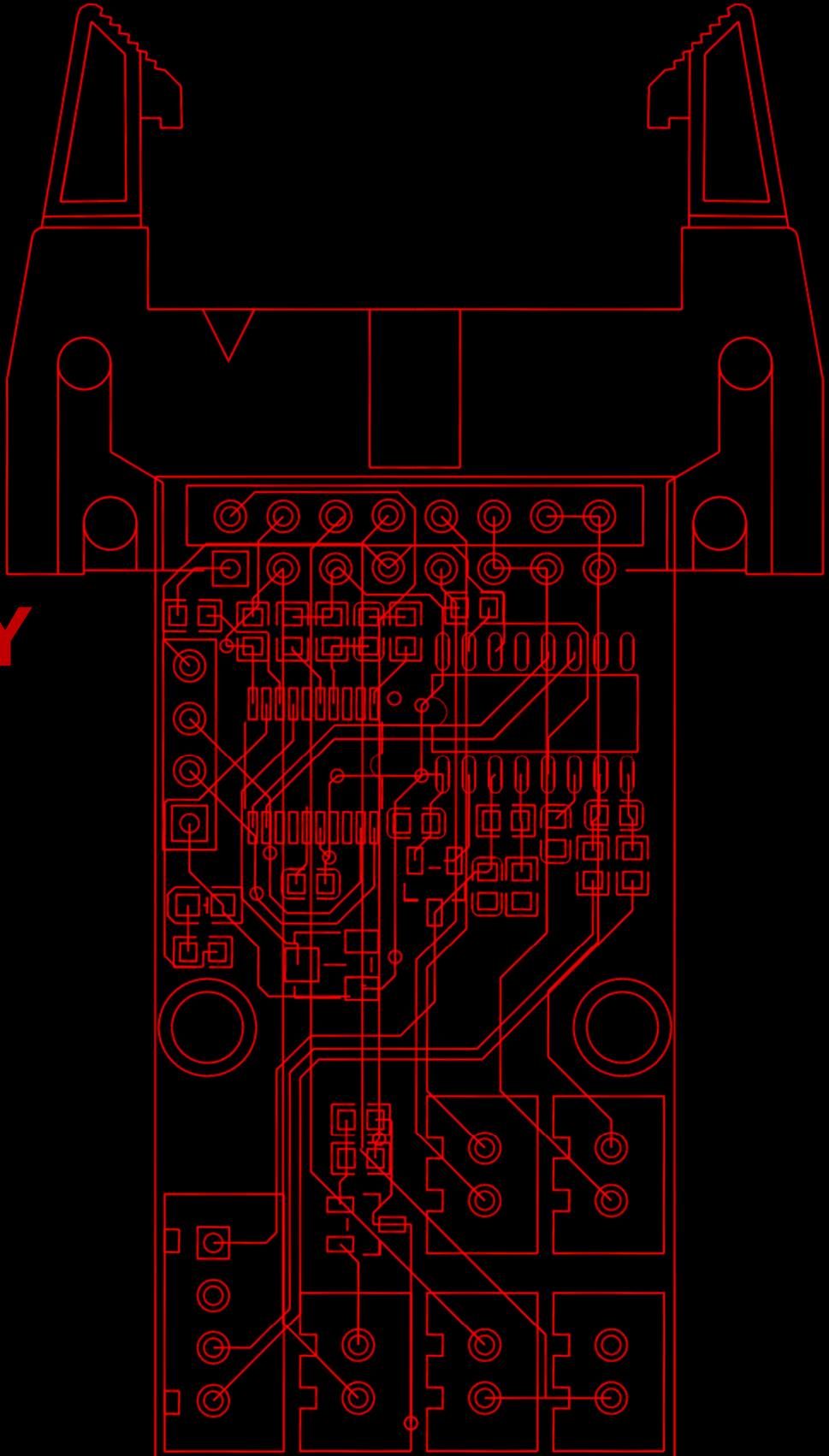


CREALITY CR-6 SE

HOT END CIRCUIT BOARD
COMMUNITY EDITION



REV: 0.1

CREALITY
Community Edition

Preface

This document contains the required information to order the community version of the CR-6 SE Hotend circuit board for your CR-6 SE or CR-6 MAX printers. This is a reverse engineered version of the Creality circuit board of which no files have been released officially.

