

# 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](#).<sup>1)</sup>

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 suite<sup>2)</sup>:
    - 'Inventor integration'
    - 'Solidworks integration'
    - 'Freecad integration'
    - 'Blender integration'
  4. 'Settings Guide' - provides more verbose pop-up help
  5. Sidebar GUI - transform the slightly annoying floating settings menu into a regular sidebar

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: [



]

- Sanitize jobname parts: [



]

## Cura profile

Cura maintains 'profiles' which are essentially set of slicing parameters. The application comes with several default profiles suggestively named "extra fine", "fine", "normal", ..., "extra coarse". While the parameters of these profiles are reasonable, it is recommended to add our own blends, which reflect the current state of the leapfrog.

### 1. download print profiles:

- leapfrog\_fein\_20januar2023.curaprofile
  - 20% infill; outer layer height = 0.1 mm; infill layer height = 0.4 mm
- leapfrog\_grob\_20januar2023.curaprofile
  - 20% infill; outer layer height = 0.3 mm; infill layer height = 0.9 mm
- leapfrog\_voll-fein\_20januar2023.curaprofile
  - 99% infill<sup>3)</sup>; outer layer height = 0.1 mm; infill layer height = 0.4 mm
- leapfrog\_voll-grob\_20januar2023.curaprofile
  - 99% infill; outer layer height = 0.3 mm; infill layer height = 0.9 mm

## 2. import the profiles 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.

These profiles will get you started. Feel free to tweak the parameters to suit your needs even better. This is especially true for layer height and speed. With large prints every additional mm/s reduces the time to print significantly. Small models with delicate details benefit from slow nozzle movements.

By default, cura only exposes a severely reduced set of parameter in the GUI. Use the three-horizontal-lines button next to the search field to switch to your preferred level of settings detail.

## 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 requests are currently not available. Copy F4223B8E2C1B486B9445ABB58D1431E0 to the edit field instead. 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.
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!



If the above recipe does not work, you can still use cura to print on the printer:

1. slice your model
2. go to the preview tab
3. press the blue button on the lower right to save the \*.gcode to a file
4. log in to the web interface of octoprint → <https://leapfrog.iqo.uni-hannover.de>
5. click the button “upload” on the bottom left. A file chooser dialog pops up. Select your gcode file. Your file will show up on the top of the list of print jobs.
6. click on the tiny printer icon in the row of buttons associated with your file → the printer should immediately start the print job by heating the bed and the nozzle.

## Archive of cura profiles

[n/a: Access denied]

<sup>1)</sup>

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

<sup>2)</sup>

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](#).

<sup>3)</sup>

At 100% infill cura treats the whole volume as “wall”. 99% infill allows the profile to differentiate between wall and infill

From:

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

Permanent link:

<https://elektroniq.iqo.uni-hannover.de/doku.php?id=3d-druck:start&rev=1674412627>

Last update: **2023/01/22 18:37**

