

3D-Printing

To produce gorgeous objects with the 3D printer on the 3rd floor, you need to install a 🧑🏻 [slicer](#) on your desktop or laptop. The recommended slicer is [cura](#).¹⁾

There is no card reader on the printer and no wlan, either. All data is communicated exclusively via cable based network.

Install cura

Linux:

Cura is probably natively available in the repo of your distro.

Alternatively, install the 🧑🏻 [ApplImage](#) of the latest release [from github](#).

Microsoft Windows:

Download the installer of the latest release [from github](#).

Apple MacOS:

Download the installer of the latest release [from github](#).

Cura plugins

It recommended to install some plugins in cura:

- click on 'Marketplace' on the top right of curas main window.
- wait about two minutes for cura to assemble the catalogue of plugins
- select:
 1. 'OctoPrintConnection' - necessary to talk to the printer via network
 2. 'Printjob Naming' - to prefix your job file with your name
 3. plugins to import native 3D model files from your favourite CAD suite:
 - 'Inventor integration'
 - 'Solidworks integration'
 - 'Freecad integration'
 - 'Blender integration'
 4. 'Settings Guide' - provides more verbose pop-up help

Configure the printjob naming plugin to put your name in front of the job name:

- cura → Extensions → Custom_Printjob_Naming → set_name_options
- Prefix: replace the default value with your name
- Enable prefix and postfix: [



]

- Add '_' between jobname parts: [



]

- Sanitise jobname parts: [



]

Configure the default printer in cura

1. go to: Settings → Printer → Add_Printer...
2. click 'Add a non-networked-printer'
3. scroll down to 'Custom'
4. choose 'Custom FFF printer'
5. change the printer name to something more specific. E.g.: 'Leapfrog'
6. click the button 'Add' on the lower right
7. in the dialogue 'Machine Settings', tab 'Printer':
 1. Print Settings (left column)
 - X (width) = **180.0 mm**
 - Y (depth) = **240.0 mm**
 - Z (height) = **160.0 mm**
 - Build plate Shape = Rectangular

- Origin at center: []
- Heated bed: [



]

- Heated build volume: []
- G-code flavor = Marlin
- - Start G-code: (no change, keep the default)

2. Printhead Settings

- X min = -66
- Y min = -52
- X max = 66
- Y max = 90
- Gantry height = **2.0 mm**
- Number of extruders = 1
- Apply Extruder offsets to GCode: [



]

- End G-code: (no change, keep the default)

8. In tab 'Extruder 1' of the 'Machine 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
- G-Code: no changes, keep the default.

To revisit this dialogue later, go to: Settings → Printer → Manage_Printers...

Connect printer

1. make sure, the printer is switched on
2. in cura, go to Settings → Printer → Manage_Printers → Printers
3. click 'Connect OctoPrint'
4. click 'Add'

5. type in ..
 - Instance Name: Leapfrog
 - IP Address or Hostname: 'leapfrog.iqo.uni-hannover.de'
 - Port Number: 443
6. click 'OK'
7. click 'Request...' under 'API Key'
8. go to <https://leapfrog.iqo.uni-hannover.de>
9. login with your IQO credentials
10. allow API access
11. done

Cura profile

1. download:
`cura-settings_leapfrog_11april2022c.curaprofile`
2. Import the profile to cura:
 - settings → printer → manage_printer..
 - choose 'Profiles' in left tab → button 'Import'
 - the imported profile should appear under “Custom profiles”
 - select the imported profile in the list
 - click the button 'activate' to actually use the profile for the next slicing. Alternatively, you can choose the profile from the drop-down menu at the top of the 'Print settings' dialogue.

Archive of cura profiles

[n/a: Access denied]

¹⁾

If you don't like cura, feel free to pioneer an alternative like [Slic3r](#), [PrusaSlicer](#) or [SuperSlicer](#).

From:
<https://elektroniq.iqo.uni-hannover.de/> - **ElektronIQ**

Permanent link:
<https://elektroniq.iqo.uni-hannover.de/doku.php?id=3d-druck:start&rev=1652815270>

Last update: **2022/05/17 19:21**