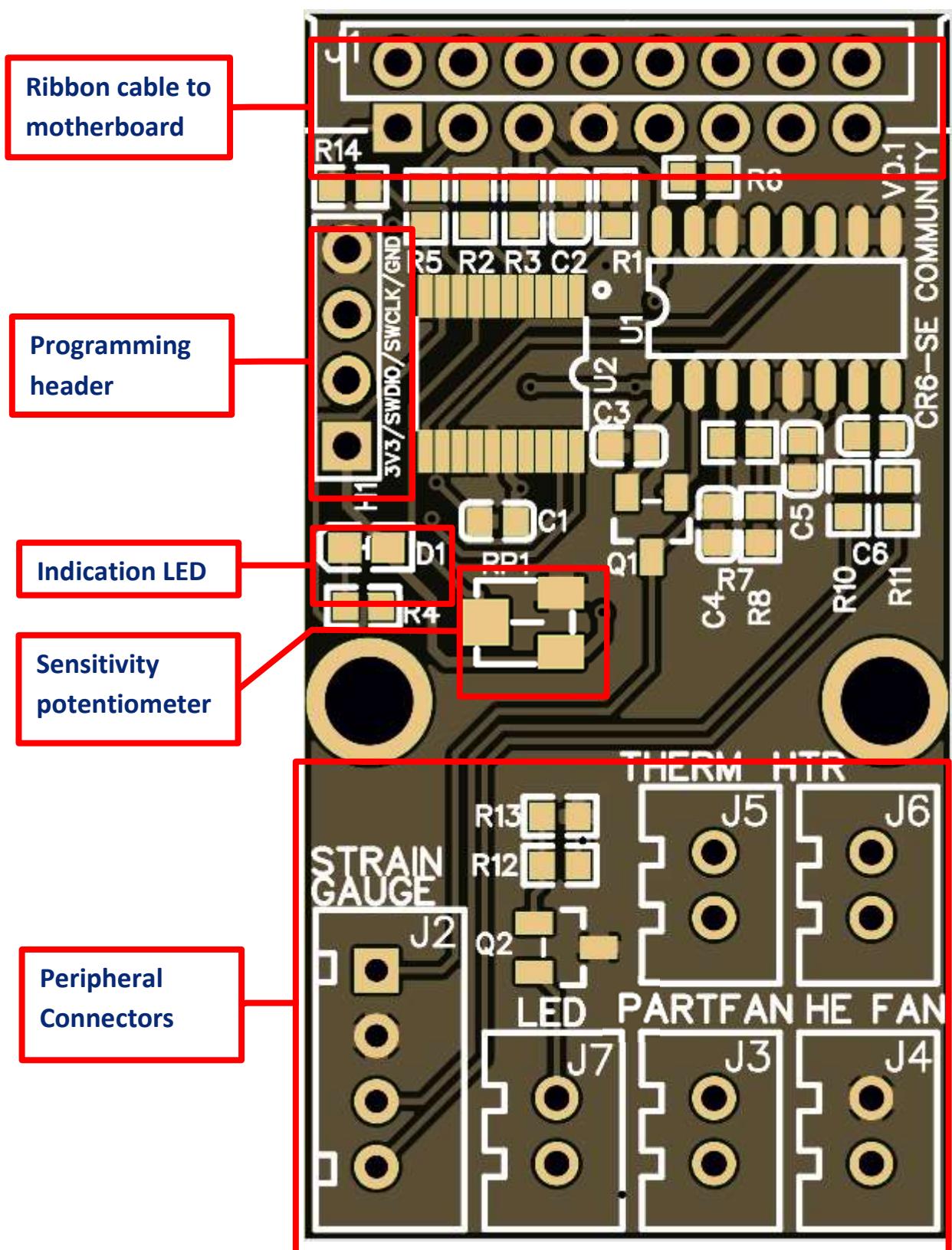
Disclaimer

Ordering, programming and using this opensource version of the circuit board is at your own risk. The community and designers are by no means responsible for any errors, malfunctions or damages resulting from using this circuit board or any procedures in this manual.

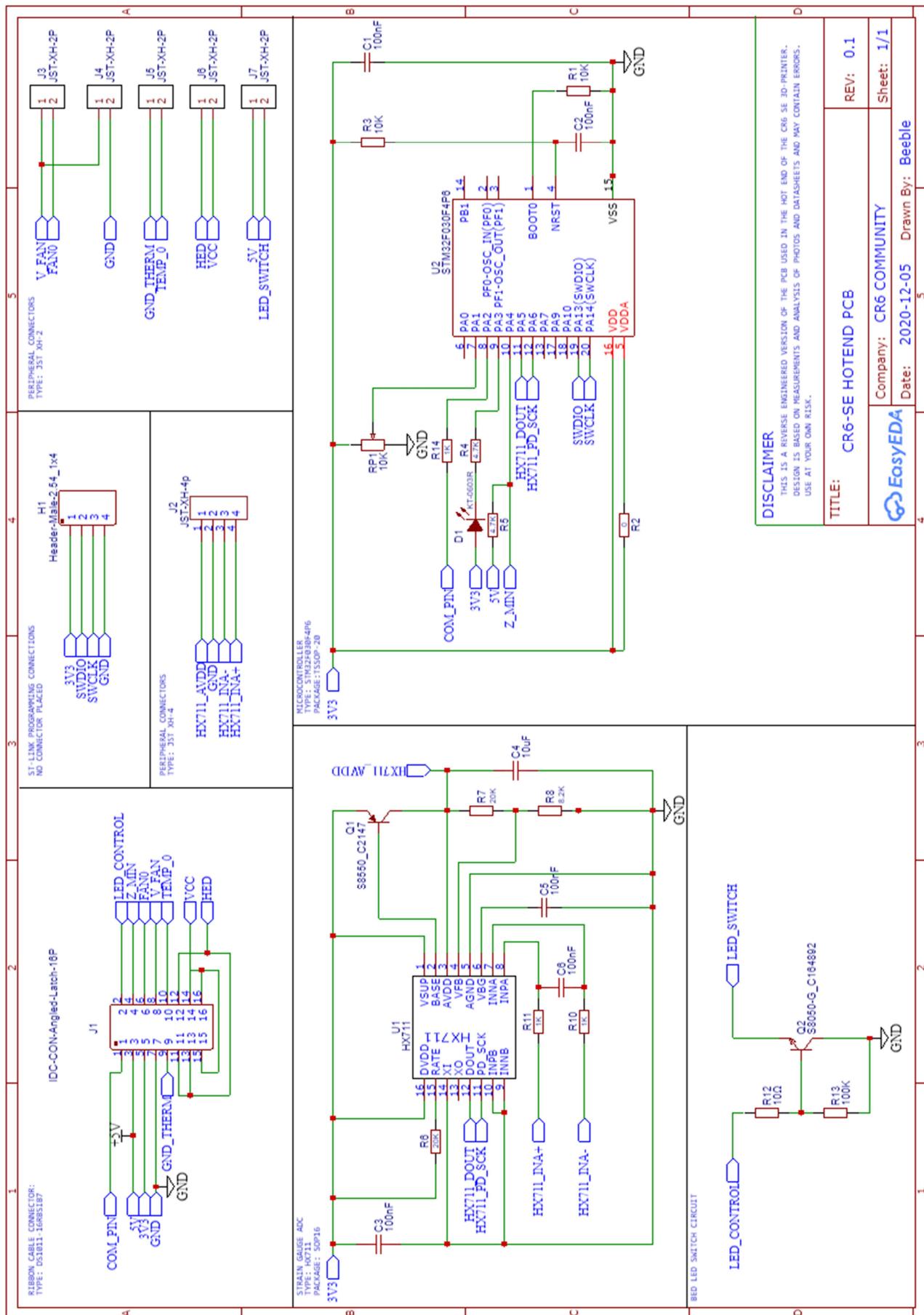
Index

Circuit board layout	4
Circuit board schematic	5
Ordering the PCB	6
Step 1: Uploading the Gerber files.....	7
Step 2: The Bill of Materials and pick and place files.....	8
Step 3: Final check.....	9
How to flash	10
Step 1: Preparation	10
Step 2 (optional) Updating firmware	11
Step 3: Preparing the ST-LINK programmer (optional)	12
Step 4: Flashing	12
Installation	14
Step 1: Removing the fan shroud.....	14
Step 2: Disconnecting the peripheral connectors.....	14
Step 3: Removing the old circuit board.....	15
Step 4: Installing the new circuit board	15
Testing and calibrating.....	16
Step 1: Testing	16
Testing the LED.....	16
Testing the fans	16
Testing the heater and thermistor	16
Testing the strain gauge	16
Step 2: Calibrating the strain gauge.....	17
Measuring	17
Adjusting	17

Circuit board layout



Circuit board schematic



Ordering the PCB

The circuit board has been designed in the online design software *EasyEDA*, this is software developed and maintained by the *JLCPCB* circuit board manufacturer.

