EEZ PSU H24005 Building Instructions (r5B12)

Work in progress

Building instructions for prototype version r5B9 can be found here or here.

If you need assistance please contact us using this form or post your problem on EEVblog forum here. Feel free to open a *New issue* in GitHub here (please check first is similar issue already exist).

1. PCB assembly

The PSU has four PCBs where SMT parts are used to the greatest extent. Only connectors and few specialized parts (e.g. power resistor, PCB mounted AC/DC adapter, post-regulator's power mosfet, etc.) are THT.
SMT parts are selected to be easily mounted with hand soldering and assistance of magnifying glass with light or low magnification microscope. In general, a magnification of x8 to x10 should be more then appropriate for this task. Almost all passive components are of 0805 size and never smaller then 0603. Selected IC packages are SOIC, TSOP, TSSOP and similar that have exposed pins (i.e. no QFN or BGA package are selected).
Only two ICs (IC1, IC4 on Power board) that has exposed power tab cannot be simply mounted with soldering iron and need hot air soldering station. That requires different skills but one can find many useful videos on the Internet with instructions how to do that efficiently at home *without* use of e.g. stencil and reflow owen.

1.1. Required tools

- Soldering iron with conical sloped tip example
- Hot air soldering station example
- Solder wire 0.25 mm example, and 0.7 mm example (optionally solder paste, for ICs with exposed power tabs but take into account that its shelf life is very limited even when refrigerated, therefore use small package, example)
- Solder wick / desoldering braid example
- Flux example
- Magnifying glass with light (desktop magnifier with backlight) example or microscope example
- Self-locking tweezers example
- Set of tweezers example1, example2
- Isopropyl alcohol example and paper wipes example for cleaning
- PCB holder example

1.2. Test and measurement equipment

The basic tool used for measurement is an oscilloscope. But, if assembling is performed carefully, that no single mistake has been done, (e.g. wrong part value or place, etc.) a simple DMM will be enough to check basic functionality.
During testing and taking measurements please take into account that channel's negative output (OUT-) is NOT on ground potential. Therefore if you are using multichannel oscilloscope without isolated channels (that is default!) you cannot concurrently connect test probe ground of one channel to the PGND and another one to OUT-. That will interfere with normal operation of current control loop (IC6A, IC7) because the current sense resistor (R63 or R65) will be shorted in that way.

1.3. Where to start?

The total number of parts that have to be soldered is almost 800. On the first sight that can easily discourage many, but it's not so bad. First, two most demanding PCBs for the Power boards are identical and you can try to assemble it side by side following steps mentioned below. The AUX PS is the simplest one but also has AC mains section that require additional care when operating. Finally, physically

the biggest one – Arduino Shield is modestly populated but also carry extra parts such as TFT touch-screen display, encoder, Arduino board, binding posts, etc. that dictate some other set of assembling rules.
A good start could be to check that all parts from the consolidated BOM are available and sorted by type and values and can be easily accessible. As you have probably already learned, simple SMT parts with 2-3 terminals (passives, diode, transistors) can be easily lost even if you have well arranged and clean benchtop. Therefore instead of crying for lost one, simply order few parts more and take another one when previously selected was just gone. That issue is present with both self-locking and regular tweezers.
There is a few methods of storing and sorting SMT parts like small part snapboxes (example), SMT storage books (example), etc. Each of them is valuable as far as it can reduce possibility of replacing one part mistakenly with another. That is especially important for ceramic capacitors (MLCCs) that do not carry value marks.
A discipline of selecting a single value at the time (even in larger quantity) and placing them on proper places is of a paramount importance to ensure that PCBs are assembled correctly and will work properly.
When needed tools, bare PCBs and SMT parts are on disposal, one important task still remain before we can start with PCB assembling – it's how to identify part value since PCB's silkscreens (top and bottom) only carry reference designators (i.e. R..., C..., IC..., etc.).
If you are using Eagle then open **.brd** file and when populating top layer make sure that among other layers 21, 25, and 27 are visible (see *Top assembly selected layers.png*). For assembling bottom layer set layers 22, 26, and 28 as visible (see *Bottom assembly selected layers.png*).
The freeware edition of Eagle also allows you to open .brd file and switch layers on and off.
When installation of the Eagle is not an option, you can use the following images:

- AUX PS r5B12a assembly (top layer).png
- AUX PS r5B12a assembly (bottom layer).png
- Arduino shield r5B12 assembly (top layer).png
- Arduino shield r5B12 assembly (bottom layer).png
- Power board r5B12 assembly (top layer).png
- Power board r5B12 assembly (bottom layer).png

- Above mentioned order of PCBs is not mandatory for assembling process but represents a logical sequence since AUX PS is required for powering Arduino shield and Arduino shield is required for controlling Power boards. Suggested order is also sorted by PCB complexity starting with the simplest one. If not otherwise specified, we'd like to recommend that parts are soldered in the following order on the *same* layer:

- SMT IC that require hot air soldering (i.e. IC1, IC4)
- Small SMT parts (e.g. passives, diodes, transistors, etc.)
- SMT ICs
- Bulk SMT parts (elco capacitors, power inductors, mosfets and diodes) and
- THT parts (connectors, switches, chokes, etc.)

1.4. Default jumper positions

For various reason on few places on the PCBs a jumpers section are used where zero-ohm resistor is used to define signal path. Some of them define behavior of the circuits (e.g. JP5, JP6, and JP7) and when wired in wrong way or left unpopulated that can create some difficulties and the firmware could not control it properly. To simplify a whole assembly process a clear mark is added next to zero-ohm resistor default position that should insure a proper functionality with current firmware revision. On the picture below above mentioned jumper sections are shown and red rectangle indicate the place for zero-ohm resistor that is in line with “U” shaped mark.

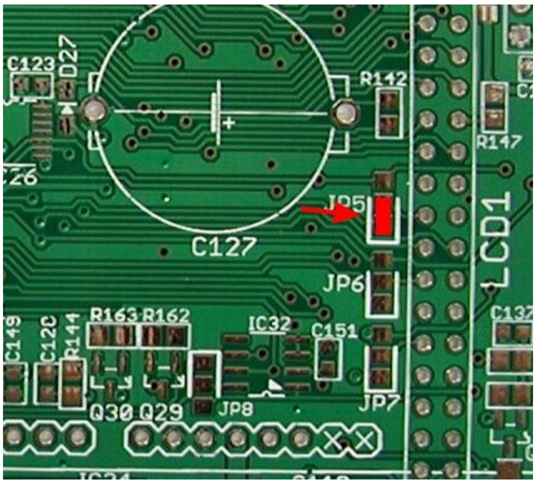

Fig. 1: Zero-ohm jumpers markings

1.5. AUX PS r5B12a assembling

Schematic for this board is shown on sheets 6/12 and 7/12. We'd like to recommend the following steps for assembling this board:

- Fan control,
- AC input terminal and protection,
- Soft start/stand-by (Q20, Q21, OK1, OK2),
- Power relay (K_DOU2),
- Dual output AC/DC module (TR1) and
- THT connectors (X5, X6, X7, X8, X10, X11)

Testing of fan control section has to be postponed until Arduino Shield is assembled. Also, it will require preparation of fan cable which comes without connector.
AC input terminal and protection section contains only THT parts that can be carefully soldered. Soft start/stand-by triacs Q21 and Q22 are sharing the same heatsink KK1 mounted on its opposite sides using single screw and nut (Fig. 2). They can be mounted on before soldering. Another possibility is to solder heatsink first and then mount on each side a triac and then solder their terminals. Take care that power resistor R103 is of proper type (wirewound), voltage and power rating.
The X5 connector has to be mounted *after* surrounding SMT parts (R122, OK3, ZD6, Q25) otherwise it could be difficult to mount them without damaging X5 by touching it unintentionally with soldering iron body or tip.

