QUICK START GUIDE TO INSTALL AND USE THE SHAREBOT SLIC3R PROFILES

This guide wants to help every Sharebot user to install and use the Slic3r profiles developed specifically for the our 3D printers. The profiles were the result of the work made by R&D department technicians to include in just a file every settings you need to print with one of our machines.

First step: Slic3r and the bundle

The first step you need to do is to download Slic3r from our download page: by clicking the button "Slic3r" you'll go on the software website: here you can find a lot of different version, both stable and unstable. These ones are the version currently developed and are prone to be changed while the Slic3r community is working on them.

We advice to get the last stable version, leaving the other ones.

The second stage you must do it to download the Slic3r Sharebot profiles. The profiles are included in a .zip folder that you have to decompress to get the profiles. Inside it you'll find the bundle with all the indications you need to have to print with a Sharebot printer.

Second step: Uploading the bundle

Now it's possible to open the software.
Don't use the program guided install mode: we suggest to use the "expert" mode (go on the "preferences" icon to pass to this modality). Close and restart the software. Now you can work using all the Slic3r features.

Click on the top left "File" icon: you'll see a drop down list where you can open the "Load Config Bundle" option. Select the integrated profiles bundle you've downloaded before. Don't close this windows until the software will warn you that the the bundle has been uploaded correctly.
You don't need to close the software to start working with these profiles.

Why

Installing the Slic3r Sharebot profiles is crucial: your 3D printing user experience will be improved and your 3D prints will have a much better quality.
Our developers are always working with aim to offer you the best and professional tools to optimize your workflow.

Setting the parameters

Once you've loaded the Sharebot profile, on the right of the Slic3r interface a drop down list will open so that you can generate the .gcode file in a easy and intuitive way. Following different software menu you can can set up all the printing settings:

1. The first menu, called "Print Setting" allows to set up the printing quality you want to use:

   - "low", if you want a printing process having a 0,3mm layer;
   - "medium", if you want a printing process having a 0,2mm layer;
   - "high", if you want a printing process having a 0,1mm layer
   - "slow for TPU", if you need a TPU specific printing process;

2. The second menu, "Filament" is dedicated to which material you want to use for your print. Select the option you prefer after you've loaded the filament inside your printer.

3. The third menu, called "Printer", allows to select which Sharebot 3D printer model you want to use:

   - Kiwi-3D
   - NG single extruder
   - NG 2 extruder-2 printing, to print with both Sharebot NG extuders
   - NG 2 extruder -1 printing, to print with just one extruder using a Sharebot NG double extruder printer (the second extuder will be placed in the parking position.

Thanks for the collaboration.

Sharebot Support Team