All the required files can be found on the CR-6 community Github:

<https://github.com/CR6Community/Hardware>

File	Description
Bill of materials	<i>All the components on the circuit board</i>
EasyEDA files	EasyEDA project files (Schematic and board)
Gerber files	Files required for manufacturing of the circuit board
Pick&place	Files required for placing the components during manufacturing

For ordering the circuit board the Bill of materials, Gerber files and Pick and Place files will be required.

If you do not have an account on the JLCPCB website (<https://jlpcb.com/>), create one first.

IMPORTANT, READ THIS FIRST!

When ordering from JLCPCB, there are some components that cannot be supplied and placed during manufacturing, these components are the following:

- 1) Ribbon cable connector type: IDC16-RA-Latched.**
- 2) Peripheral connectors type: JST-XH (2x1 and 4x1).**

JLCPCB also has a minimum of 2 pre-assembled circuit boards out of 5 bare circuit boards.

Step 1: Uploading the Gerber files.

The screenshot shows the JLPCB website's PCB ordering interface. The main form is titled "Add your gerber file" and includes fields for Layers (2 selected), Dimensions (100x100 mm), PCB Qty (5), Delivery Format (Single PCB), PCB Thickness (1.6), PCB Color (Green), Surface Finish (HASL), Copper Weight (1 oz), Gold Fingers (No), Confirm Production file (Yes), Flying Probe Test (Fully Test), and Castellated Holes (No). On the right, "Charge Details" show a Special Offer of €1.62 for 1-2 days delivery. A "SAVE TO CART" button is visible.

Press the “Add your Gerber file” button and upload the Gerber .zip file. The only settings that may be changed are the quantity and the circuit board color. However, in case you want to order the circuit board pre-assembled with all the surface mounted components the color choices are limited the either green or black for low quantities. In addition, the option “Assemble top side” must be selected for the components to be placed during manufacturing. There are some important downsides to this:

- The minimum quantity of boards ordered must be higher than 5.
- The number of boards that will come pre-assembled must at least be 2.
- Only surface mounted components can be installed during manufacturing.

Step 2: The Bill of Materials and pick and place files.

Once done press the “Save to Cart button”. Now the Bill of Materials (BOM) and the Pick and Place (CPL) files need to be uploaded.

The screenshot shows the JLCPCB website with a progress bar at the top. The second step, 'Upload BOM/CPL', is highlighted. Below the bar, there are two main input fields: 'Add BOM File' and 'Add CPL File'. Each field has a 'View Sample' link. A note below the fields specifies file types: 'Only accept XLS,XLSX,CSV.' for BOM and 'Pick&Place File.Only accept XLS,XLSX,CSV.' for CPL. A tip at the bottom left says 'Not sure where to start? Check our [SMT FAQs page](#).'. Another tip at the bottom right says 'Tips: With EasyEDA, you can generate BOM/CPL files with a single click.' At the bottom right of the form area is a 'NEXT' button.

Once uploaded a table of all the components will be shown. This is the list of components that JLCPCB can supply. Any component that is not “confirmed” cannot be delivered and installed by JLCPCB. All connectors and through-hole components must be supplied and installed yourself by default. Any other components that JLCPCB does not have in stock must also be supplied and installed yourself.

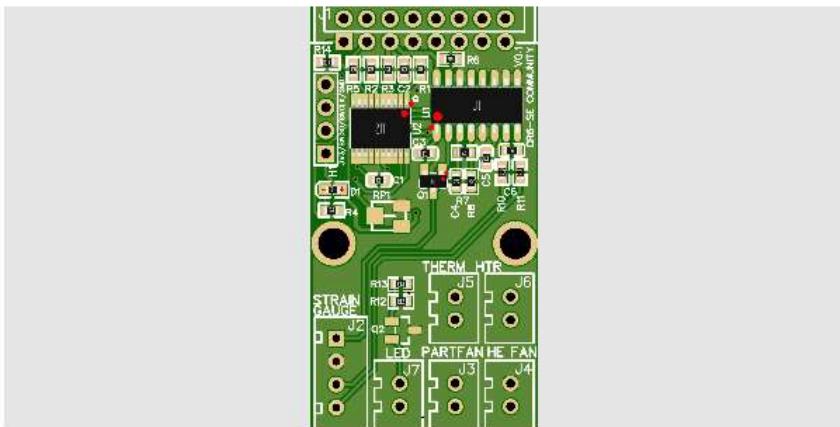
The screenshot shows the JLCPCB website with the 'Review Matched Parts' table. The table lists six components with their details and status. The columns are: Designator, Name, Footprint, Matched Part Detail, Qty, Unit Price, and Operate. The 'Operate' column contains buttons for 'confirmed' (with a checkmark icon) and 'not confirmed' (with a crossed-out checkmark icon). The table shows the following data:

Uploaded BOM Data			Review Matched Parts				
Designator	Name	Footprint	Matched Part Detail	Qty	Unit Price	Operate	
U1	HX711	SOP-16_L10.0-W3.9-P1.27-LS6.0-BL	HX711 SOP-16_3.9x10.1x1.27P	Extended Part C43656	1	€0.4340	confirmed
RP1	10K	RES-ADJ-SMD_3313J-1	10KΩ ±20% ±100ppm°C SMD-R_3_3.4x3.4x1.05P	Extended Part C132891	1	€0.5713	confirm
Q2	S8050-G_C164892	SOT-23-3_L2.9-W1.3-P1.90-LS2.4-BR	S8050-G SOT-23-3	Extended Part C164892	1	€0.0097	confirm
Q1	S8650_C2147	SOT-23-3_L2.9-W1.3-P1.90-LS2.4-BR	S8650 SOT-23-3	Extended Part C2147	1	€0.0154	confirmed
R5,R4	4.7K	R0603	4.7KΩ(472) ±5% 0603	Extended Part C105428	2	€0.0006	confirmed
R13	100K	R0603	100KΩ (1003) ±1% 0603	Basic Part C25803	1	€0.0017	confirmed

Step 3: Final check

Once done, scroll down and press “Next”. On this page the list and physical placement of the components that will be installed during manufacturing is shown. Please check carefully if the locations line up to the markings on the circuit board.

Review Parts Placement



- The parts placement is for reference purpose only. If you are sure the rotation and polarity of your design are correct, you can omit the preview and continue placing your order.