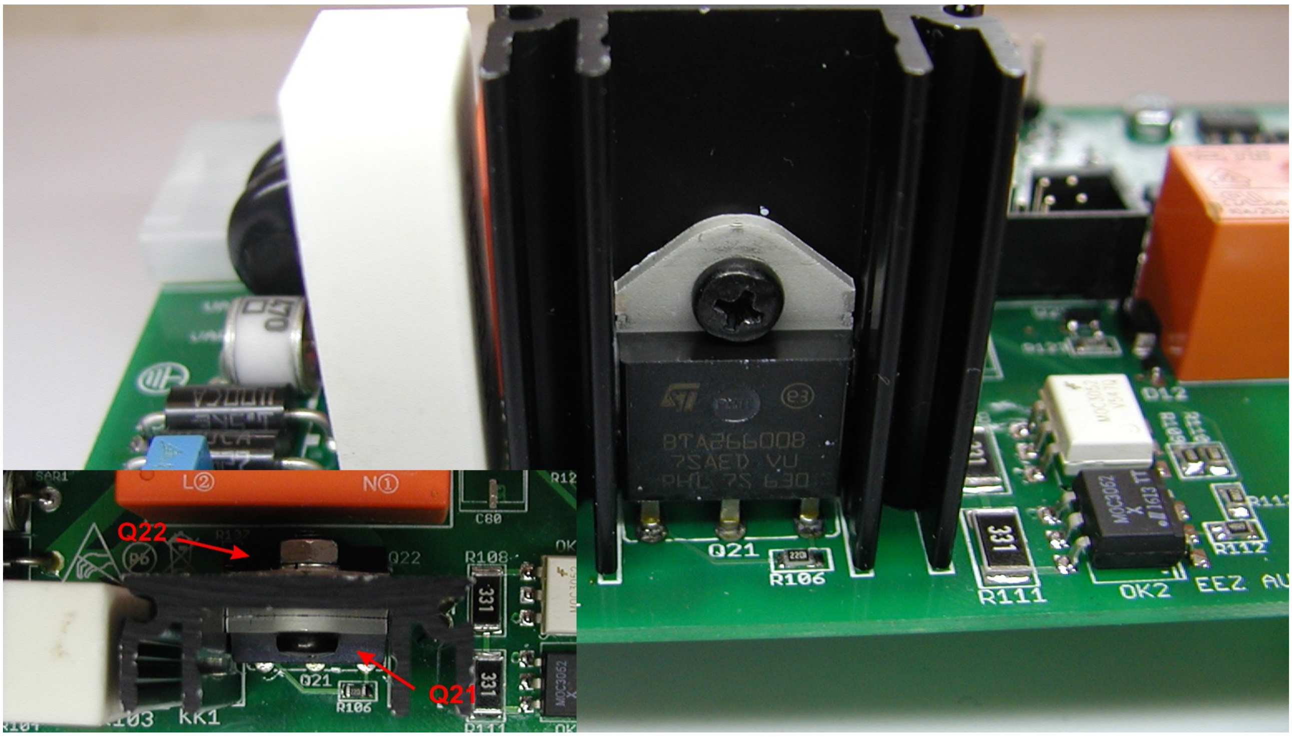

Fig. 2: Q21 and Q22 shares the same heatsink (KK1)

Finally, place TR1 as the latest part on the top PCB layer. Now it's possible to test once again +5 V power supply by carefully applied AC mains on the X4 pin 1, 2 and 5.

On the bottom layer only three parts have to be mounted – DOUT2 (X6), Ethernet (X7) and USB (X11) connectors. If you got customized enclosure with pre-drilled holes on its rear panel, you can mount the PCB on the rear panel first (using 14 mm spacers), and then solder above mentioned connectors. That will insure that everything fits perfectly.

