

## 3D Printer Manual

The OEH PhD Studentsrepresentation purchased a FELIX 3.0 three dimensional printer which can be used by every PhD Student, who is enrolled at the university.

3D printing is the process for creating a three-dimensional object from a computer designed file. By an additive process successive layers of plastic (the same material as LEGO) are laid down under computer control.

For PhD students in life science this is the perfect tools to print your objects such as bones and joints or protein structures. The plastic is highly heat and chemical resistant and can also be used to create cell culture dishes or objects where cells can grow on.

If you are interested in getting your ideas 3D printed just design your objects in any CAD software, widely available as freeware online (e.g. [www.freeCADweb.org](http://www.freeCADweb.org)). When your design is finished just send the data to [phd@i-med.ac.at](mailto:phd@i-med.ac.at). It will be printed within a week for the cost of about 20€.

Installing Software for your LapTop:

Install [www.freeCADweb](http://www.freeCADweb.org) or similar CAD drawing programs to design an object

Install Repetier Host

<http://shop.felixprinters.com/downloads/index.php?path=software/>

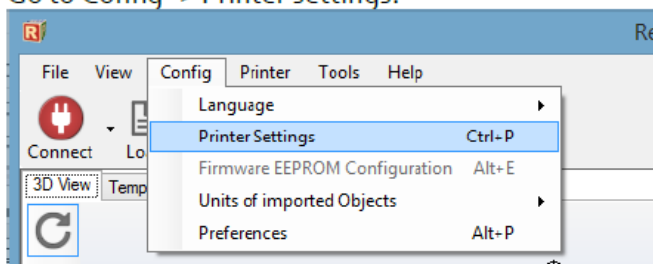
Find Manuals provided by the company

<http://shop.felixprinters.com/downloads/index.php?path=instruction+manuals/>

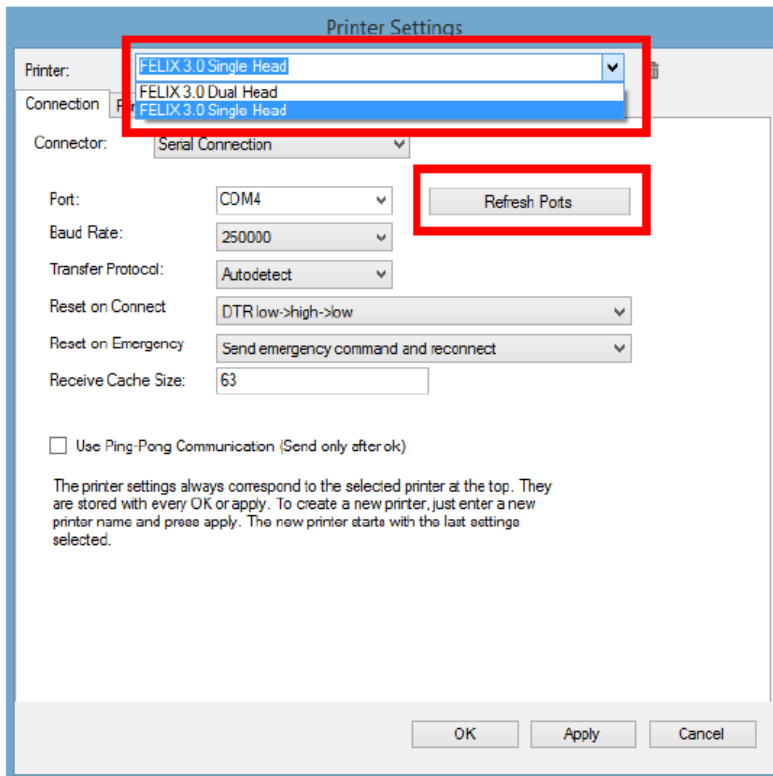
Connect Laptop to Printer by following steps from the manual

[http://shop.felixprinters.com/downloads/instruction%20manuals/20140821%20-%20User%20Manual\\_FELIX\\_3\\_0\\_V10.pdf](http://shop.felixprinters.com/downloads/instruction%20manuals/20140821%20-%20User%20Manual_FELIX_3_0_V10.pdf)

2. Run the **setupRepetierHostFELIXprinters\*.exe** file and follow the installation instructions.
  - During installation it will ask to install a driver. This is recommended to do.
3. Make sure the power cable and USB cable of the printer are connected.
4. Start Repetier-Host.
5. Go to Config -> Printer settings.