Selected Parts(16 items) [?](#)

Part Detail	Selected By	Designator	Price
Uniroyal Elec 0803WAF8201T5E Basic Part: C25981 Chip Resistor - Surface Mount ...	System	R8	Qty: 2*1 Ext Price: €0.0292 ?
Uniroyal Elec 0803WAF1003T5E Basic Part: C25803 Chip Resistor - Surface Mount ...	System	R13	Qty: 2*1 Ext Price: €0.0341 ?
Avia Semicon (Xiamen) HX711 Extended Part: C43656 Analog To Digital Converters (A...	System	U1	Qty: 2*1 Ext Price: €3.2675 ?
YAGEO RC0803JR-074K7L Extended Part: C105428 4.7K ±5% 1/10W ±100ppm/°C ...	System	R5,R4	Qty: 2*2 Ext Price: €0.0730 ?
Samsung Electro-Mechanics C19856 Basic Part: C19856 Multilayer Ceramic Capacitors ...	System	C1	Qty: 2*1 Ext Price: €0.2962 ?

Charge Details

Special Offer: €3.24

SMT Price:

Setup fee: €5.68

Stencil: €1.22

Panel: €0.00

Large Size: €0.00

Components: €6.87

Extended components fee: €7.30

Assembly: €1.13

Build Time:

PCB: 2-3 days €0.00

SMT: 72 hours

Total Price: €26.43

Weight: 33.96g

SAVE TO CART

By placing your order, pls read the [Terms and Conditions of JLCPBCB SMT Service](#).

Once the result is satisfactory you can press “Save to cart” and order the circuit boards.

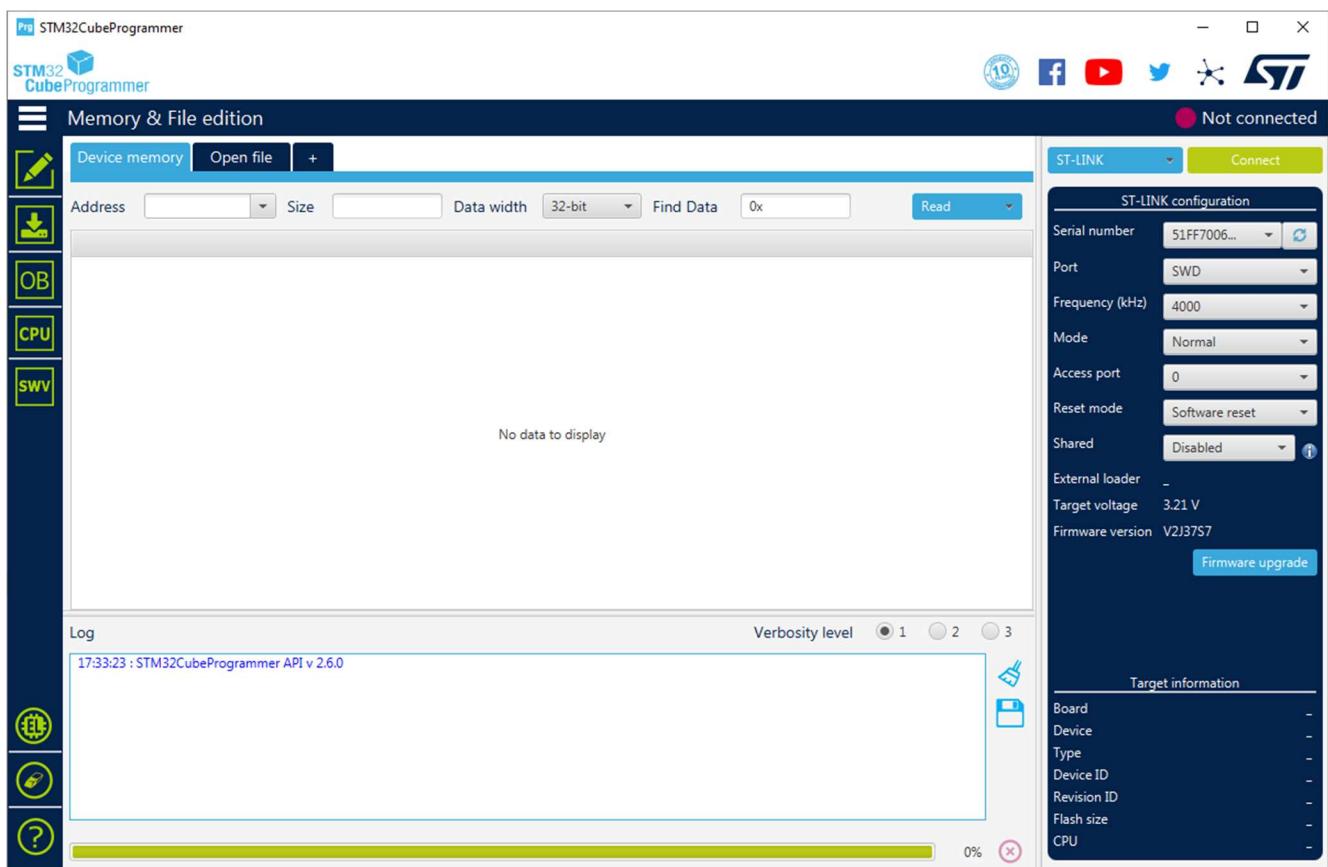
How to flash

Step 1: Preparation

Required components:

