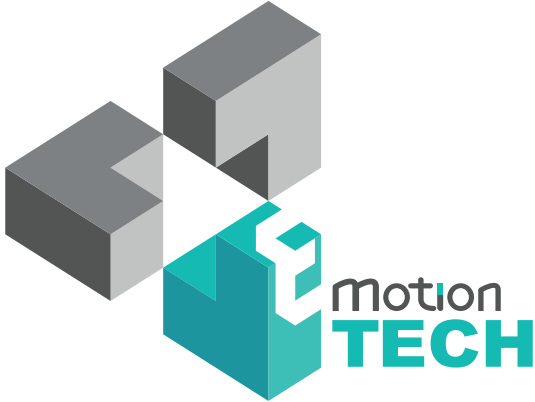
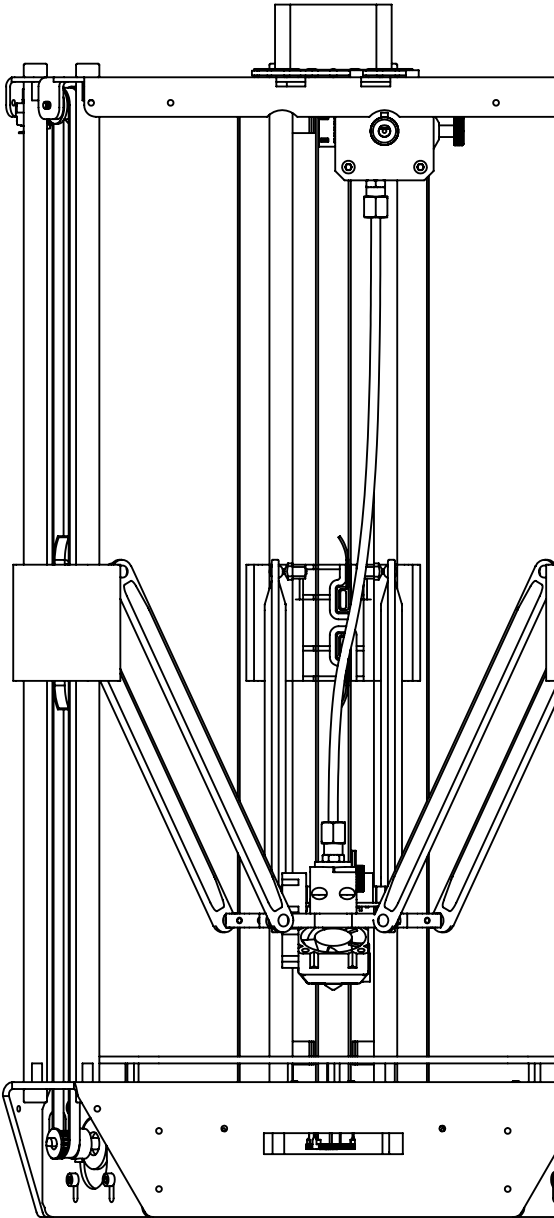




micro
delta
REWORK

USER'S GUIDE





INTRODUCTION

INTRODUCTION

- **Objective:**

Provide a visual guide of the different steps to set-up and get started using the MicroDelta Rework 3D printer.

- **Authors of this document :**

eMotion Tech – <http://www.emotion-tech.com>
Hugo FLYE
Anthony BERNA

- **Photographics credits :**

Pictures and 3D representations made by eMotion Tech :
<http://www.emotion-tech.com>

- **Sources :**

<http://reprap.org/wiki/RepRap>
<http://www.repetier.com/>

- **Licence :**

MicroDelta Rework : CC BY-NC-SA 4.0
This document : CC BY-NC-SA 4.0
<http://creativecommons.org/licenses/by-nc-sa/4.0/>



- **Update:**

Last update : 09/10/2017

- **Links :**

You can find more informations on the following links :

eMotion Tech's website : <http://www.emotion-tech.com>
RepRap community : <http://reprap.org/wiki/reprap>





SUMMARY

INTRODUCTION	2	APPENDIX	27
SUMMARY	4	Mounting of the spool holder	28
SOFTWARE INSTALLATION	5	Spring filament pressure adjustment	30
A. Software installation on Mac OS	6	Printing a 3D model	33
PREPARATION	7	PID calculation	36
A. Files to copy on the TF card	8	To go further	37
B. Connecting your printer to Repetier-Host	9	Maintenance	38
C. Printer's settings	11	Recommandations	38
D. Slicing settings	13	Restore the TF card	39
E. Motion test	15		
F. Heating test	16		
G. Charging filament	17		
H. Extrusion test	18		
I. Calibration process using command lines	19		
PRINTING	22		
A. Spool Holder Print	23		



SOFTWARE INSTALLATION

Software installation on Mac OS

Objective : Install the necessary software to use the MicroDelta Rework on a Mac operating system computer.

1°) Download the latest version of Repetier-Host from the support section of eMotion Tech's website (www.emotion-tech.com).
Support / MicroDelta Rework / Software / Mac /

2°) Find all the necessary informations for the installation on the following link :
<https://www.repetier.com/documentation/repetier-host-mac/installation-and-connection/>

Note: the version of Repetier for Mac OS X is less developed than those on Windows / Linux. Therefore the use of a virtual machine is advised to take advantage of the latest updates and added options. However, we will guide you through this documentation for the installation of the official version of Repetier for Mac OS X.

PREPARATION

Files to copy on the TF card

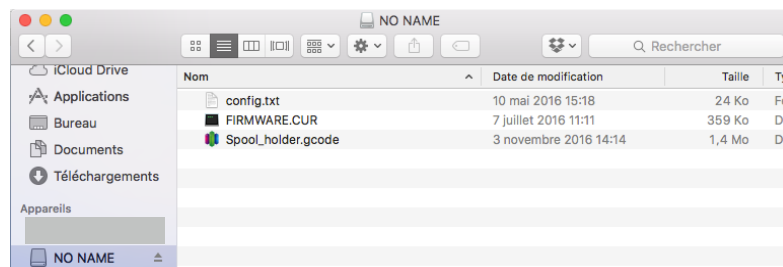
Objective : copy the required files in the eMotronic's TF card.

1°) Go to the eMotion Tech's website support section, then go to «MicroDelta Rework / Software / Configuration» and download the corresponding configuration files to your MicroDelta Rework depending on it's version and options.

2°) Connect your printer to the computer

3°) Once downloaded, unzip the file and copy all its content in the eMotronic's TF card.

Note: TF card is usually automatically detected and its content displayed once you connect your 3D printer to you computer. The TF card generally appears as drive «NO NAME» but this can vary depending on your hardware.



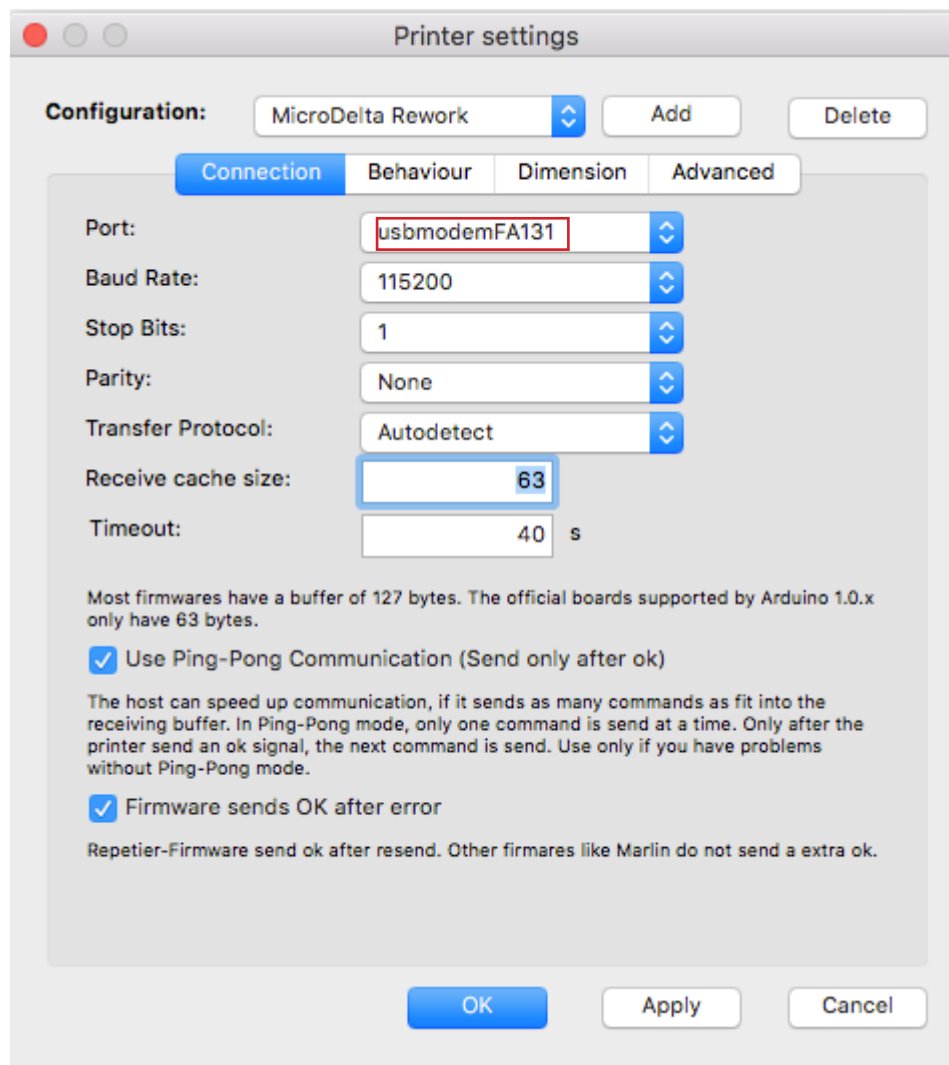
Mac OS X : what you should have on the TF card after this step

4°) Press the «Reset» button on the 3D printer (located on one side of the lower plate).

5°) Check that the device has been properly recognized by your operation system.

Connecting your printer to Repetier-Host

- 1°) Run Repetier-Host software.
- 2°) Go to «Config» then «Printer Settings».
- 3°) Check that all the parameters correspond to those on the image below. **Select the COM port assigned to your MicroDelta Rework.**



The screenshot shows the 'Printer settings' dialog box in Repetier-Host. The 'Configuration' section at the top shows 'MicroDelta Rework' selected in a dropdown menu, with 'Add' and 'Delete' buttons. Below this are four tabs: 'Connection' (selected), 'Behaviour', 'Dimension', and 'Advanced'. The 'Connection' tab contains the following settings:

- Port: **usbmodemFA131** (highlighted with a red box)
- Baud Rate: 115200
- Stop Bits: 1
- Parity: None
- Transfer Protocol: Autodetect
- Receive cache size: 63
- Timeout: 40 s

Below the settings, there is a note: 'Most firmwares have a buffer of 127 bytes. The official boards supported by Arduino 1.0.x only have 63 bytes.'