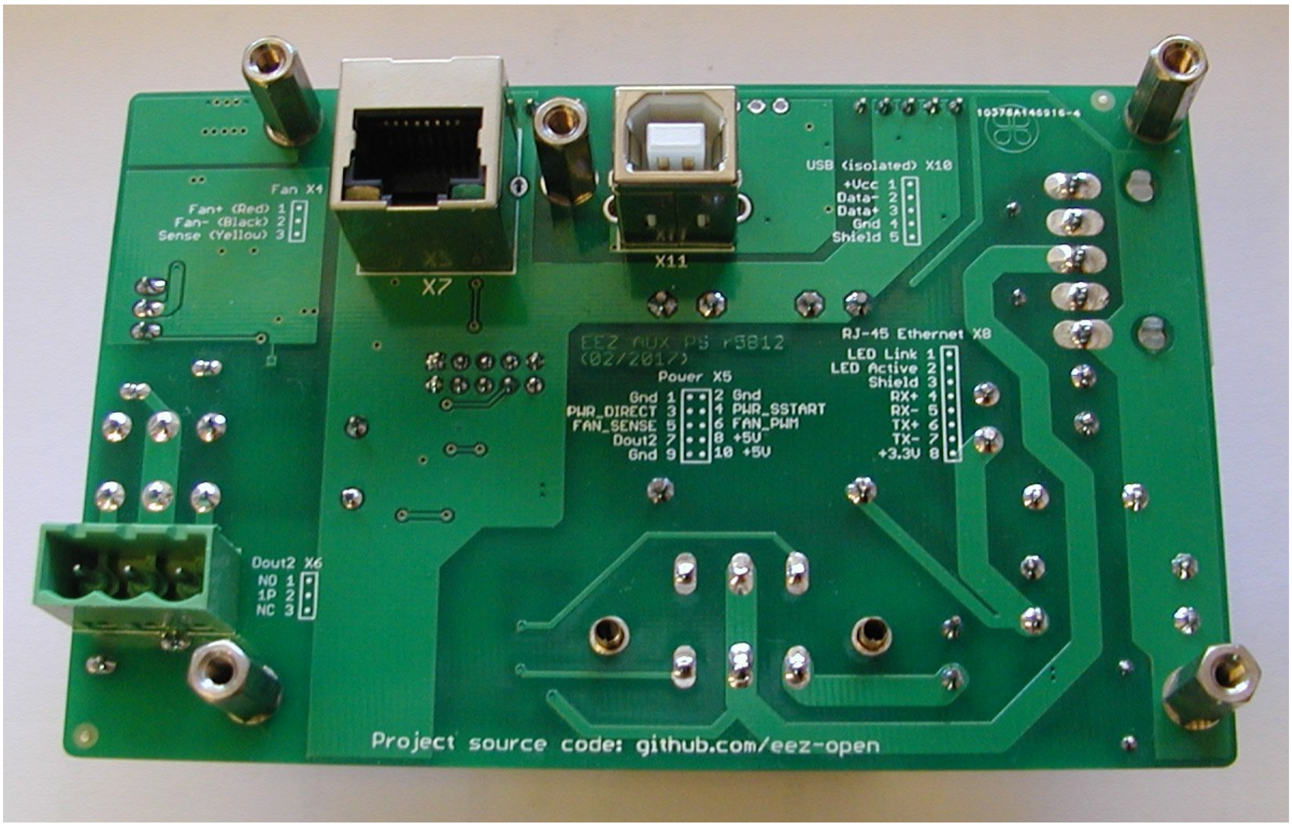

Fig. 3: Bottom view (mounted THT connectors and spacers)

1.6. Arduino Shield r5B12 assembling

There is no recommended order in which sections on this PCB should be soldered. One possibility is to simply follow the order of sheets in consolidated schematics related to this board (sheet 8/12, 9/12, 10/12, 11/12, and optionally 12/12) and to leave the following parts for the end:

- Power relays (K_SER, K_PAR) due to their dimensions
- Super capacitor (C112) for RTC backup
- Headers for Push-in connectors (X12, X14)
- 40-pin connector (LCD1) for the TFT touch-screen display
- Front panel power switch (SW1) and incremental encoder (SW3)

Push-in connectors distance adjustments

Distance of approx. 18 mm between rear side of the front panel and the Arduino Shield PCB is defined by TFT display height (or depth) when it's fixed on display mounting rail on one side and plugged in to 40-pin (LCD1) on the Arduino shield.