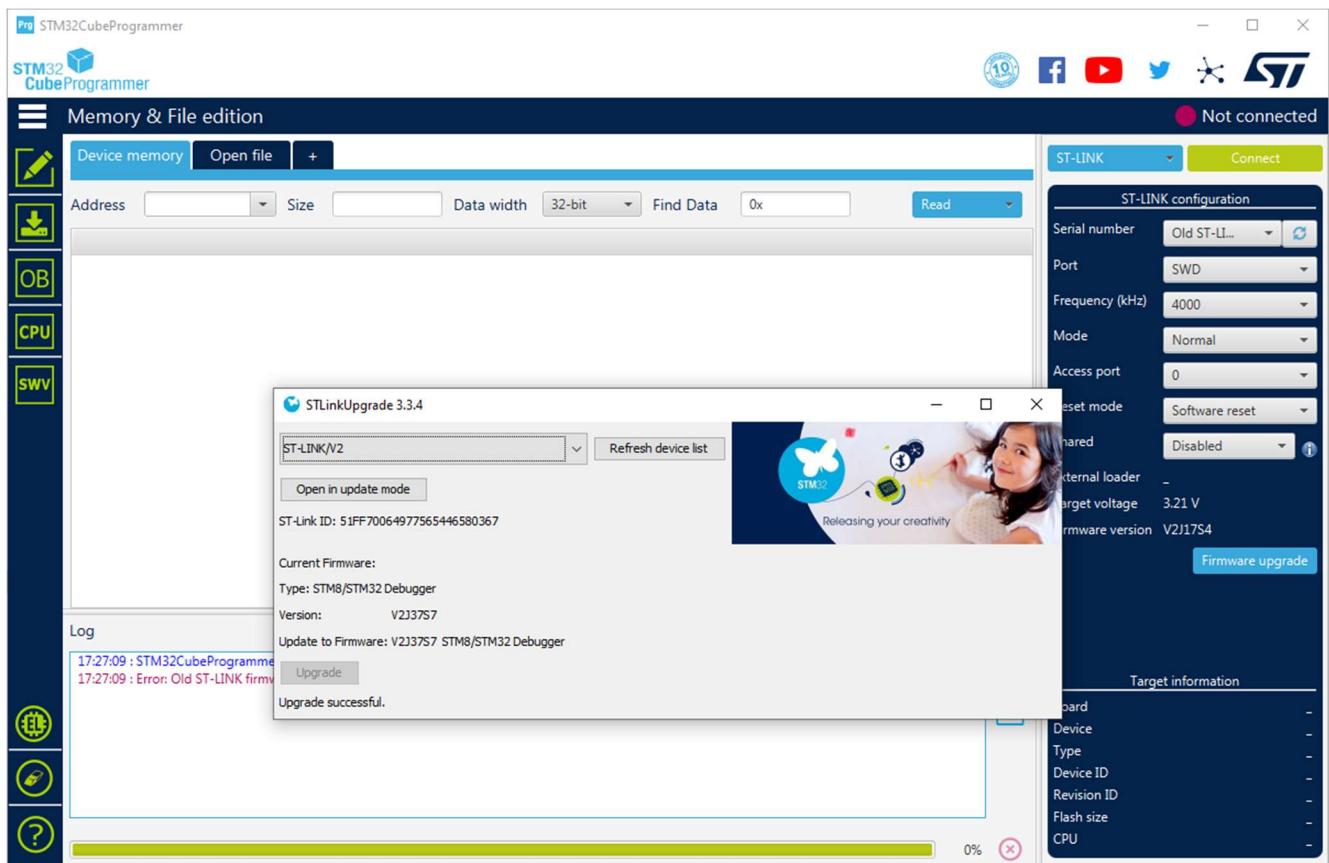
- Laptop with STM32CubeProgrammer installed,
- ST-LINK debugger/programmer,
- 4-pin male header,

Once you boot up the STM32CubeProgrammer utility you should see a similar screen as shown below. This is the main interface used for programming the STM32 microcontroller on the CR-6 SE hot end circuit board via the ST-LINK programmer.



Step 2 (optional) Updating firmware

If the programmer is new it might not have the most up-to-date firmware installed yet. Make sure to upgrade this first via the “Firmware upgrade” button.



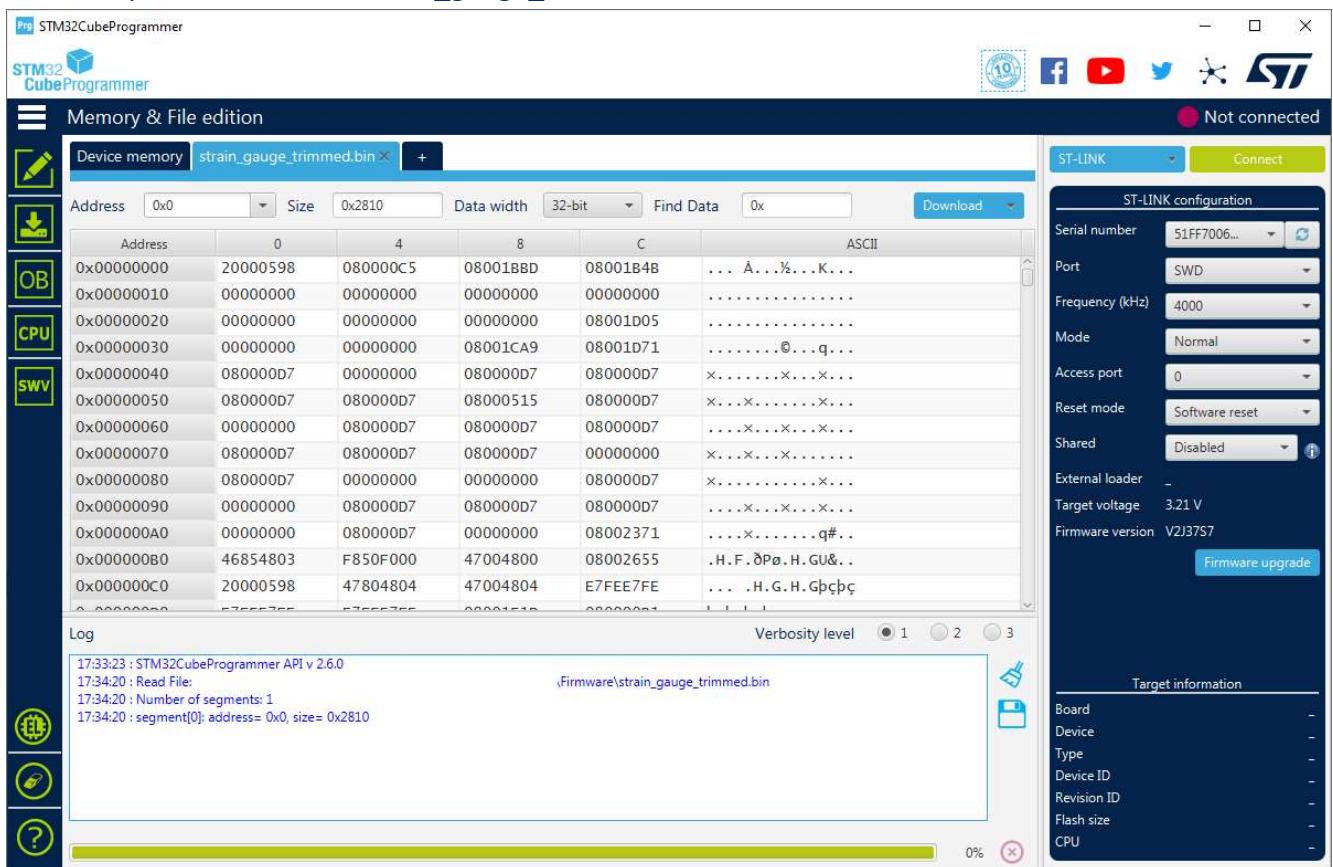
Step 3: Preparing the ST-LINK programmer (optional)

Once the firmware has been upgraded to the most recent version the ST-LINK can be wired to a male header and connected to the programming header on the circuit board. You can choose to either solder the male header to the circuit board in advance but be careful. The header will be too tall to fit in the fan shroud later and thus must be removed.

