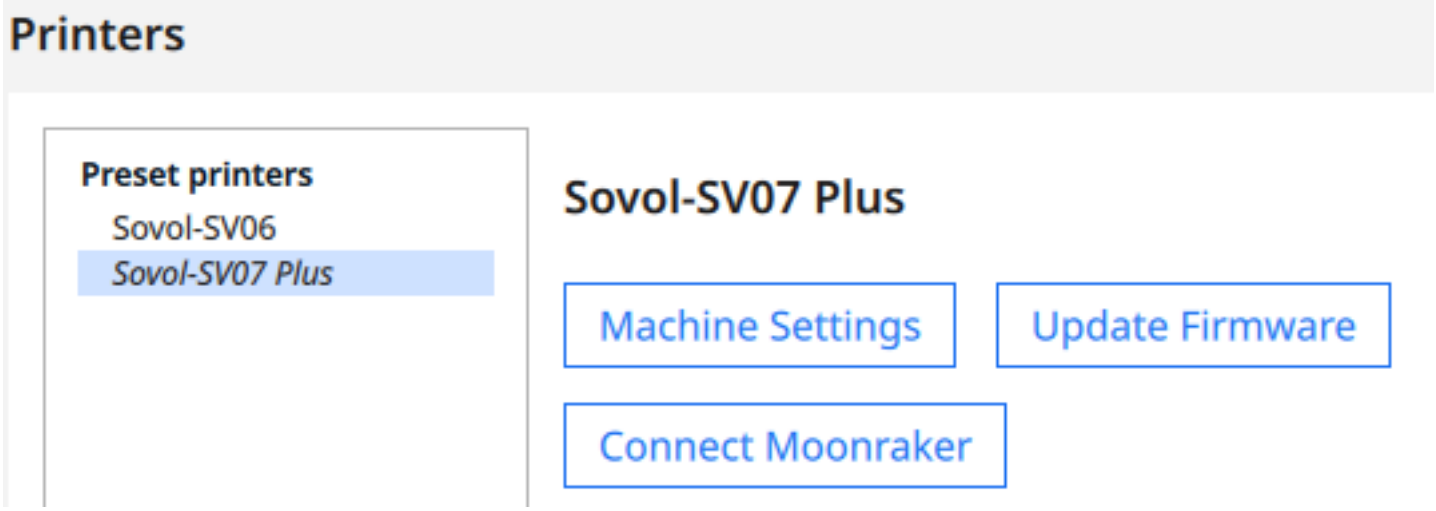
- In Cura, open the menu "Help" and select "Show Configuration Folder".
- Open the folder "scripts" , which you find in the download of the Klipper Printer Additions.
- Copy the file "KlipperPrintAera.py" to the folder "scripts" of Cura's configuration folder.
  - Should you have the **script already installed** from another source, **remove it**, and replace it with the version from the Klipper Printer Additions download.
- Quit Cura. Launch Cura.
- In Cura, open the menu "Extensions" --> "Post Processing" --> "Modify G-Code".
- Click "Add a script". Chose then "Klipper print area mesh".

Optional

- Click "Marketplace", then search for "Klipper".
- You may want to use the Plugin "Moonraker Connection", to upload your gcode-files directly to the printer via network. And the Klipper Settings Plugin:

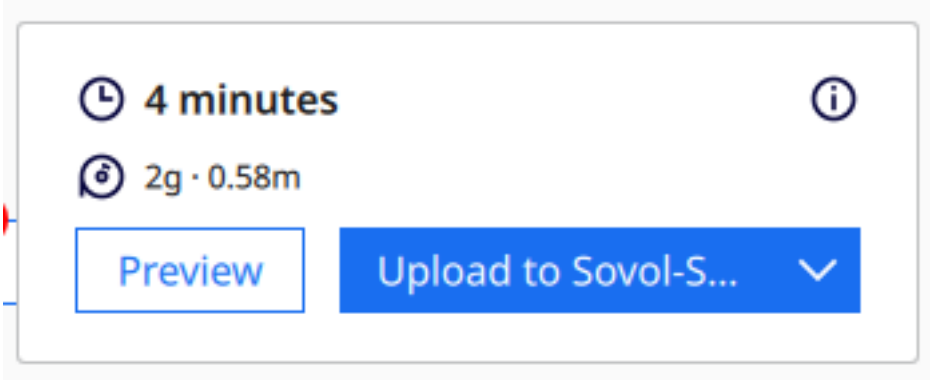


- In the Moonraker Settings, under "Connect Moonraker"...



...you just enter the URL of your printer (the IP address alone does not work):

• In the slice-window you will then get an arrow, which let's you chose whether to save your sliced files to disk, or to directly upload to the printer:



Support

If my work supports you, please support me too:
• PayPal: https://www.paypal.com/paypalme/Morgennebel/

When you buy something, please use my Referral-Link, at no additional cost:
• AliExpress: https://s.click.aliexpress.com/e/_De0Vu1h
• Amazon: https://amzn.to/3L35OX2

Thank you!
Christian 🙂