6. Select your printer type, single or dual head  
In the configuration window press the **Refresh Ports** button.



Choose the COM-port which belongs to your printer. If no port shows up, it could be because of the following:

- your printer might not be plugged into the USB port.
- The driver is not installed, or currently installing by windows.

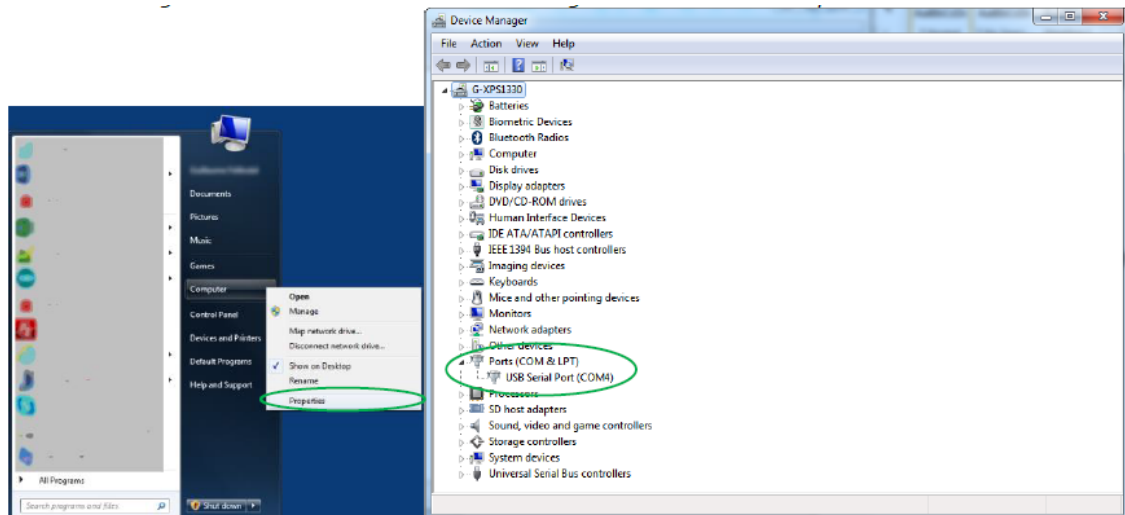
If this is not the case, please install the driver manually according to next chapter.

#### 4.1.2 Driver Installation

1. Plug in the power supply cable.

**i NOTE:** The electronics board is NOT USB powered, so without the power cable the electronics won't work. It needs the voltage of the purple wire to drive the logic of the board. The power to the rest of the board is controlled by the CPU when needed.

2. Plug the USB cable into the computer
3. Your operating system should find the correct drivers automatically, or download them automatically with Windows update.
  - a. If the drivers are not found automatically, then download drivers from here: <http://www.ftdichip.com/Drivers/VCP.htm>.
4. Windows 7: Right-click on "Computer" in the Start Menu, then select "Properties". In the left pane, click "Device Manager". On Windows 8 the Device Manager is accessible directly from the Windows menu.



Note what COM-port is present. If there are more than one COM ports available, unplug the USB cable of the printer and then re-plug it in again. Check what port number is appearing and disappearing. This port number will be used for the next step.

**i** **NOTE:** Sometimes Windows is still not able to find the port. To fix this follow this tutorial:  
<http://forum.arduino.cc/index.php/topic,107098.o.html>

## Prepare CAD File for printing in freeCAD (german version):

Alle einzelformen in ein Projekt zusammenführen: Boolesche Operation

Netze – Erzeuge Netz aus Geometrie – Netz exportieren in STL

## 5 Prepare a CAD file for printing - Slicing

This chapter will prepare a CAD file for printing. It basically converts an STL file to a gcode file. This process is called slicing.

To keep things simple for your first print we recommend you get the test file from here:

[http://shop.FELIXprinters.com/downloads/index.php?path=print\\_files%2Ftest\\_print\\_files/](http://shop.FELIXprinters.com/downloads/index.php?path=print_files%2Ftest_print_files/).

Choose the file. **FELIXprinters\_gadget.STL**.

