

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.

Print settings ×

Profile cris SV07 Plus Klipper Test - Draft - 0.2mm ▼

× ≡

★ Special Modes i ▼

Relative Extrusion 🔗 f_x ☒

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

Print settings ×

Profile cris SV07 Plus Klipper Test - Draft - 0.2mm ▼

× ≡

🔧 Experimental ▼

Enable Coasting ☐

Start-G-Code

- Open the menu "Preferences" --> "Printers" --> "Machine Settings".
- Set the **G-code flavor** to "[Klipper](#)", if available. Otherwise set it to "[Marlin](#)".

G-code flavor

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}  
; 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_1  
M140 S{material_bed_temperature_layer_0} ; Let a Power f  
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:

Marketplace

Install Plugins

 Streamline your workflow and customize your Ultimaker Cura experience with plugins contributed by our amazing community of users. [Learn More](#)

 klipper

PluginsMaterials

Search in the browser



Moonraker Connection 1.7.2

Upload and Print with Klipper via Moonraker API.
...

By emtrax

Uninstall



Klipper Settings 1.0.1

Adds a new Klipper Settings category with a number of Klipper-specific settings and features to the Cura settings list. All features are designed to work without the need for additional Klipper translation macros. ...

By JJFX

Install

No more results to load

- In the Moonraker Settings, under "[Connect Moonraker](#)"...

Printers

Preset printers

Sovol-SV06

Sovol-SV07 Plus

Sovol-SV07 Plus

Machine Settings

Update Firmware

Connect Moonraker

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

Sovol-SV07 Plus

Connection

Address (URL)

<http://192.168.178.57/>

- 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:

🕒 4 minutes



📦 2g · 0.58m

Preview

Upload to Sovol-S...



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 😊