Height of the push-in connectors (X12, X14) is about 13.5 mm and when they are soldered directly on the PCB they will be 4.5 mm apart from the front panel surface. That does not mean that their pins are not accessible. It's more a question of visual appearance and to overcome that a pair of low profile (5 mm) 8 pin sockets (example) are used for anchoring that connectors.

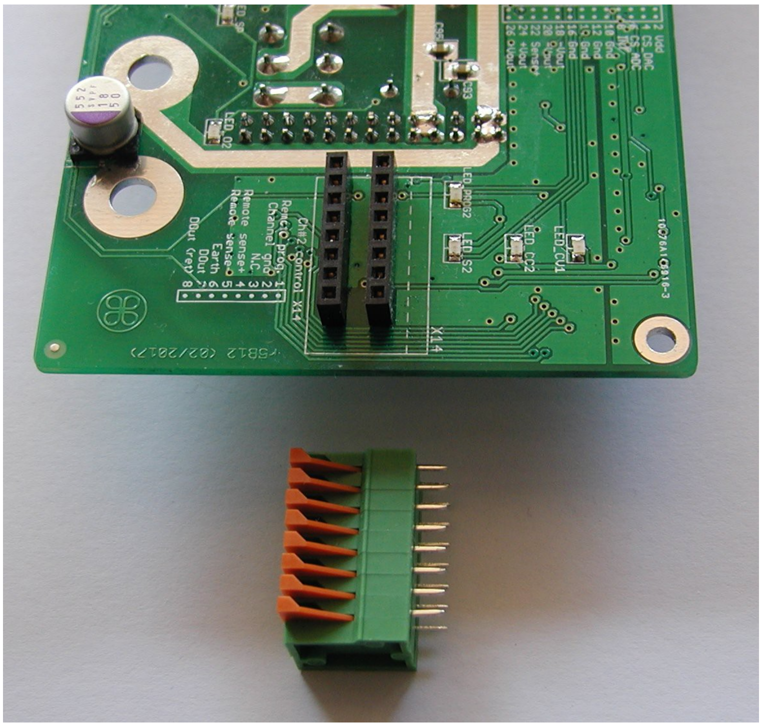

Fig. 4: Low profile sockets for push-in connectors

When all SMT parts are mounted, followed by THT parts, we can solder few parts mentioned on the beginning:

- The power relay now should not interfere with mounting of any surrounding parts.
- The super capacitor is polarized as elco capacitors or battery cell. Check its terminals polarity twice before mounting! (see Fig. 5)
- If push-in connectors X12 and X14 is not inserted into sockets, as it's suggested above, then we can now proceed with their soldering on the bottom side of the PCB.
- TFT touch-screen display 40-pin connector (LCD1) has to be soldered on the bottom side too. If we want to be completely sure that it's aligned with display, we can mount display on the display mounting rail and enclosure first, then insert connector on display's header, and finally place the PCB on top of display. We have to check alignment of the PCB with output binding posts holes and when everything is in place 40-pin connector can be soldered.
- Finally, we can mount power switch (see Fig. 6). Once again, it's recommended to see how it's aligned with the corresponding hole on the enclosure's front panel first, then adjust its position

and start with soldering.