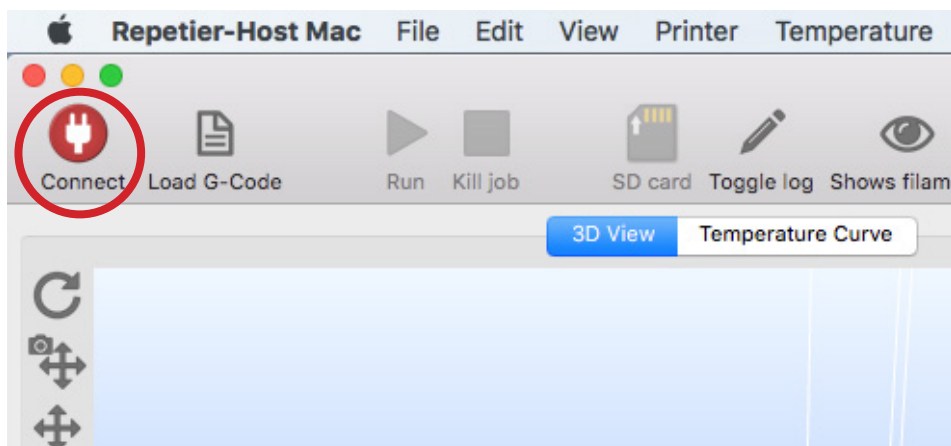
There are two checked checkboxes:

- ☒ Use Ping-Pong Communication (Send only after ok)
- ☒ Firmware sends OK after error

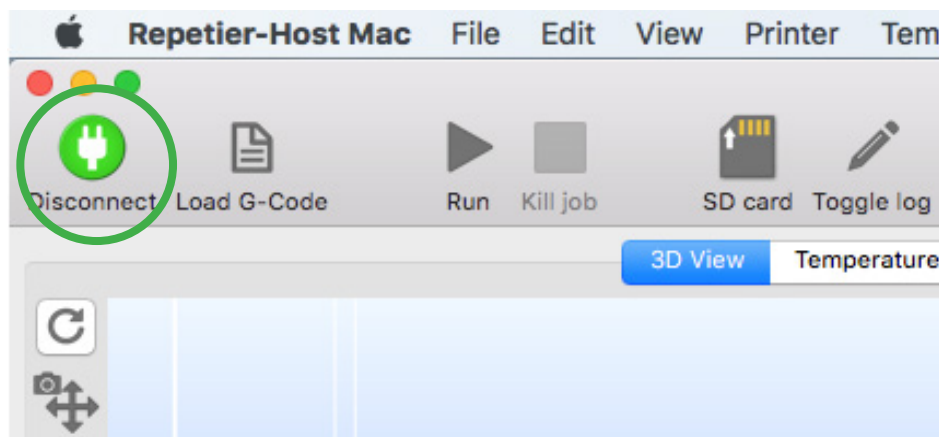
At the bottom, there is a note: 'Repetier-Firmware send ok after resend. Other firmwares like Marlin do not send a extra ok.'

At the bottom of the dialog are three buttons: 'OK', 'Apply', and 'Cancel'.

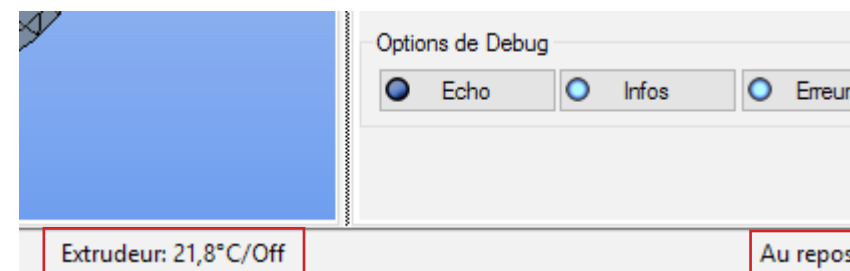
4°) Click on Connect.



This one should change color, becomes green when connected :



5°) Check on the bottom of the screen that the extruder's temperature is consistent (logical value) and the printer's status is «Idle» as shown below:



If you have a Heated Bed, its temperature will be displayed next to the extruder's.

Printer's Settings

A manual set-up of the printer is required. Please follow the following steps:

Configuration:
MicroDelta Rework
Add
Delete

Connection
Behaviour
Dimension
Advanced

Port: usbmodemFA131
Baud Rate: 115200
Stop Bits: 1
Parity: None
Transfer Protocol: Autodetect
Receive cache size: 63
Timeout: 40 s

Most firmwares have a buffer of 127 bytes. The official boards supported by Arduino 1.0.x only have 63 bytes.

☒ Use Ping-Pong Communication (Send only after ok)

The host can speed up communication, if it sends as many commands as fit into the receiving buffer. In Ping-Pong mode, only one command is send at a time. Only after the printer send an ok signal, the next command is send. Use only if you have problems without Ping-Pong mode.

☒ Firmware sends OK after error

Repetier-Firmware send ok after resend. Other firmwares like Marlin do not send a extra ok.

OK
Apply
Cancel

Configuration:
MicroDelta Rework
Add
Delete

Connection
Behaviour
Dimension
Advanced

Travel Feedrate: 10000 [mm/min]
Z Axis Travel Feedrate: 10000 [mm/min]
Default Extruder Temperature: 195 [°C]
Default heated bed temperature: 55 [°C]
Number of Extruder: 1
☒ Check extruder & heated bed every 1 seconds
☒ Don't log temperature requests (M105)
Dump area position: X= 135 Y= 0 Z-Min= 0
☒ Go to dispose after job/job kill
☒ Disable extruder after job/job kill
☒ Disable heated bed after job/job kill
☐ Disable motors after job/job kill
Add to comp. printing time: 0 [%]

OK
Apply
Cancel

Connection

Behaviour

Dimension

Advanced

Home X:

x = 0

Home Y:

y = 0

Home Z:

z max

X Min:

0

[mm]

X Max:

200

[mm]

Y Min:

0

[mm]

Y Max:

200

[mm]

The min and max values define the possible range of extruder coordinates. These coordinates can be negative and outside the print bed. Bed left/front define the coordinates where the printbed itself starts. By changing the min/max values you can even move the origin in the center of the print bed, if supported by firmware.

Printer type:

Delta Printer with Cylindric Print Shape

Diameter:

150

[mm]

Height:

220

[mm]

For delta like printer you still need to set the x/y min/max values to the extreme values possible. If you do not want to leave the printable circle set it to +/-radius.

OK

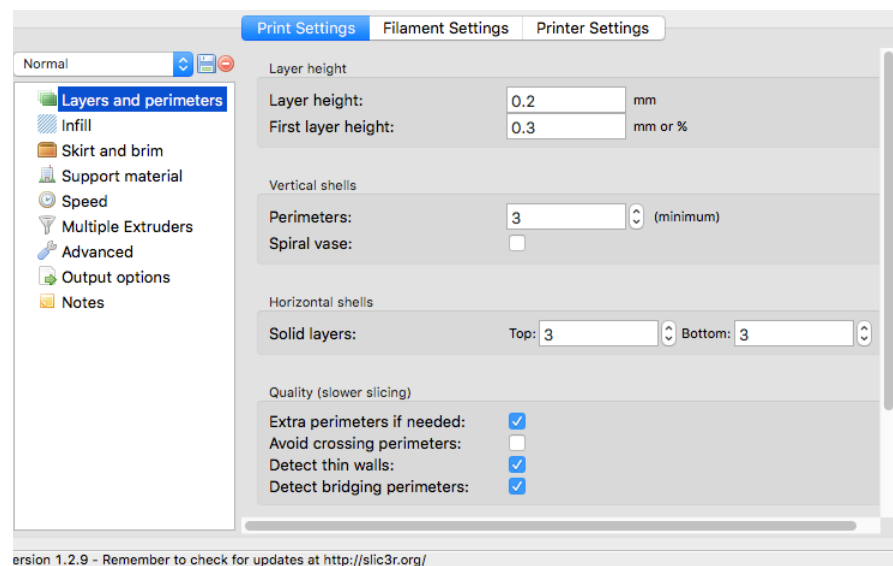
Apply

Cancel

Slicing Settings

Go to the slicer tab, and click on the «Configuration» button.

Make sure to modify you Slic3r configuration values in respect with the following figures.



Print Settings | Filament Settings | Printer Settings

Normal

- Layers and perimeters
- Infill
- Skirt and brim
- Support material
- Speed
- Multiple Extruders
- Advanced
- Output options
- Notes

Layer height

Layer height: 0.2 mm

First layer height: 0.3 mm or %

Vertical shells

Perimeters: 3 (minimum)

Spiral vase: ☐

Horizontal shells

Solid layers: Top: 3 Bottom: 3

Quality (slower slicing)

Extra perimeters if needed: ☒

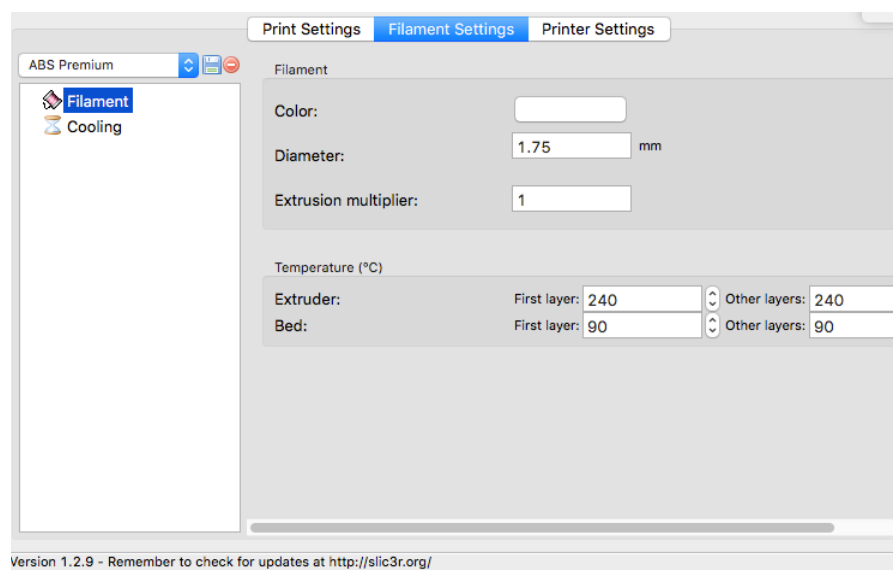
Avoid crossing perimeters: ☐

Detect thin walls: ☒

Detect bridging perimeters: ☒

Version 1.2.9 - Remember to check for updates at <http://slic3r.org/>

Save



Print Settings | Filament Settings | Printer Settings

ABS Premium

- Filament
- Cooling

Filament

Color:

Diameter: 1.75 mm

Extrusion multiplier: 1

Temperature (°C)

Extruder: First layer: 240 Other layers: 240

Bed: First layer: 90 Other layers: 90

Version 1.2.9 - Remember to check for updates at <http://slic3r.org/>

Print Settings

Filament Settings

Printer Settings

PLA Premium

Filament

Cooling

Filament

Color:

Diameter:

1.75

mm

Extrusion multiplier:

1

Temperature (°C)

Extruder:

First layer: 195

Other layers: 195

Bed:

First layer: 55

Other layers: 55

Version 1.2.9 - Remember to check for updates at <http://slic3r.org/>

Save

Print Settings

Filament Settings

Printer Settings

Hexagon hotend

General

Custom G-code

Extruder 1

Size

Nozzle diameter:

0.4

mm

Position (for multi-extruder printers)

Extruder offset:

x: 0

y: 0

mm

Retraction

Length:

2.5

mm (zero to disable)

Lift Z:

0

mm

Speed:

60

mm/s

Extra length on restart:

0

mm

Minimum travel after retraction:

2

mm

Retract on layer change:

☐

Wipe while retracting:

☐

Retraction when tool is disabled (advanced settings for multi-extruder setups)

Length:

10

mm (zero to disable)