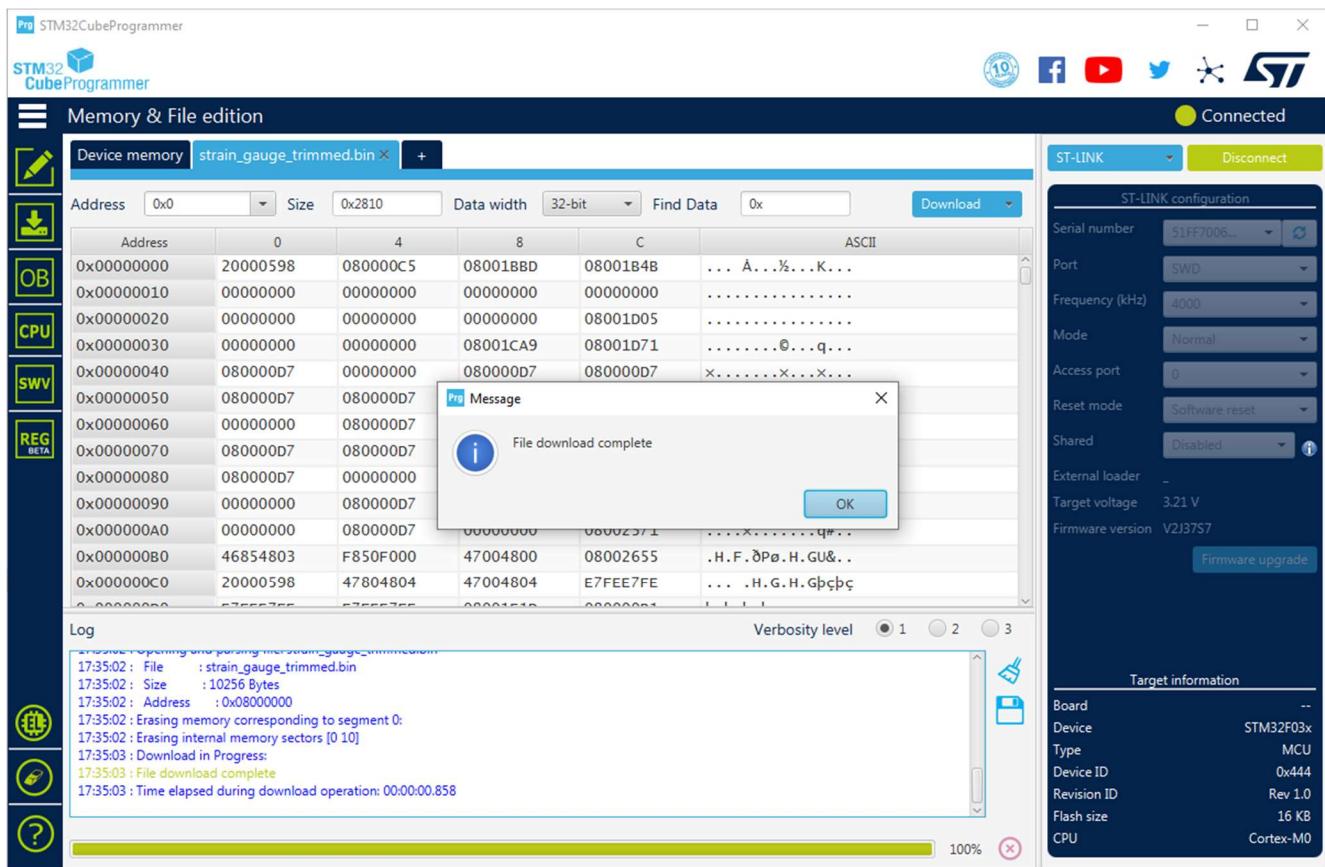


Step 4: Flashing

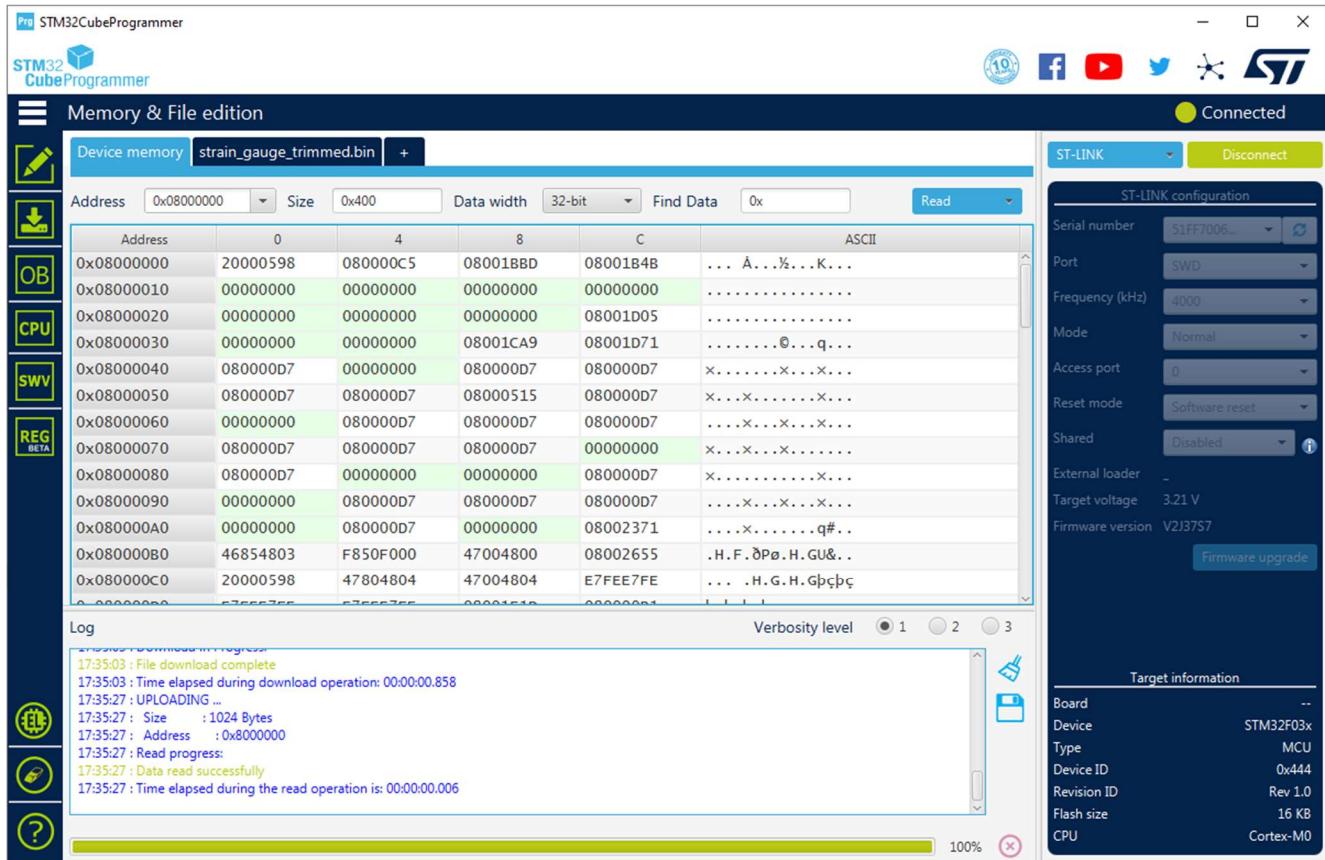
At the “Open file” tab the “strain_gauge_trimmed.bin” firmware file can be loaded,