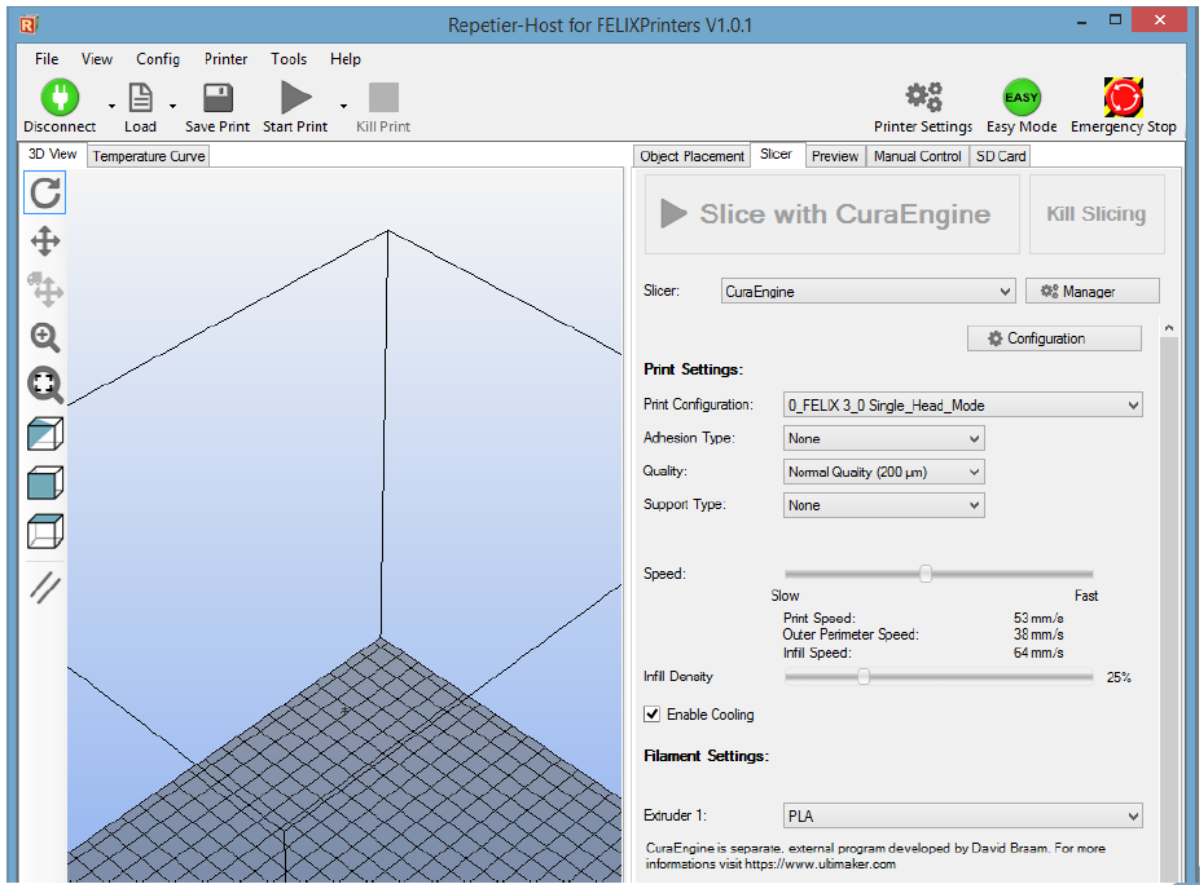
For slicing there are several options available. All have their strengths.

In the Repetier-Host application there are three slicers embedded.

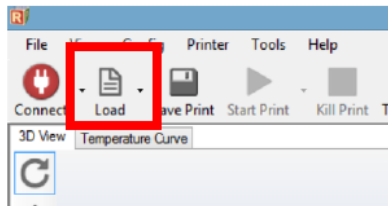
- **CuraEngine (recommended)**
- **Slic3r**
- **SFACT(Used in the previous Repetier-Host versions)**

Next to that we also recommend an external slicing program called Kisslicer. This program generates at the time of writing the best gcode- files for dual extrusion and also very good code for single extrusion.