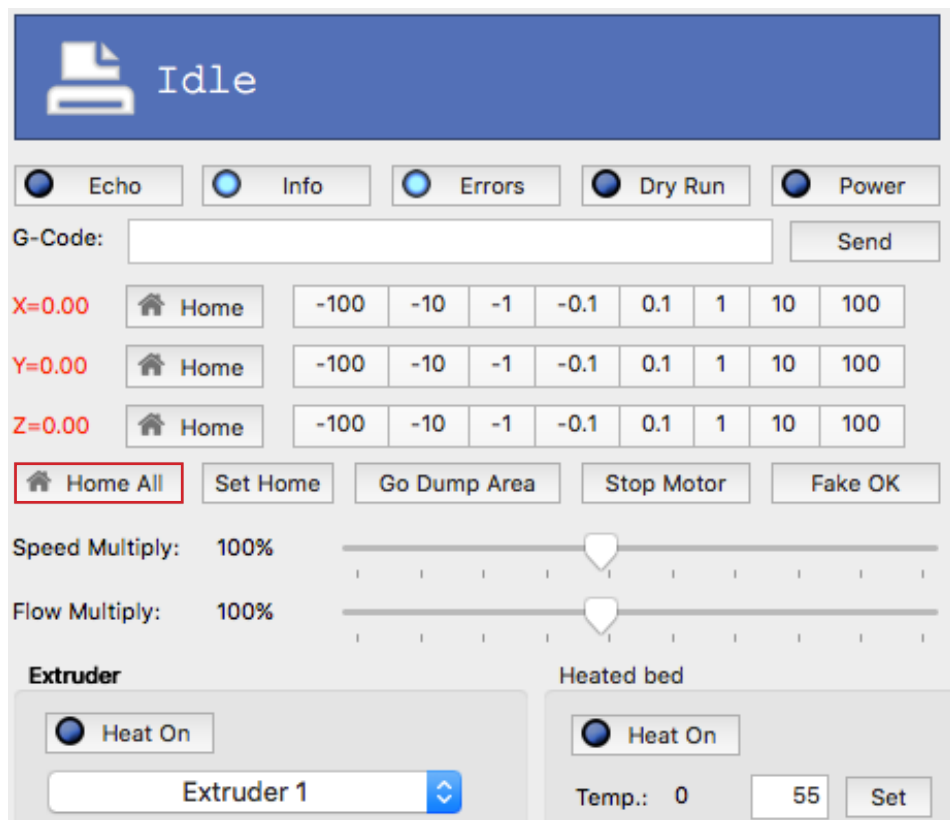
Version 1.2.9 - Remember to check for updates at <http://slic3r.org/>

Save

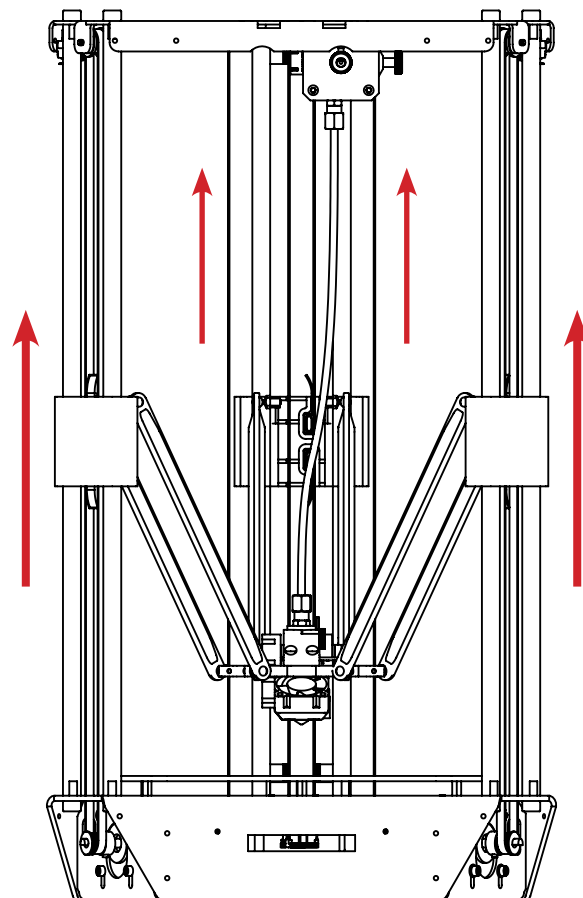
Motion test

1°) Go to the «Print Panel» tab and click on the «Home All» (This should initialize your printer = move all axes to origin).

Keep a finger on the «Reset» button in case of a malfunction.

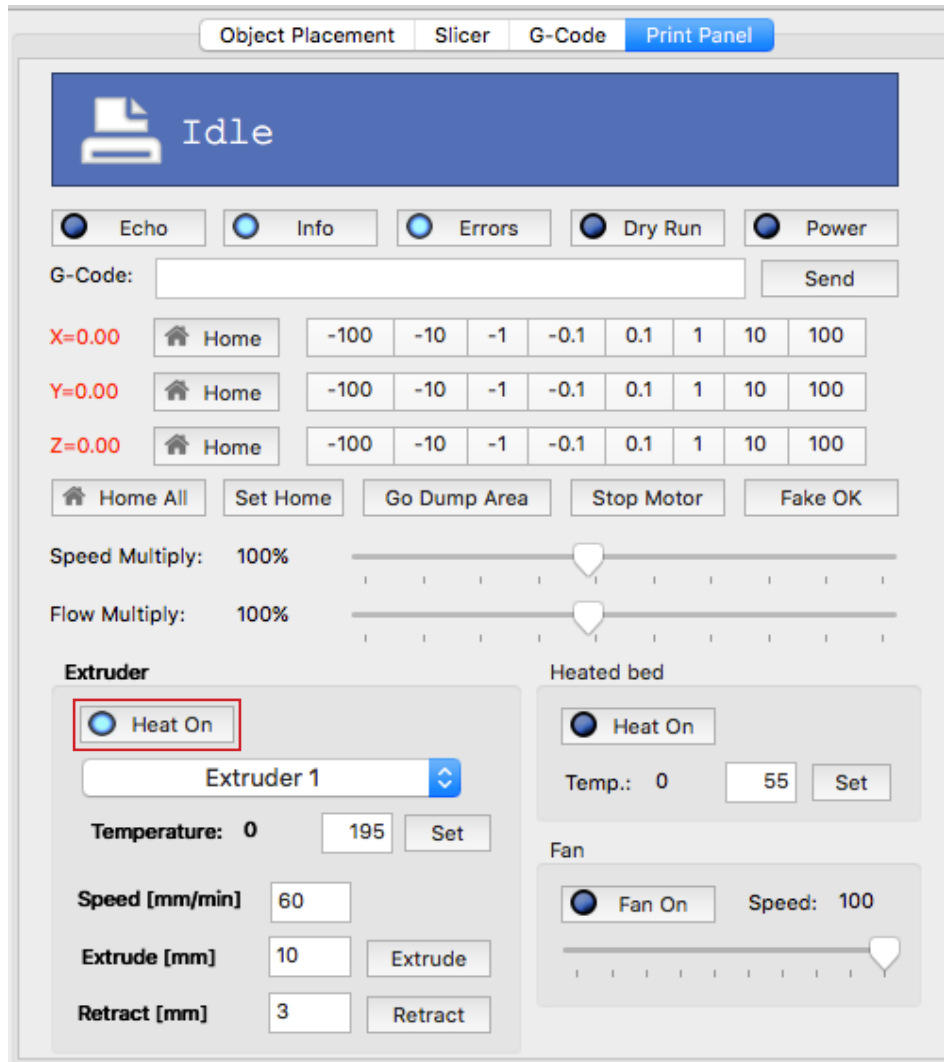


2°) Make sure that EVERY slider moves up till hitting its corresponding endstop.

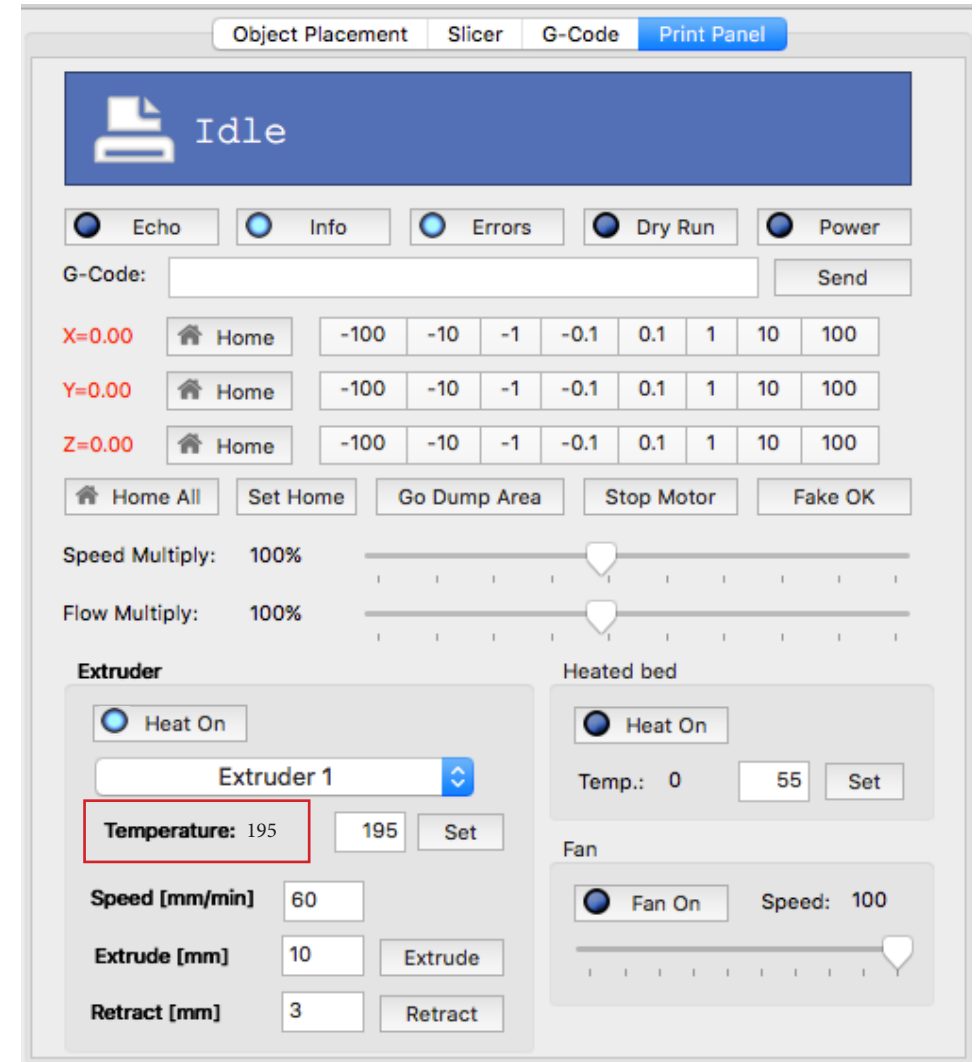


Heating Test

1°) Go to «Print Panel» tab, click on «Heat on» (Extruder heater activation) as shown below:



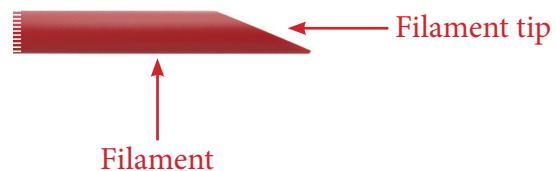
2°) Check that the target temperature is reached :



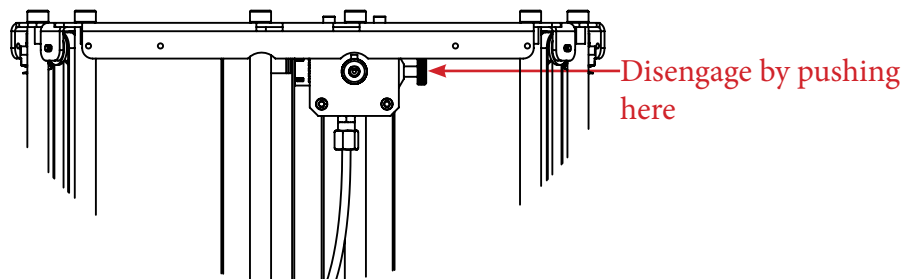
If the target temperature is not reached or the actual temperature varies greatly around the target temperature, perform a PID calculation (see appendix).