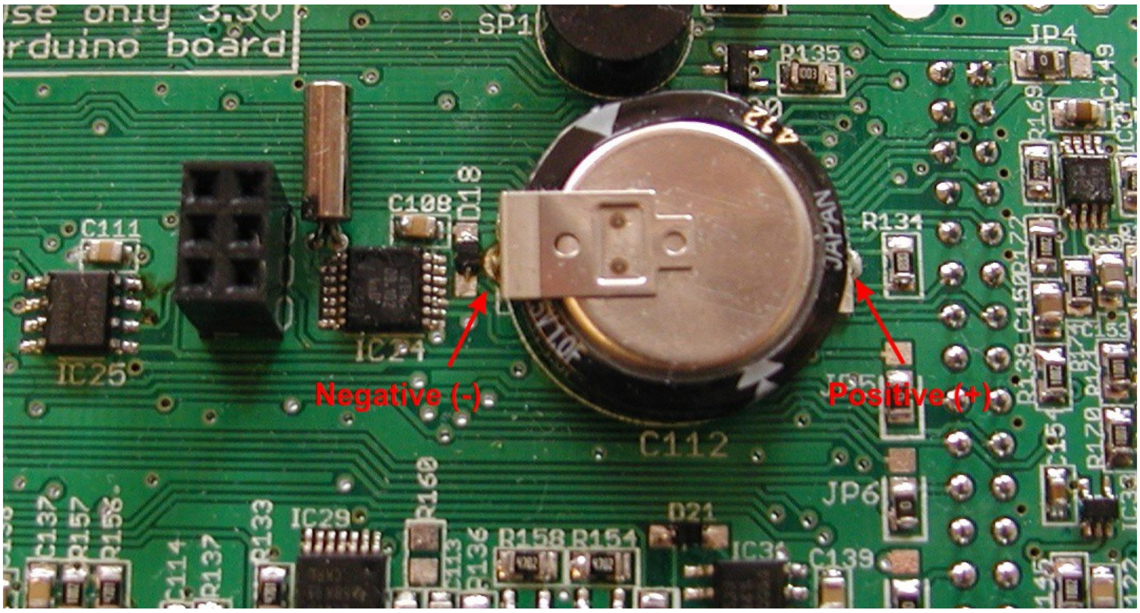

Fig. 5: Supercap for RTC backup placement

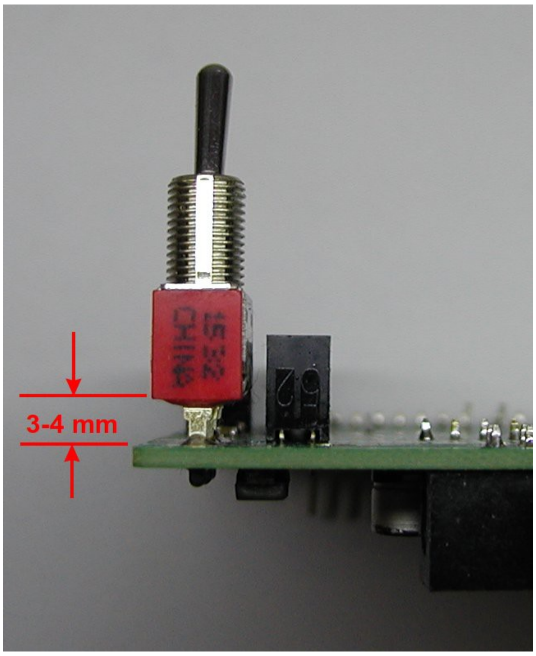

Fig. 6: Power switch mounting

1.7. Power board r5B12 assembling

According to the number of components and functionality, the Power board is the most complex part of the PSU. There is at least two possible assembling scenarios: be disciplined and carefully solder all parts, and simply plug it into Arduino shield and check if it will pass self-test procedure. It does not require any special adjustment and even when is not calibrated it should provide with good accuracy programmed output voltage and current. Second scenario could be to assemble it in at least three steps:

- Power pre-regulator,
- Bias power supply and
- Post-regulator

After each section is completed, perform simple testing described below before continuing with the assembling. In this way you can limit possible damage if one section is mistakenly assembled (especially if bias power supply output voltages are wrong!).

WARNING: Never apply DC input (+48 V) when AC/DC module is powered on (e.g. connecting cable manually to the X1 connector). That could generate spike (due to L1) up to 100 V high that can easily destroy on-board SMPS controller (LTC3864) even with ZD1 and C3 in place.