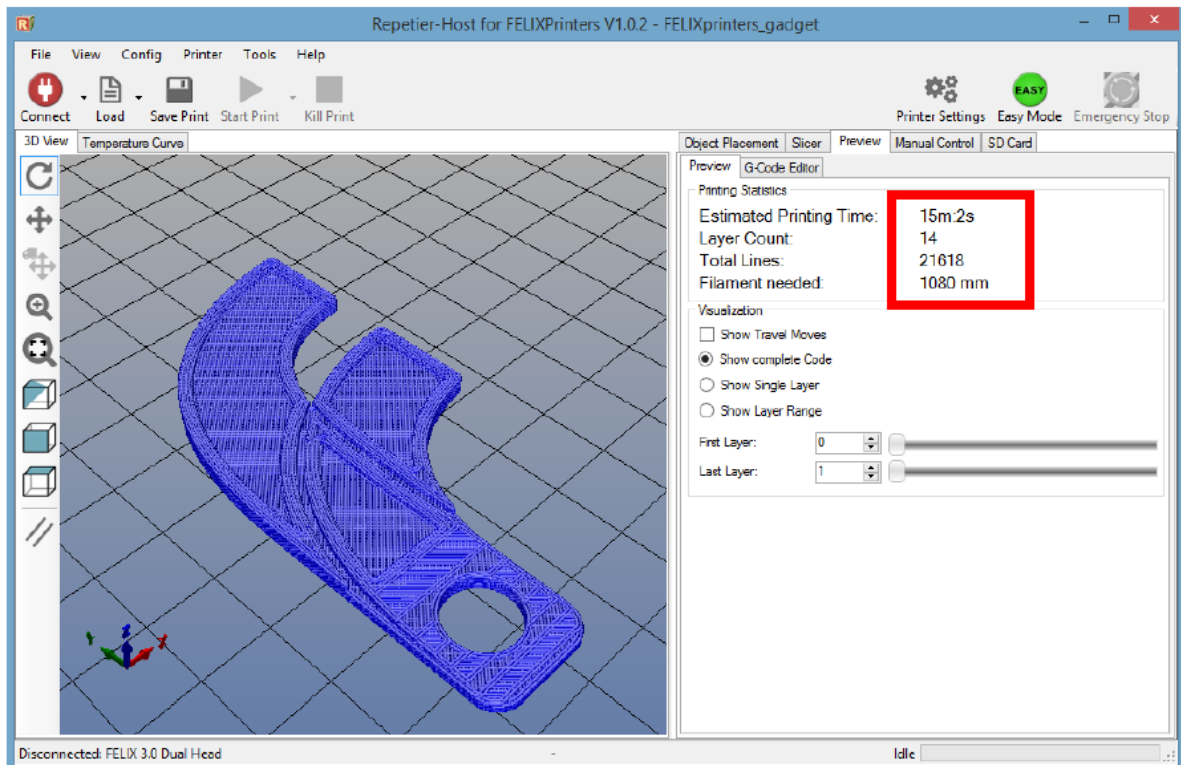
Go to the **Slicer** tab, you'll see the three different slicers. For this demo we'll take the recommended CuraEngine. For your first print, we will take the default settings. Please match the settings in the picture below.



Now load the downloaded STL file:



And press the **Slice with CuraEngine** button.



After slicing you'll see a preview of the generated g-code. Also note the printing time, it will take approx. 15 minutes to finish this object.

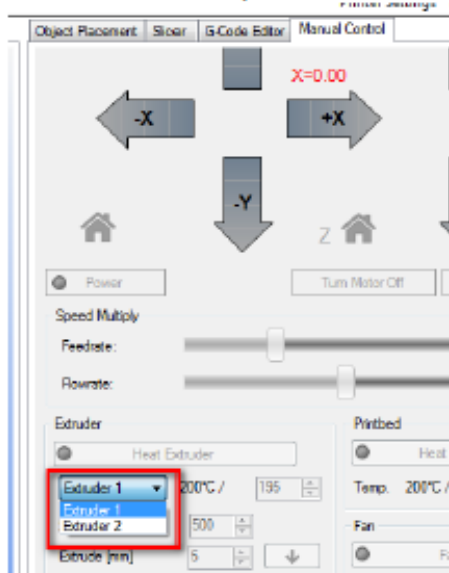
You have now sliced your first object and are ready to go to the next step.

## 6 Your first print

At this stage all we've done all preparations and we're at the final step to convert your idea or 3d file into reality.

In the manual control window do the following.

- In case of dual extrusion, select the extruder of choice.