Charging filament

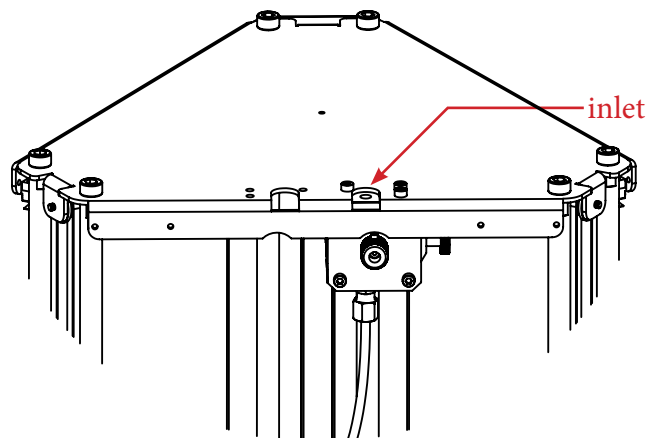
1°) Cut the tip of the filament with a certain angle to make its insertion in the extruder easier



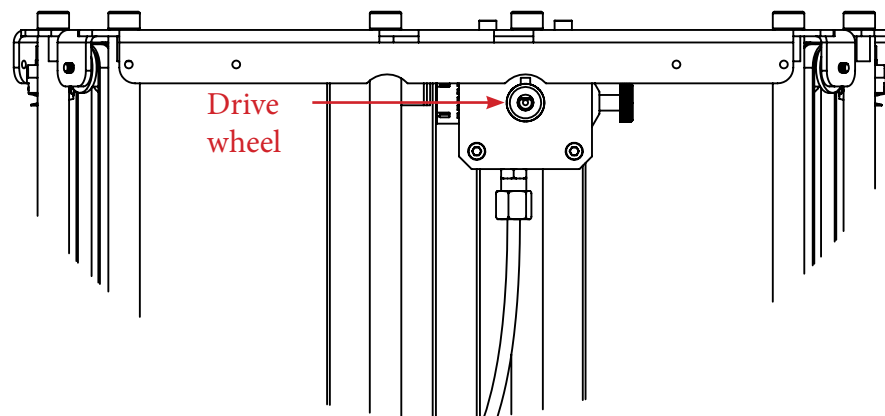
2°) Disengage the extruder



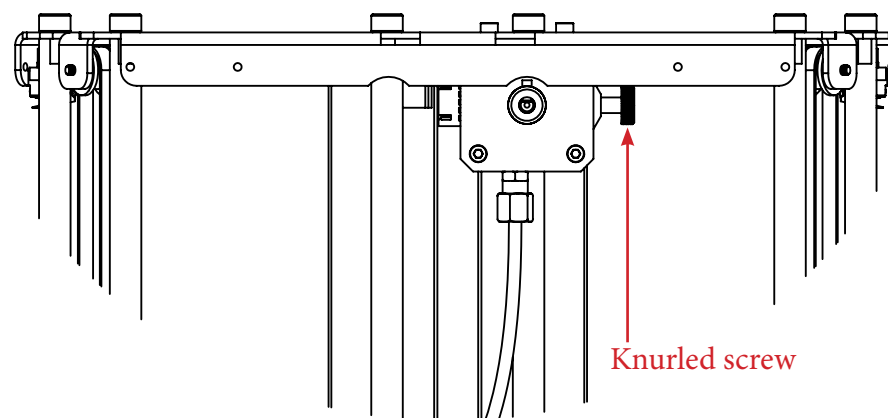
3°) Insert the filament in the extruder's inlet and push it as far as possible



4°) If you encounter a hard point while inserting the filament, engage the extruder and turn the drive wheel until the filament gets to the nozzle, melted filament should start coming out

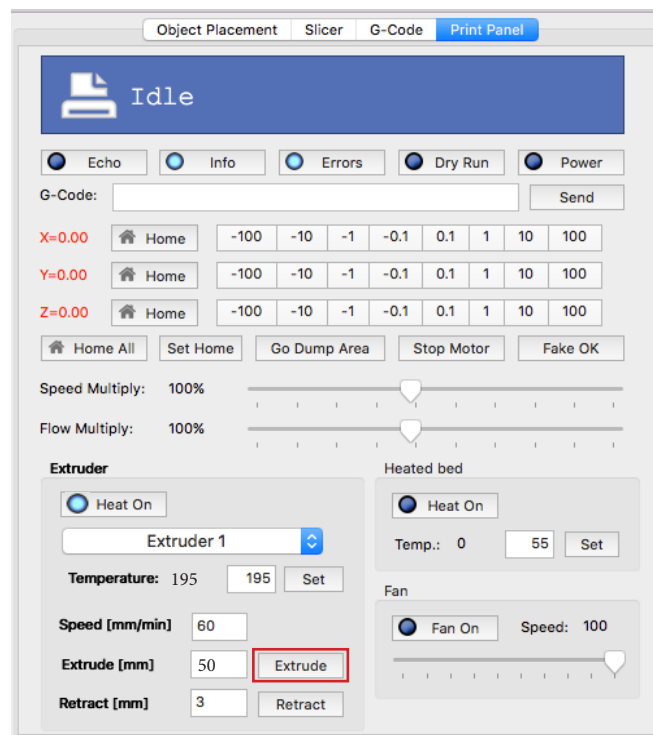


5°) Tighten the spring strongly by turning the knurled screw.
Note: if the knurled screw is not tight enough, the filament will be badly driven



Extrusion test

1°) In the «Print panel» tab, perform a extrusion of 50mm.



2°) Verify that melted filament comes out of the nozzle regularly.

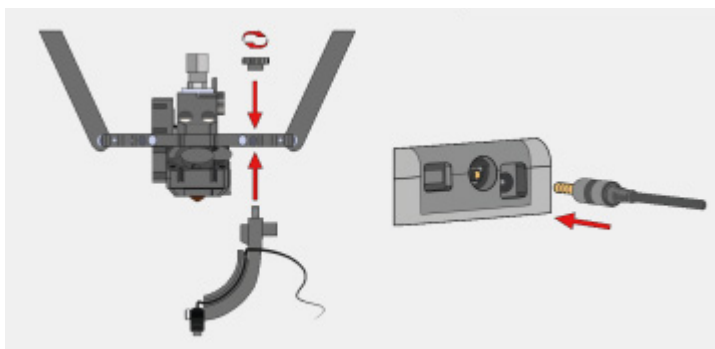
If this is not the case, adjust accordingly the tension exerted on the filament from extruder, and this by turning the Knurled Screw of the extruder.

Calibration process using command lines

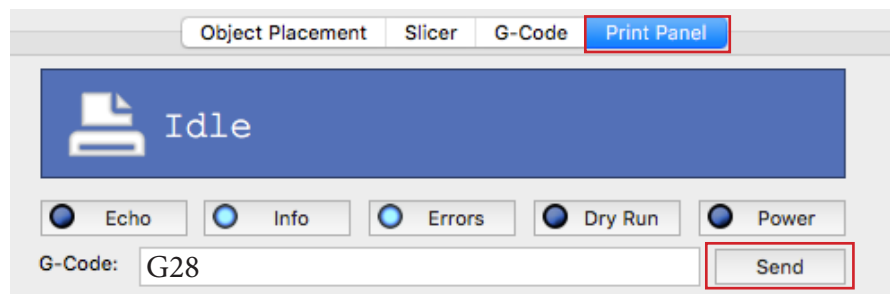
On mac OS computers the calibration process must be done using Gcode commands, here's how to do this:

1°) Make sure the printhead is at room temperature.

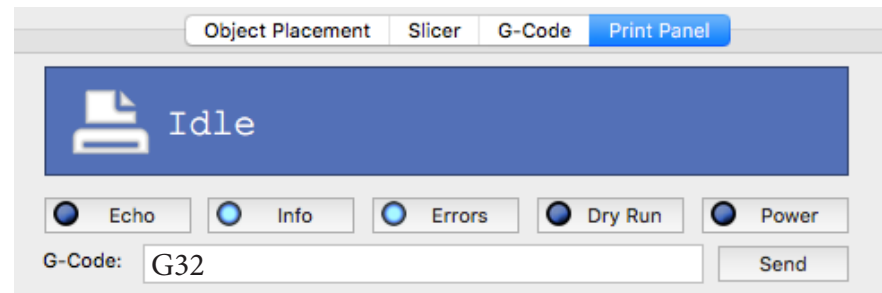
2°) Mount the calibration module on the printer's core and plug the jack connector on the eMotronic board



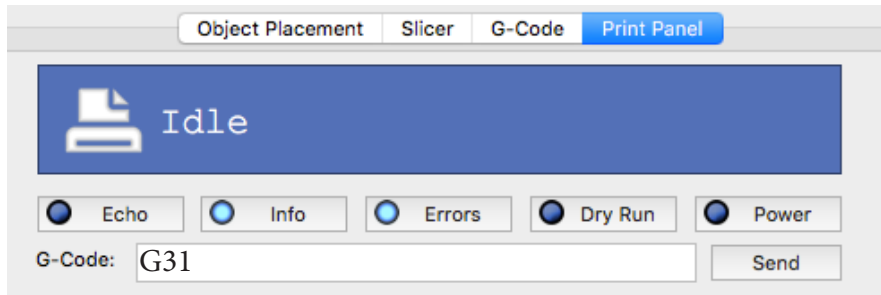
3°) Go to «Print Panel» tab. Home or initialize your printer using the **G28** command.



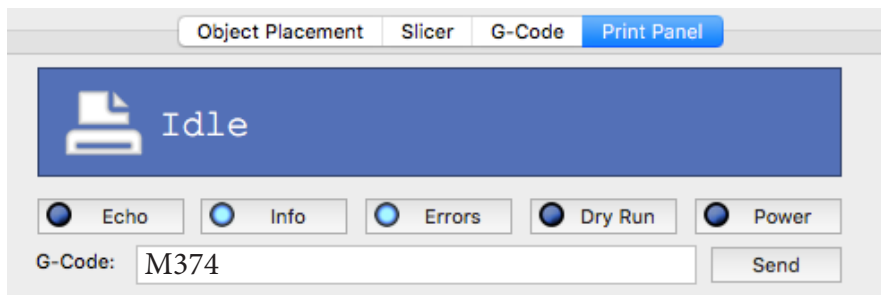
4°) Start the calibration using **G32** command, it will probe 3 points of the bed.



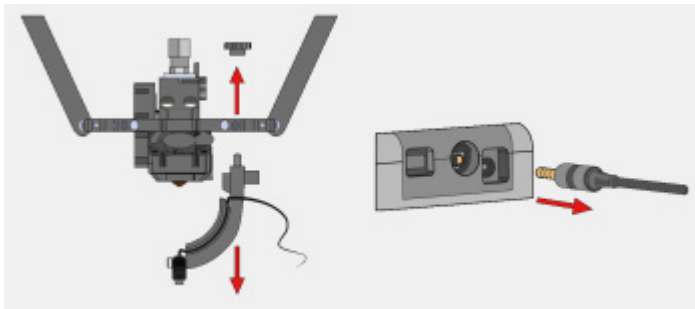
5°) When done, start the full bed calibration using **G31** command, this will probe multiple points of the bed to optimize the first calibration. This is known as grid compensation strategy.



6°) When done, save the grid values using **M374** command.