Once the firmware is loaded, the STM32 microcontroller on the circuitboard can be programmed via the “Download” button.



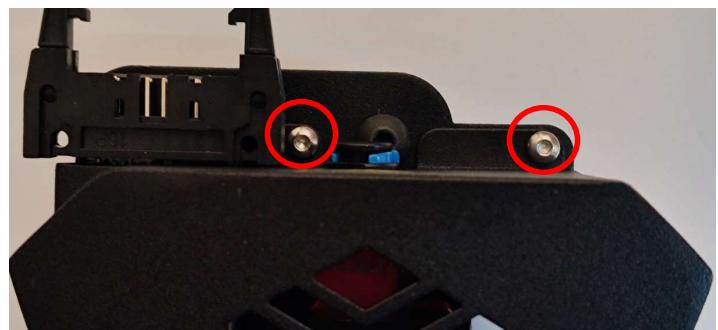
Once done, go to the “Device memory” tab and press “Read”. It should show the same data as the firmware you selected for download.



Installation

Step 1: Removing the fan shroud

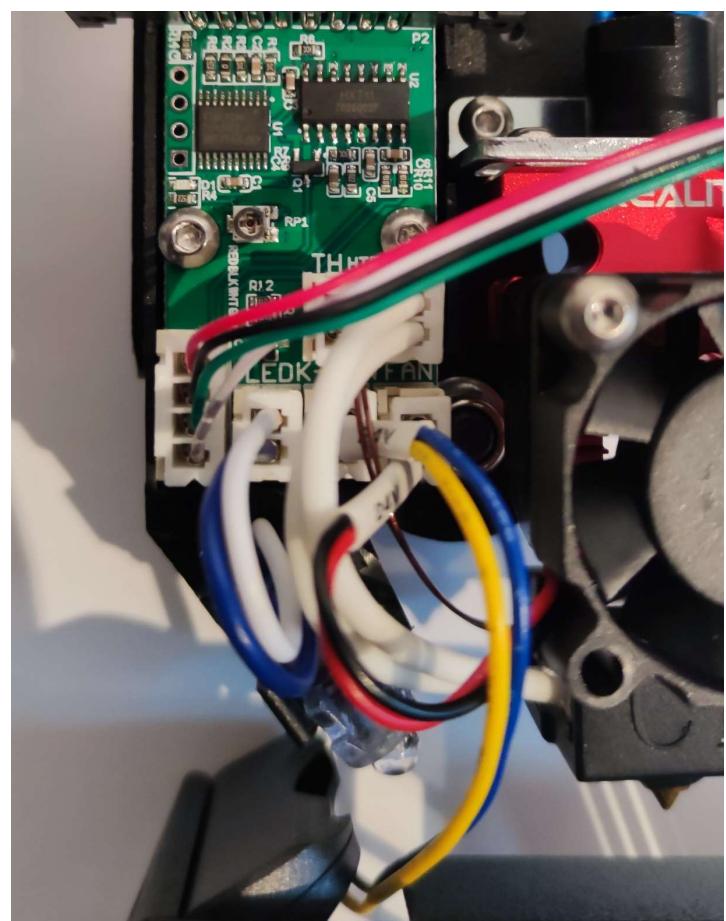
Start by removing the fan shroud of the hot end by removing the screws and disconnecting the ribbon cable.



Step 2: Disconnecting the peripheral connectors

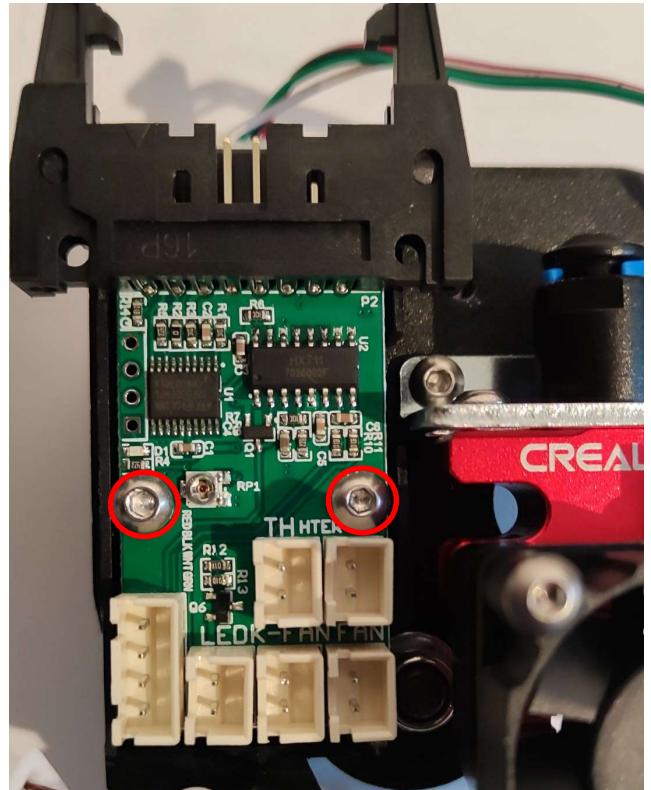
Before disconnecting the connectors, any hot glue should be removed.

Be extra careful when disconnecting the thermistor. The thin wires can easily be damaged.



Step 3: Removing the old circuit board

Remove the old circuit board by removing the two screws. These screws also connect the plastic tray that houses the LED.

