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 and no wlan on the printer. So files are communicated exclusively via the network.

Install cura

Linux:
>The application is packaged by Debian and Fedora. So chances are that cura is natively available in the repo of your distro.

>Alternatively, install the ⟨⟩ AppImage of the latest release from github.

MS Windows and 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: [YourName]
- Enable prefix and postfix: [

]

- Add '_' between jobname parts: [

]

- Sanitise jobname parts: [

]

Configure the default printer in cura

1. 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…

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=1649727829

Last update: **2022/04/12 01:43**