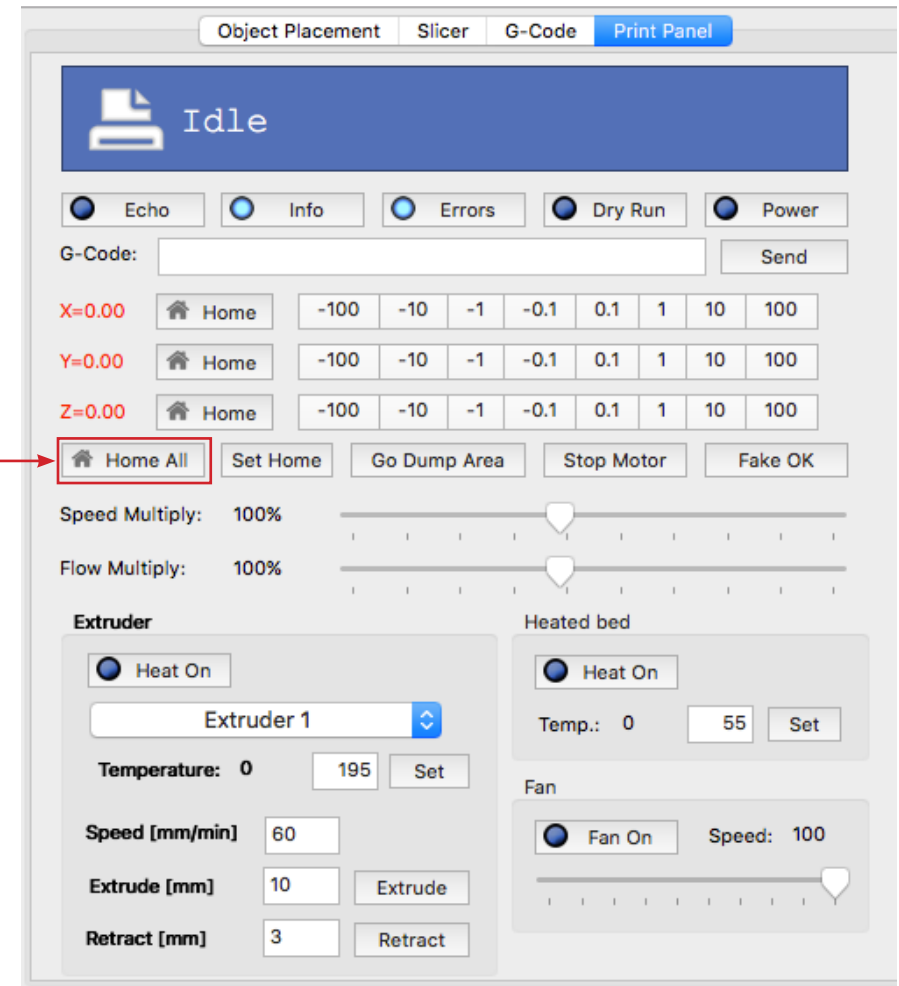


7°) Remove and unplug the calibration module.



Maximum Z height

8°) Home or initialize your printer

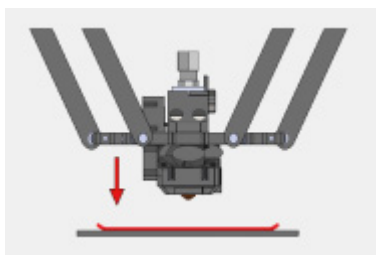


Homing

9°) Place a paper on the bed

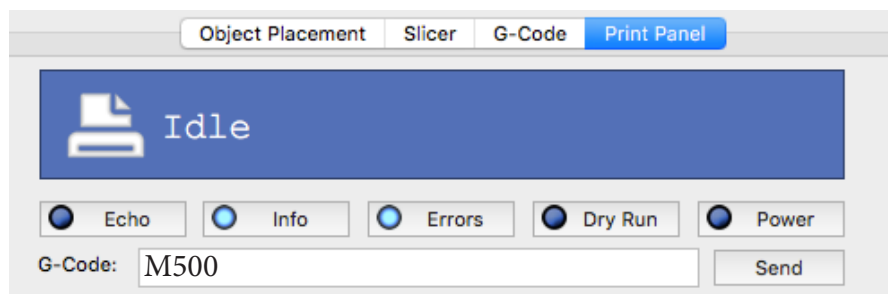
10°) Move the printhead down using the commands shown below in the manual control tab.

Once the printhead is close enough from the paper, move the printhead down slowly by jogging down of 0,1mm at a time, repeat until the printhead touches the paper and holds it from moving.



11°) Set the maximum Z height using the **M306 Z0** command

12°) Save using **M500**



The calibration process is meant to be done once and not before every print. In some cases, the calibration could become unaccurate, after disassembling and reassembling structural parts of you printer per example, you should then calibrate again your printer.

In case of troubles during the calibration process :

Calibration may not be performed correctly. Usually, this comes from the fact that it resides some mechanicals plays on the machine. In this case, it will then be necessary to review the mounting of the printer so as to ensure that everything is tight and without play (smooth rods, ball joints, belts, pulleys, etc.).

Precise adjustment of the maximum height

Objective : refine the height adjustment of the nozzle relative to the heatbed.

Description : if the material hangs too much or not hangs well enough to the 3dBedFix coating, you may need to adjust the nozzle height.

In the case the material doesn't hang well to the coating, please send the following GCODE command lines in the dedicated input field of the «Manual Control» tab :



```
G1 Z0  
M306 Z0.1  
M500
```

In the case where the material hangs too much to the coating and is difficult to unhook :

```
G1 Z0.1  
M306 Z0  
M500
```

In both cases, the value «0.1» can be changed as necessary.

You will then be able to launch an impression and check the following points of details :

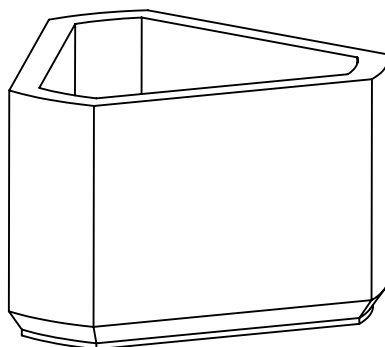
- material that does not hang well to the 3dBedFix = nozzle too far from the 3dBedFix (adjust height)
- material that hangs too much on the 3dBedFix = nozzle too close to the coating (adjust height)

PRINTING

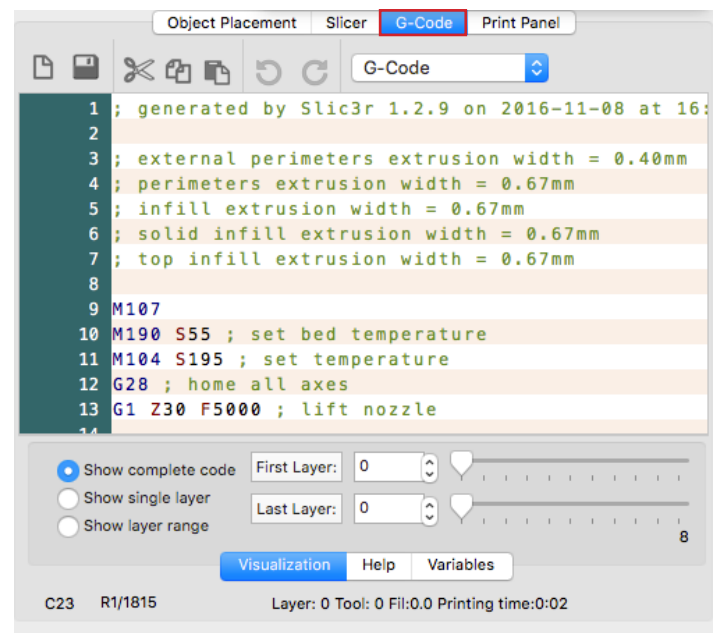
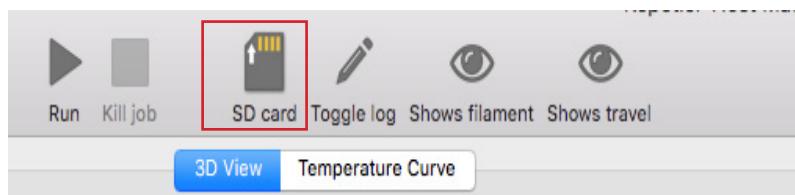
Spool Holder print

You are about to start your first print.

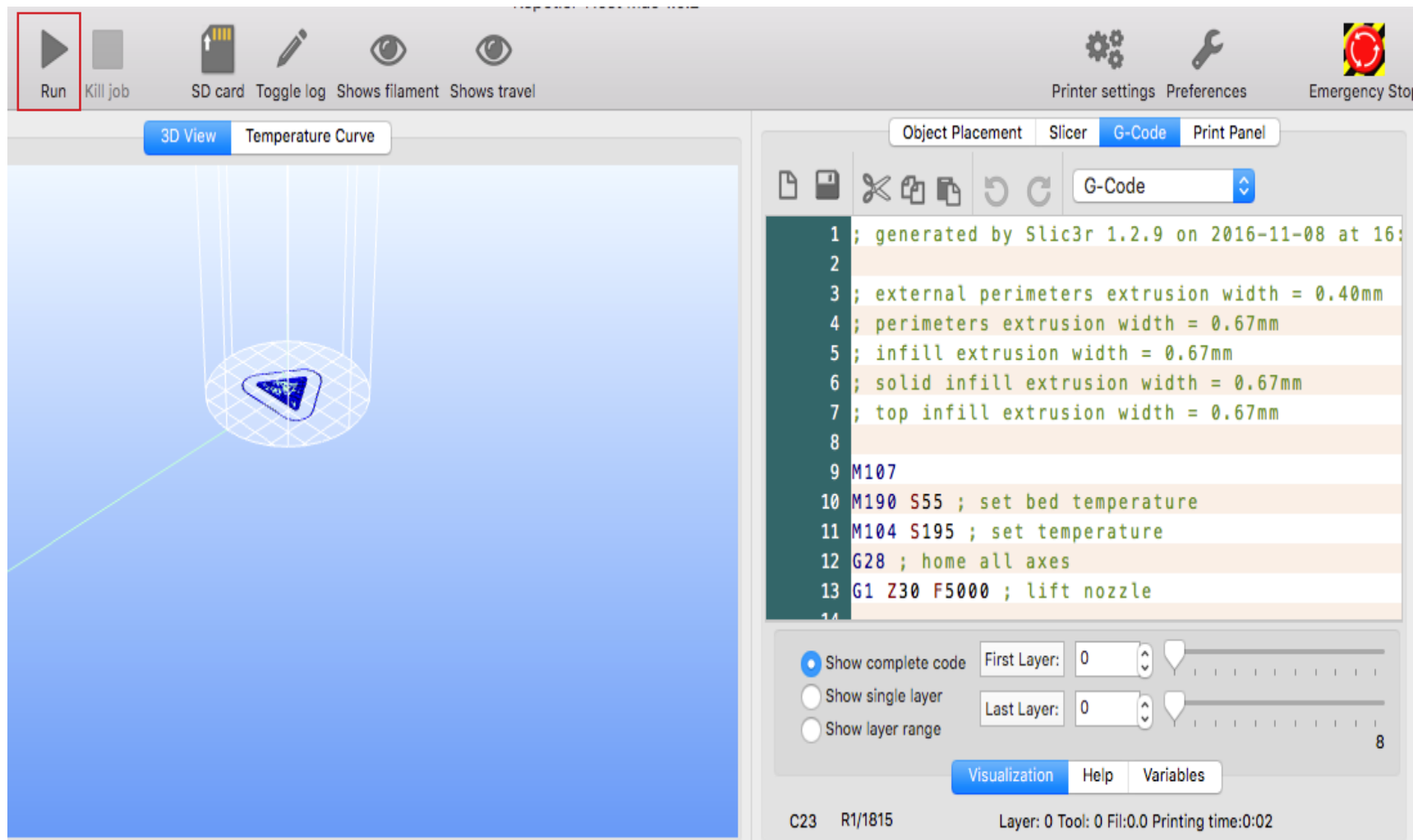
You will be print the spool holder, which will be mounted on the top plate of the MicroDelta Rework.



1°) Load the Gcode called «spool_holder.gcode» from the TF card.



2°) Click on «Run» to start the print.



Print will begin, your printer will initialize (homing) and start heating the extruder and eventually the Heat bed.

