

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](#).

- accept the terms and conditions
- do not make an account at ultimaker.com → skip
- add a printer → see the next paragraph

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...

Recommended Cura plugins

It is recommended to install some plugins in Cura:

- click on 'Marketplace' on the top right of Cura's 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. In cura v4.18 or lower: there are plugins to import native 3D model files from your favourite CAD suite²⁾:
 - '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: [



]

Connect to the printer

To connect your local cura instance to the leapfrog printer:

1. make sure, the printer is switched on
2. ~~in a web browser go to <https://leapfrog.iqo.uni-hannover.de> and login with your IQO credentials~~
 - Login with IQO credentials is currently not available. Use username 'iqo' with password 'leapfrog42' instead.
3. in cura, go to Settings → Printer → Manage_Printers → Printers
4. click 'Connect OctoPrint'
5. click 'Add'. (Do **not** use one of the existing entries, even though they may announce themselves as "leapfrog".)
6. type in ..
 - Instance Name: Leapfrog
 - IP Address or Hostname: '**leapfrog.iqo.uni-hannover.de**'

- Port Number: **443**
- 7. click 'OK'
- 8. click 'Request...' under 'API Key'
 - API key request are currently not available. Copy `F4223B8E2C1B486B9445ABB58D1431E0` to the edit field instead.
- 9. ~~in the web browser a new tab should have opened which asks you to allow API access → confirm~~
- 10. ~~in cura, the API key should manifest itself as a string of obscured characters~~
- 11. click the 'connect' button on the bottom right of the dialogue – The label of the button should change to 'disconnect'. Congratulations, your instance of cura is now connected to the 3D printer!

The API key is used by cura to authorize itself to the octoprint instance of the leapfrog printer. It lets you print directly from cura without the need to login on the leapfrog website.

Cura profile

1. download:

`leapfrog_21september2022.curaprofile`
2. Import the profile to cura:
 - go to *Preferences* → *Cura_settings...*
 - choose *Profiles* in left tab → button *Import*
 - navigate to the downloaded file and click *open*
 - 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.

Now you are ready to print with the parameters delivered by the cura profile.

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](#).

²⁾

From cura v5.0 onward the various CAD integration plugins got retracted by its author. This may or may not be connected to the [merger of Ultimaker with the stratasy owned Makerbot](#).

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

Permanent link:
<https://elektroniq.iqo.uni-hannover.de/doku.php?id=3d-druck:start&rev=1674253784>

Last update: **2023/01/20 22:29**