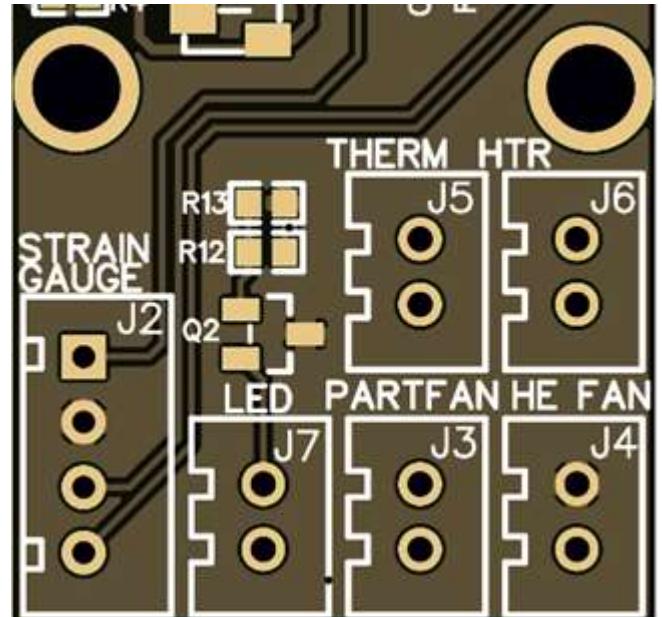


Step 4: Installing the new circuit board

Place the new circuit board in the tray and screw it to the backplate. hand-tight is tight enough.

Re-connect all the peripherals. In case it is unclear which connector goes where:

Peripheral	Connector
Strain gauge	4-pin connector
Thermistor	2-pin with thin wires
Heater	2-pin with insulated wires
LED	2-pin with blue/white wires
Part cooling fan	2-pin with blue/yellow wires
Hot end fan	2-pin with red/black wires



Finally, install the shroud cover and re-connect the ribbon cable.

Testing and calibrating

Once the installation of the new circuit board has been done it is time to test the print and calibrate the strain gauge.

Step 1: Testing

To validate the circuit board a test of each of its connected components is required.

Testing the LED.

On the touchscreen, turn the LED on and off.

Testing the fans

The hot end fan should already be turned on after turning on the printer. For the part fan a merlin command can be sent to the printer via Octoprint or Pronterface. The command being M106. After sending the command the part fan should start spinning at full speed. The part fan can be turned off again with the M107 command.

Testing the heater and thermistor

On the touchscreen, doing a preheating cycle for printing can be used for testing purposes. Please check if the hot end temperature shows as a “0” the thermistor before starting the print, if so, the thermistor is connected properly. When doing the preheating cycle the temperature should rise to its setpoint and stay at that temperature.

Testing the strain gauge

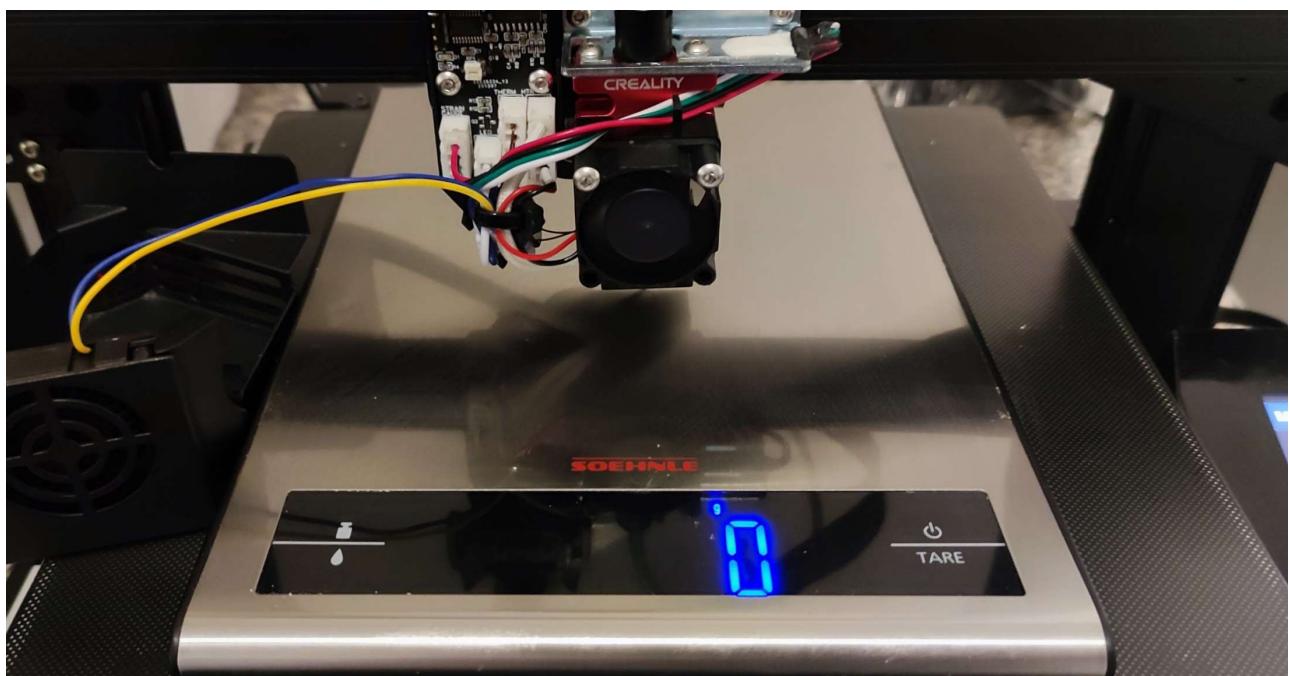
The strain gauge can be tested by pressing the heater block upward. **Make sure to do this at room temperature after the nozzle has cooled down.** The red indicator LED should start glowing after applying a little bit of pressure.

Step 2: Calibrating the strain gauge

It is possible to calibrate the strain gauge in case it is too sensitive or not sensitive enough. For this a kitchen scale is required.

Measuring

Place the kitchen scale on the build plate as shown below and tare it.



The home command can be used to test how much force it takes to trigger the strain gauge. The strain gauge should trigger at around 160 grams of force. When the home command is not accurate enough lower the Z-axis by slowly turning the leadscrews.

Adjusting

The strain gauge sensitivity can be adjusted by rotating the potentiometer



Clockwise: Reduce the sensitivity (more grams of force to trigger)

Counterclockwise: Increase the sensitivity (less grams of force to trigger)

The potentiometer is very sensitive, only increase or decrease by very small increments.