CONGRATULATIONS !
**You successfully printed your first
object !**

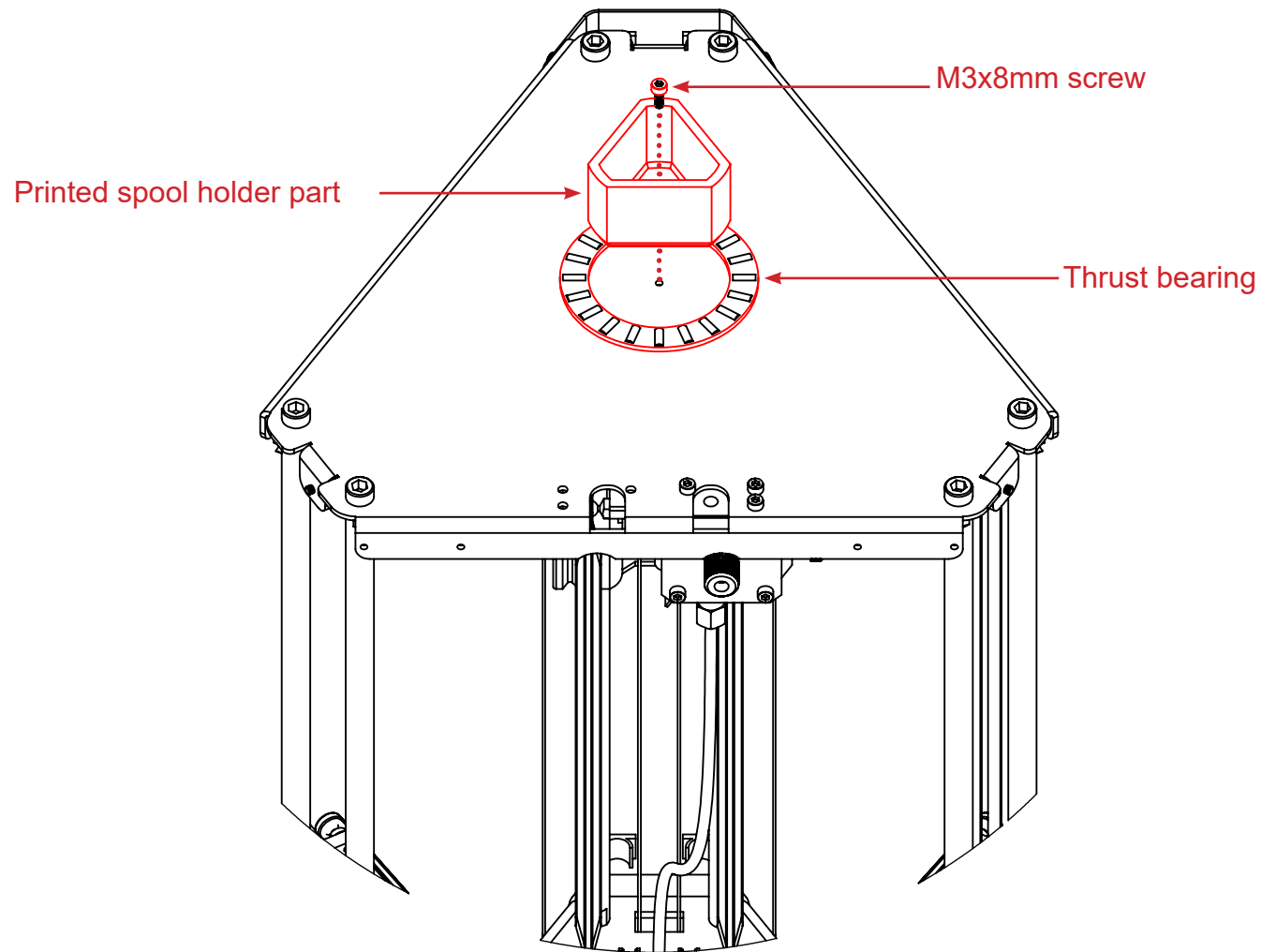


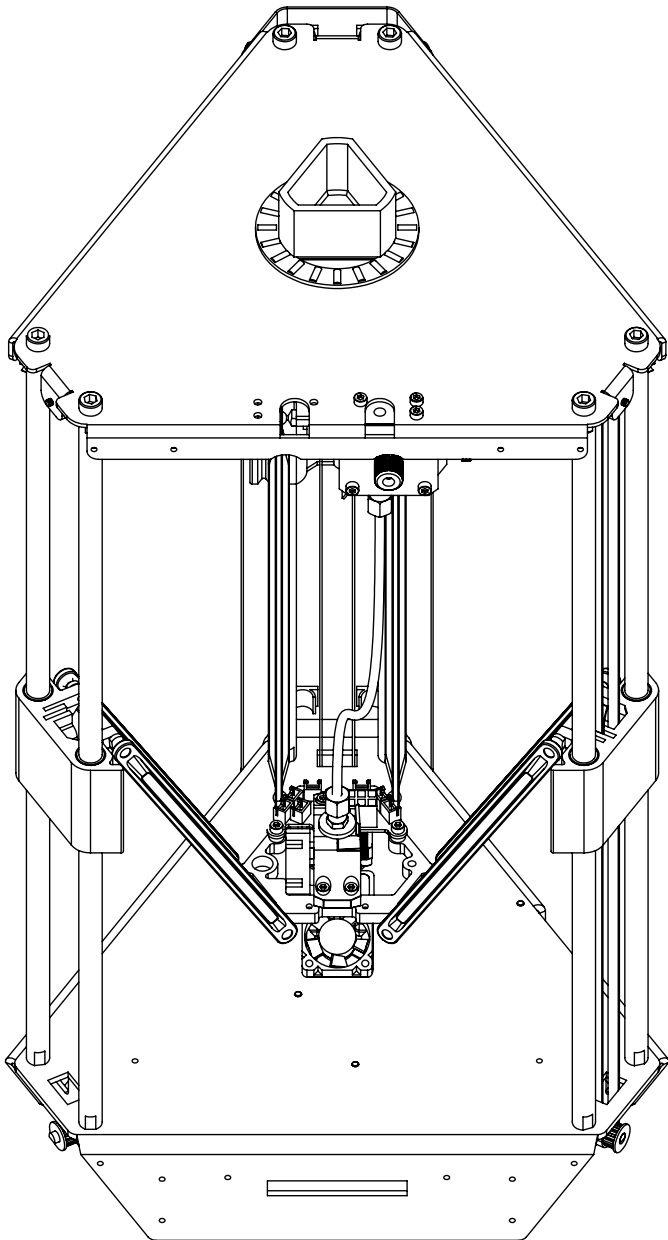
**Go to the «Appendix» to find explanations about mounting the
spool holder.**

Appendix

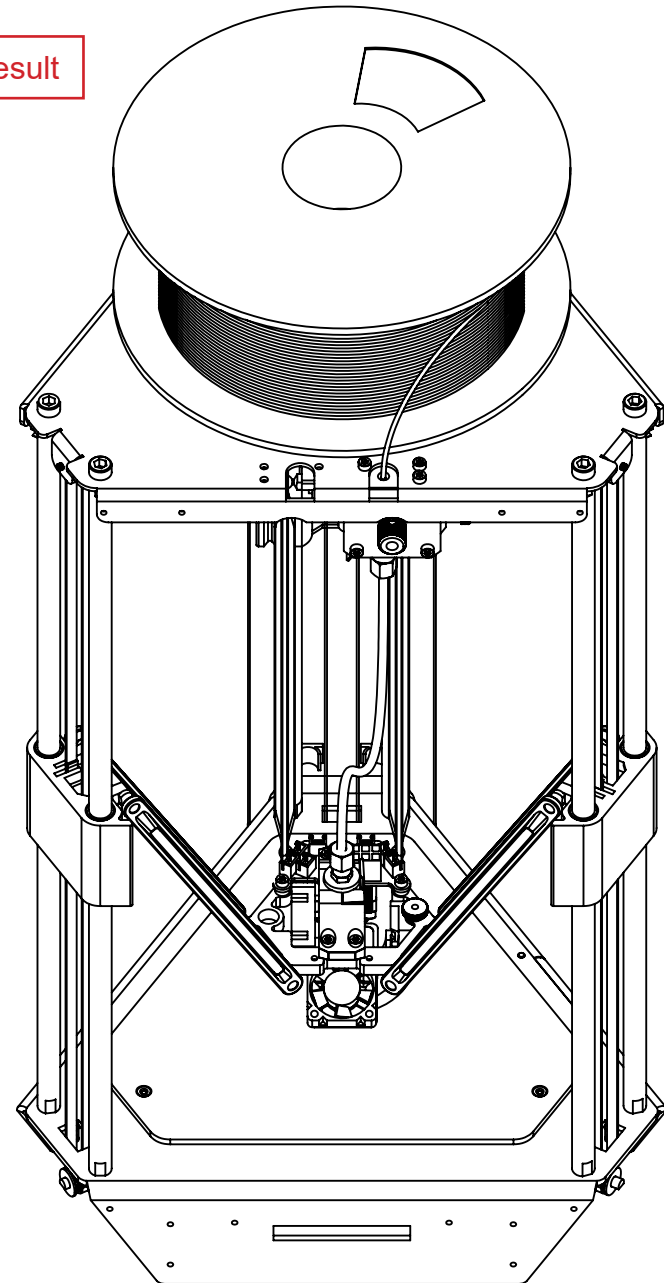
Mounting of the spool holder

Objective : mount the spool holder on the top plate





Result

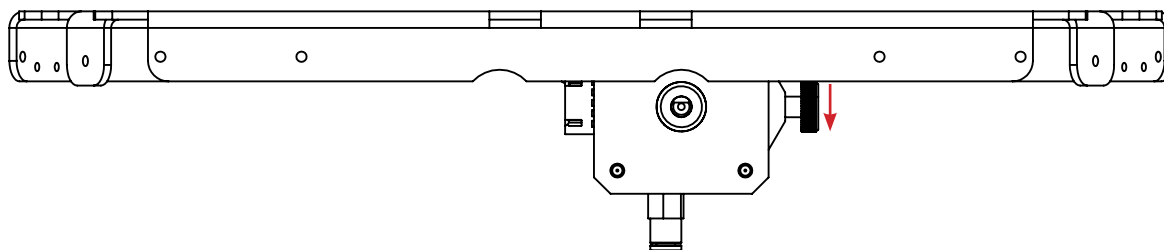


Spring filament pressure adjustment (eMostruder)

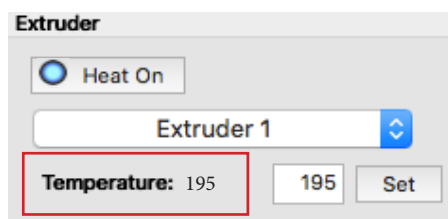
Objective : adjust the pressure exerted by the spring on the filament so that the drive is optimal and does not cause abnormalities.

Description : depending on the type of filament used (PLA, G-FIL, ABS, etc.) and especially its hardness, it will be necessary to adjust the spring pressure so that the driving wheel does not mark the filament too much and does not block in its passage to the nozzle. This method ensures that the filament drive is operational while exerting pressure on the filament as low as possible.