- Heat up the extruder to 195 degC
- Heat up the heated bed to 55 degC
- Move the table down 10mm

If the extruder and heated bed are warmed up, check or make sure that the following is true:

- ✓ Printer axes can move without obstructions
- ✓ Remove all plastic residues on the heated bed, preferably with the supplied tweezers
- ✓ The heated bed surface is clean and degreased.
- ✓ When all axes are homed, the hot-end is not touching the table.

Make sure the table is moved at least 10mm away from the extruder tip and insert the supplied PLA filament in the extruder.

Run the extruder until a steady flow of plastic comes out.

**i** **NOTE:** When there is not a continuous flow coming out of the nozzle, make sure the extruder arm is properly tensioned. Turn the m4 bolt **COUNTER CLOCKWISE** to put more tension on the filament, so the filament will be pulled into the hot-end better. But watch out, too tight and the motor will have a hard time turning, and in the worst case the filament might be squashed so wide that it gets clogged into the hot-end.

### Recommended settings for printing:

Absolutley no air flow in the room!

Nozzle 203°C

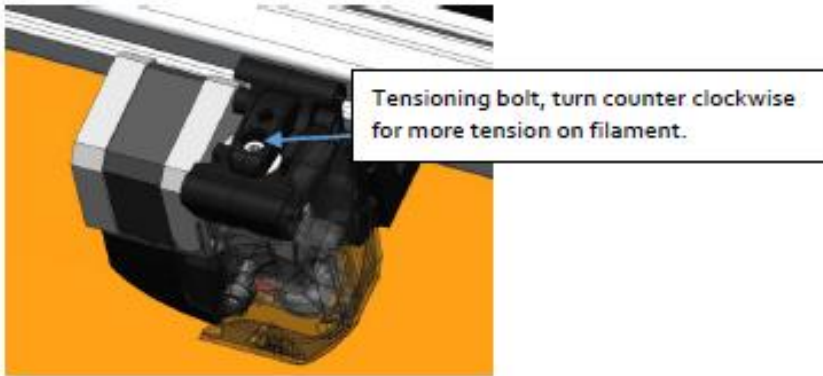
Geschwindigkeit 80 – 90

Fluss 122

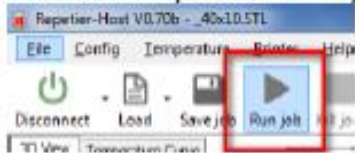
Fan 60 – 100

Bett 45°C

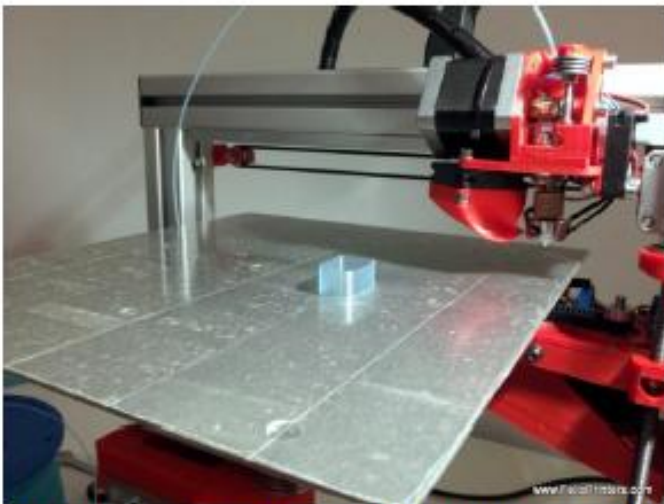




When these steps above are OK, you are ready to press that print button.



If the calibration was done correctly, the print should finish without any problem.



If the print looks like the picture above, then congratulations your printer setup is completed!!

If the print fails, it is usually because the distance between the bed and nozzle is too large. Try to adjust the z-axis limit switch vane to get the distance closer.

To know more about software please continue reading at chapter 7

**i IMPORTANT:** To keep the hot-end running as long as possible, let the filament run through a dust collector like a small piece of sponge. The filament picks quite a lot of dust due to static charge or dirt from the factory. This will all enter the hot-end and partially stick in there and finally clog the nozzle. When this happens the hot-end needs to be cleaned out while hot from both sides. A tutorial for this is available on the forum on the website.

**i NOTE:** If you have a dual extruder, continue to chapter 7

**i NOTE:** For owners who assembled the printer themselves it is strongly recommended to follow assembly checks in appendix 15.