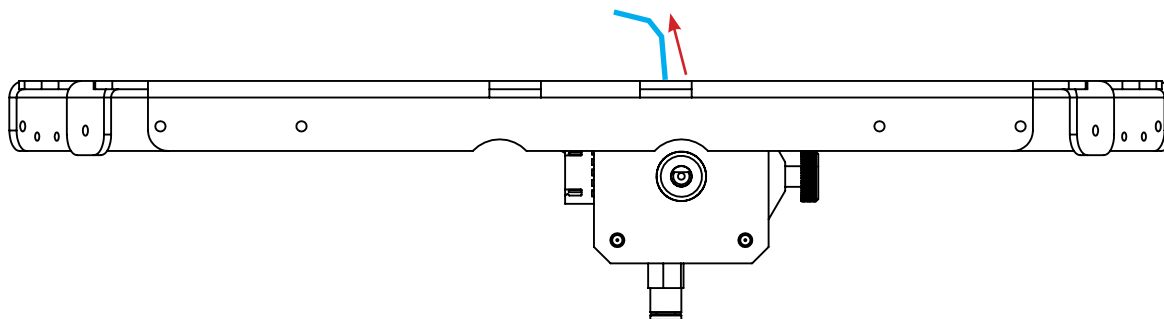
1°) Relieve the spring pressure completely by unscrewing the horizontal knurled screw of the eMostruder



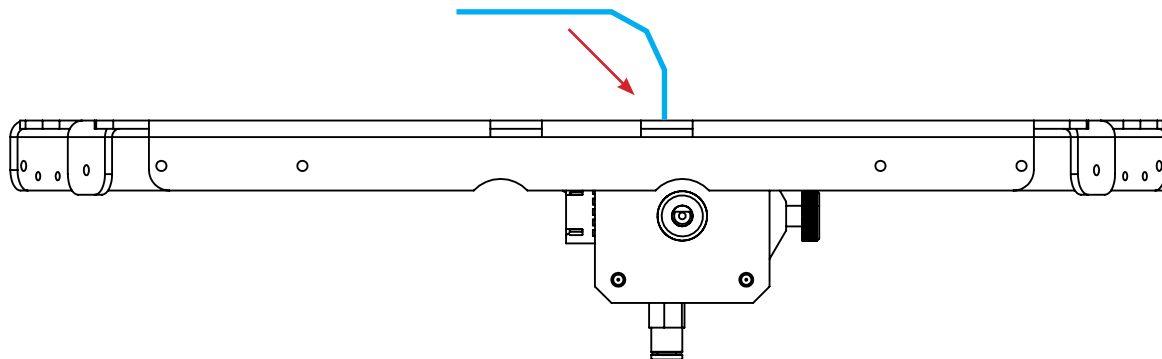
2°) In the Repetier-Host software, with the 3D printer connected and turned on, from the «Manual control» tab, ask for a target temperature dedicated to the used filament and wait until it is reached.



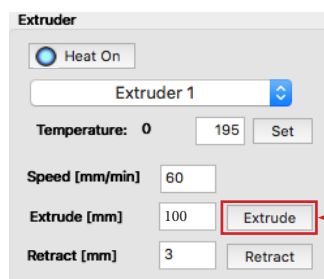
3°) If some filament is already engaged in the machine, remove it and cut it off to a healthy base not marked by the driving wheel.
(with the end of the new filament bevelled in the winding direction of the spool or sample provided)



4°) Engage the new unmarked filament **to the nozzle**.

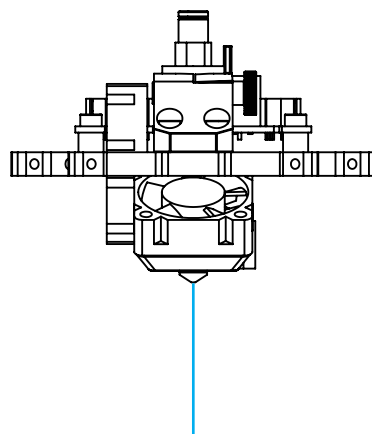


5°) Always in the «Manual Control» tab of Repetier-Host, ask for one or more slow extrusions of 100 mm.

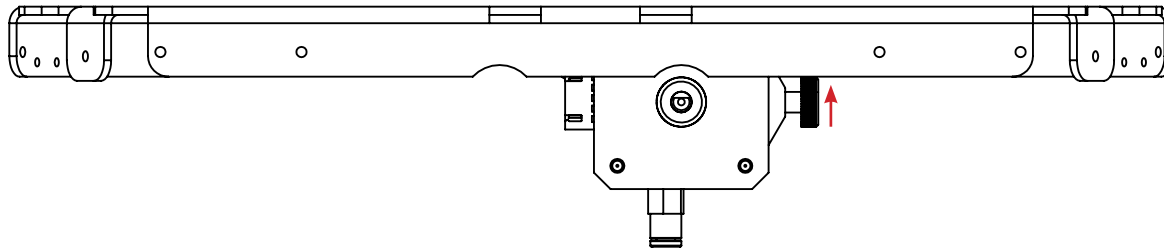


Click the button to extrude

6°) At the same time, observe the filament coming out of the nozzle and check that the extrusion is continuous and homogeneous at a speed corresponding to the speed of the driving wheel.



7°) If the filament does not come out of the nozzle: tighten the screw a little more until it is homogeneous.



Printing a 3D model

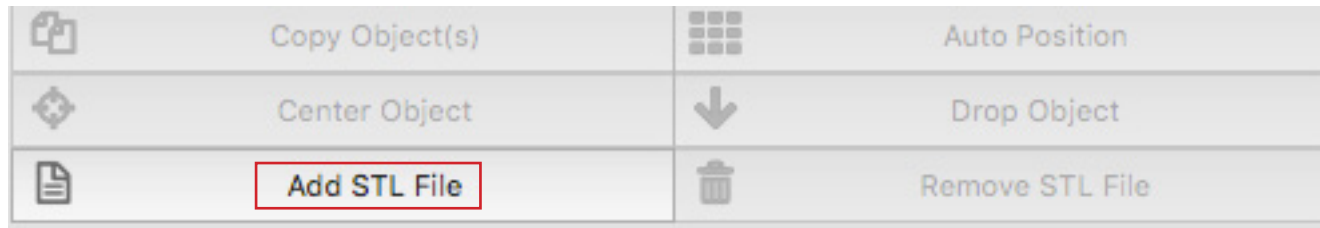
Prerequisite : Fully calibrated 3D printer

1°) Download a 3D part, we suggest you the eMotion Tech's keyring :

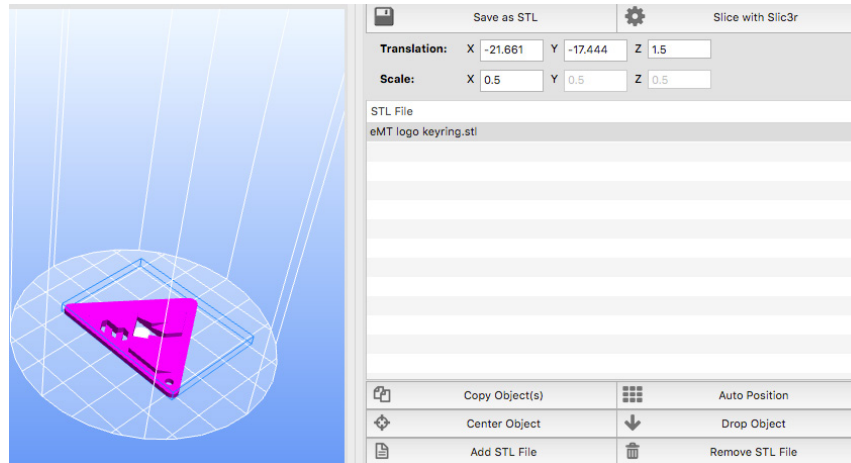
https://data.emotion-tech.com/ftp/Ressources_3D_eMotion_Tech/Porte_clef_eMotion-Tech.stl

2°) Import your model on Repetier-Host :

- In the «Object placement» tab, click on «Add STL file»
- Select your desired 3D model file and open



Virtual representation
of the objet on the print
surface



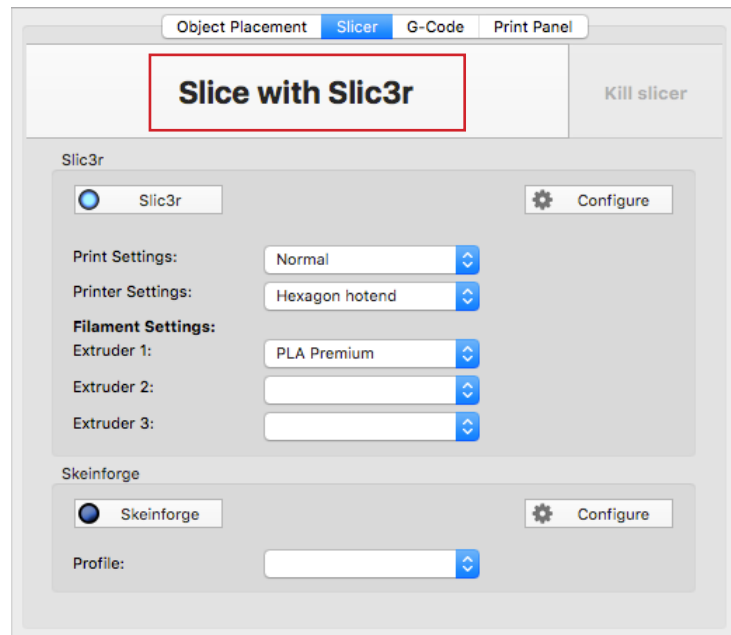
← List of the objects you added

2°) You can edit your object with many options :

	Copy Object(s)		Auto Position
	Center Object		Drop Object
	Add STL File		Remove STL File

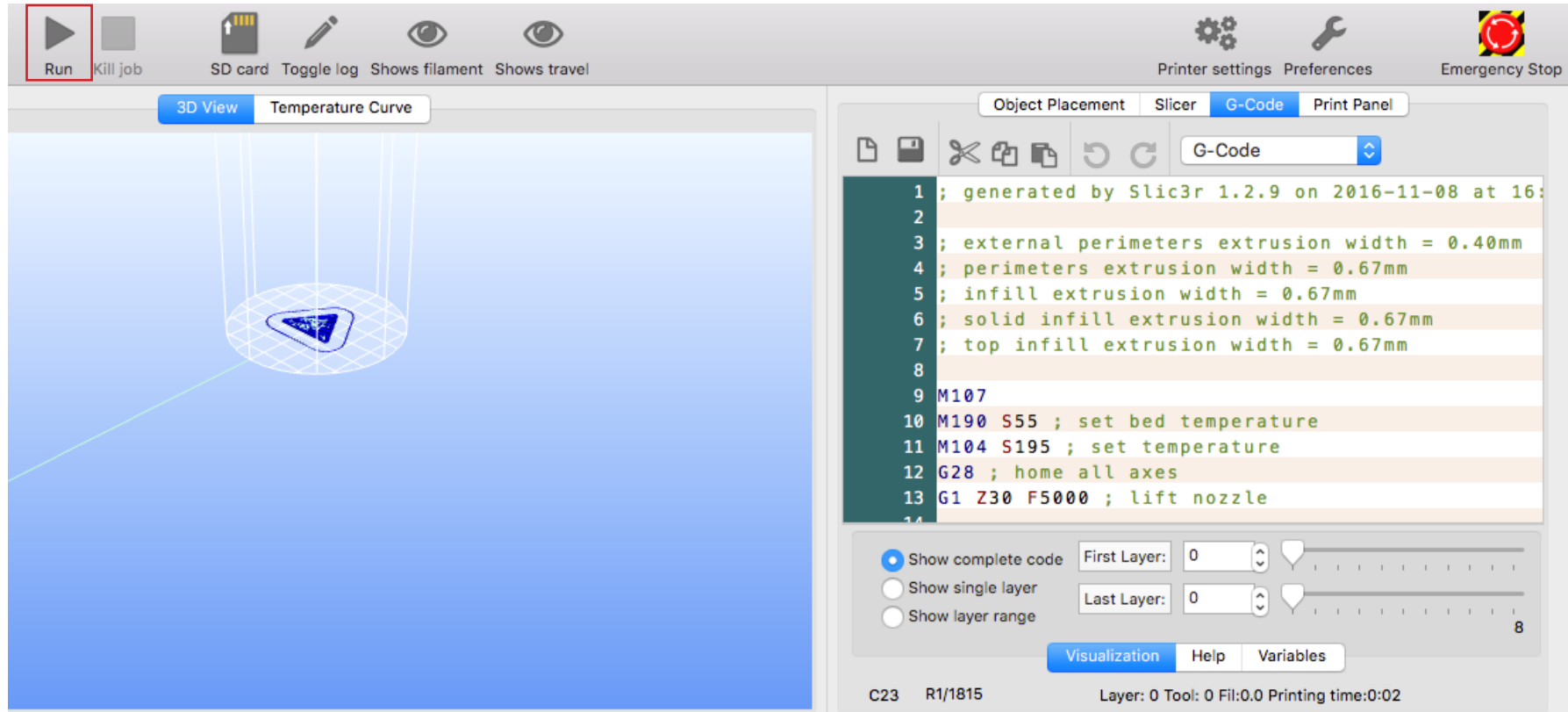
3°) Slice your object with the pre-configured Slic3r profile :

- In the «Slicer» tab, select «Slice with Slic3r».



← When done click on «Slice with CuraEngine»

4°) When slicing is done, a virtualization of the printed object is generated, only one thing left is to click on «Print» button.



To go further

On regular basis updates for your MicroDelta Rework are uploaded on our website. You can find them in the «Support» section of the site, under «MicroDelta Rework / improvements.»

Do not hesitate to regularly check that folder to keep your printer up-to-date.

PID calculation

Preamble : The PID values are necessary for the temperature control of the heating elements. If you find that the target temperature is difficult to reach or the actual temperature varies a lot around the target temperature, it may be useful to recalculate the PID values.

By GCODE commands :

1°) Make sure the print head temperature is at room temperature.

2°) Disable the secondary fans.

3°) In the «Manual control» tab, use the input field for sending GCODE commands :

Send the following command : **M303 E0 S250 C8**

Details :

- E0 = extruder number 1
- S250 = target temperature at 250°C
- C8 = 8 cycles of regulations around the target temperature

Once the command is sent, Repetier-Host logs will show you the progress of the calculation. When the calculation is finished, the new values P, I and D are indicated in the logs.

4°) Send the following GCODE command to save : **M500**

If you have the LCD screen :

In the menu of the screen is integrated an option to directly calculate the PID.

Just go to the «Calibrate / PID hotend» menu.

Maintenance

A monthly maintenance of the 3D printer is recommended.

Below are some recommendations:

Using a brush, dedust the following:

- Emotronic board
- All the fans to ensure a good airflow
- Cold part of the printhead

- To clean the printhead follow the guide dedicated to the Hexagon hotend on the following link:

http://data.emotion-tech.com/highlights_fr/Hexagon%20-%20Notice%20montage-debouchage.pdf

- Clean the teeth of the drive wheel using a sharp ended tool like a needle tip, the end of tweezers or a cutter blade.

- Check & tighten the screws of the 3D printer.

- Lubricate the various mechanical transmission elements with multipurpose grease.

Recommendations

Turning off your 3D printer :

After printing, if you want to turn off the machine, wait until the print head cools down to room temperature to ensure that the print head doesn't get clogged.

Transport :

If the printer has to be transported by car or another mean of transport in which it could be subject to vibration, it is recommended to unplug all the motors from the eMotronic board to avoid damaging it.

Troubleshooting :

A FAQ on the MicroDelta Rework is available on our website in the «Support» section, please refer to it to troubleshoot your printer, most problems could be solved through this tool !

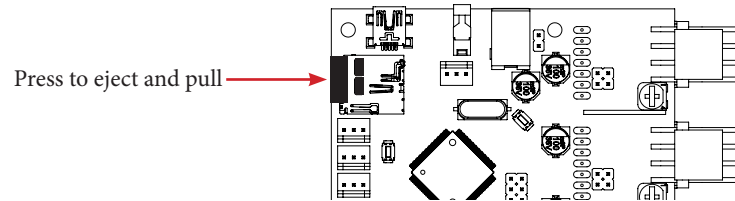
Appendix

Restore the TF card

Objective : prepare the files of the TF card again in order to eliminate the corrupted files and defective cells.

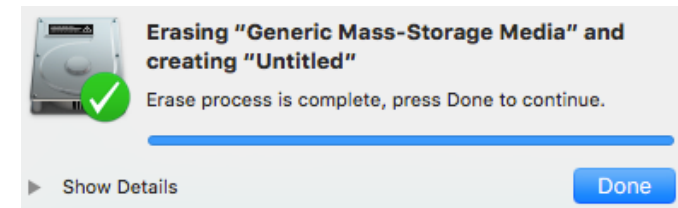
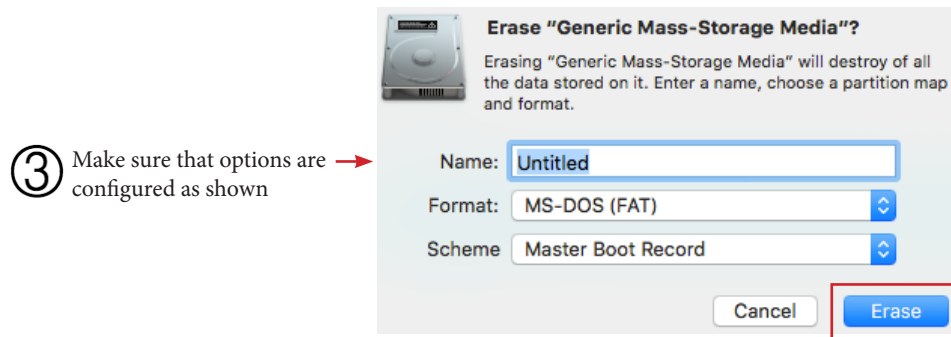
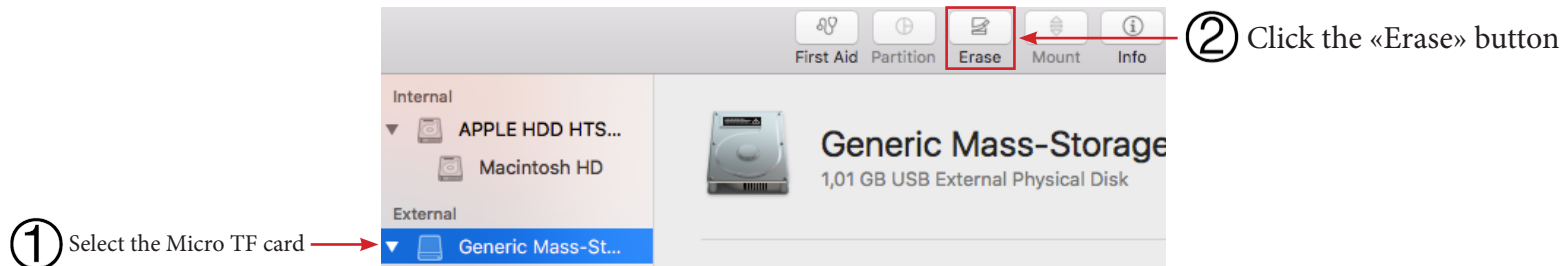
Description : for various reasons, the files present in the TF card and / or the cells of this card can be damaged. You will find through this procedure how to restore the TF card and the necessary files to MicroDelta Rework.

1°) **Remove the TF card by pressing it, you will hear a click indicating that the card is no longer locked and can be removed without damage to the reader.**

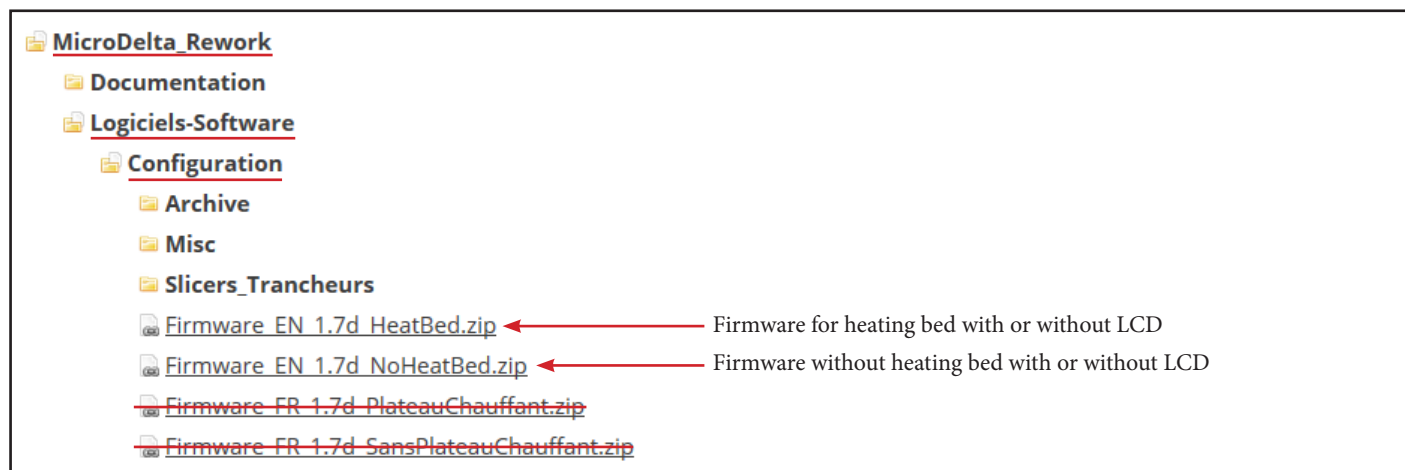


2°) Then read this TF card with an external drive on your computer (very useful because the formatting is too long by USB cable).

3°) Perform a formatting of the TF card in MS-DOS FAT format using the tool named «Disk Utility».



4°) Download the latest firmware on our website «reprap-3d-printer.com» in the support section, in the tree dedicated to your machine.



5°) Go inside the firmware folder, select all the files and finally copy them to the root of the TF card.

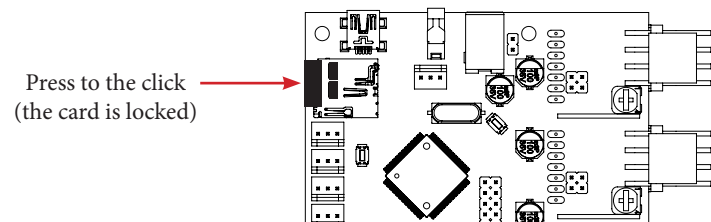
Name	^	Date Modified	Size	Kind
config.txt		24/04/17 16:40	17 KB	Plain Text Document
firmware.bin		24/04/17 16:46	379 KB	MacBinary archive
Read_Me.txt		27/04/17 09:39	1 KB	Plain Text Document
Spoolholder_V2.gcode		08/03/17 09:54	1,2 MB	TextEdit Document

6°) Eject the TF card (right click on the TF card then «Eject»).



7°) Disconnect the Micro TF card reader from the computer.

9°) Insert the TF card into the eMotronic drive.

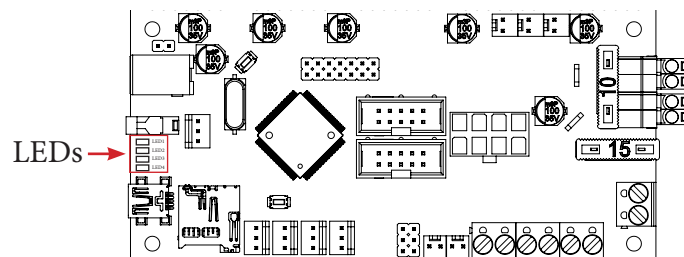


10°) Press the «Reset» button located on the lower metal part to take into account the new firmware.

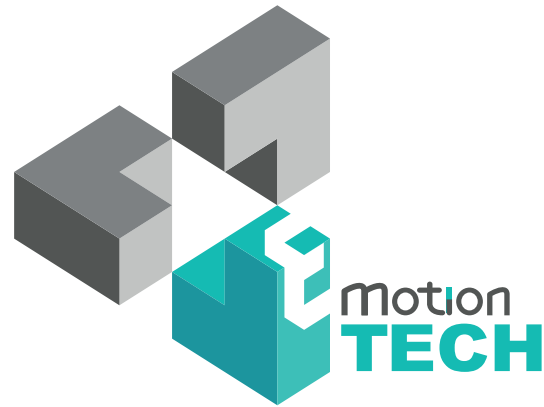
11°) In the TF card, make sure that the «firmware.bin» file is changed to «firmware.cur».

12°) On the eMotronic board, between the USB connector and the auto-leveling sensor connector, make sure that:

- LED1 remains on
- LED2 and LED3 flash continuously
- LED4 remains on



From there, the eMotronic card should be recognized by the Device Manager and the contents of the TF card should be displayed in the Files Explorer. It will then only remain to realize again the calibration of the 3D printer.



Thank you for choosing us!

www.emotion-tech.com