Power pre-regulator

The power pre-regulator circuit is shown on Sheet 1/12. We recommend you to start with IC1 that require soldering of exposed thermal pad. A hot air soldering station instead of soldering iron should be used here. This section also require soldering on the PCB's bottom side (L1 and X1). That should not be a problem if you are using PCB holder. If you choose to supply Power board with AC instead of DC input, follow Hack #1.
Once again, keep in mind to mount all smaller parts first and than continue with bulky one such as L2, C3, C12, C15, etc.
When everything is in place, you can proceed with basic testing. Since the board is not fully assembled, it cannot be done without some tricks. The first possible obstacle to check if pre-regulator works properly is existence of Q3 that is used to enter 100% duty cycle when the pre-regulator is bypassed. The IC1 is then effectively switched off and instead of switching frequency a DC signal will be present on Q1 gate (hence the name 100% duty cycle). But, we'd like to test in this step if switching works or not. We have two possibilities here:

- To apply +5 V (use some external source in this stage) to Q3 gate by soldering a thin wire e.g. used for wire wrap to that position or
- simply place zero-ohm resistor on position R14. *Note: don't forget to remove it before start testing the Post-regulator section and IC8 is mounted.*

Before we can apply input power we need to provide some signal on trace *OUT+*. The easiest way is to locate any of pins 23 to 26 on X2 connector.
Again, an external regulated source is needed. We can use the same +5 V used to turn on Q3 or something else with output of up to +40 V. In fact, we can start testing by simply connecting *OUT+* to the ground. If *OUT+* is grounded we can check with the scope two points to see if pre-regulator works:

- First is "hot-spot" (SW, magenta trace) where Q3 drain, power inductor L2 and D1 cathode are connected. Use x10 probe.
- Another test point is *PREG_OUT* (Preg, cyan trace), output from the power pre-regulator.

Fig. 7 shows measurement when *+OUT* is set to 0 V (grounded). Its output voltage will vary depending on connected load and difference between post-regulator output, and power pre-regulator output. Mentioned voltage difference could be about +6.5 V for no load to less then +3 V for full load of 5 A. Fig. 8 is another example without load when +5 V is used for pre-regulator's output voltage programming (measured *PREG_OUT* is +10.8 V).

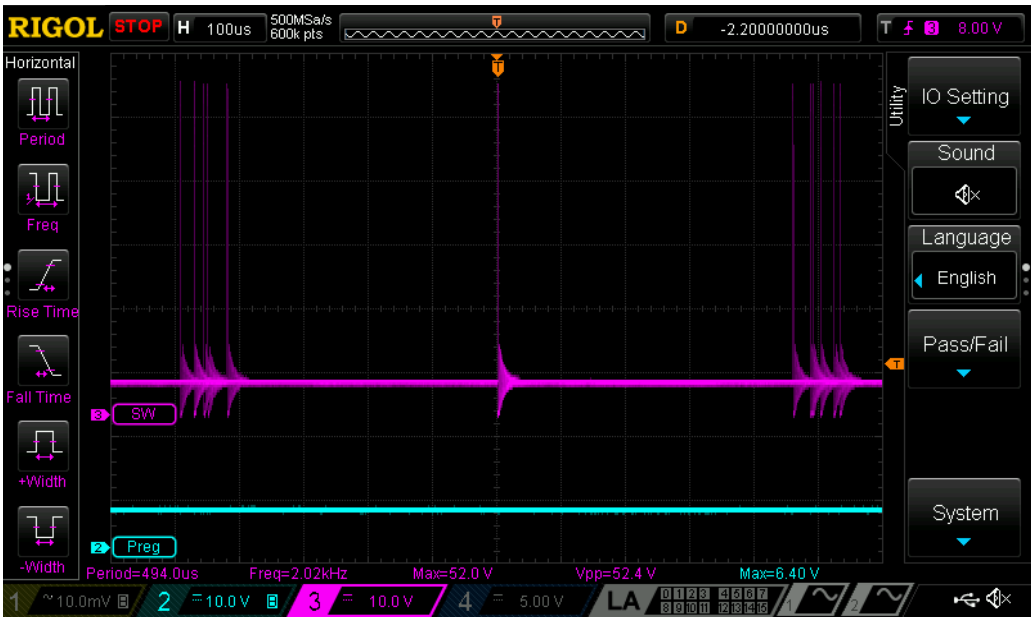

Fig. 7: Pre-regulator measurements, +VOUT = 0 V (grounded, no load)

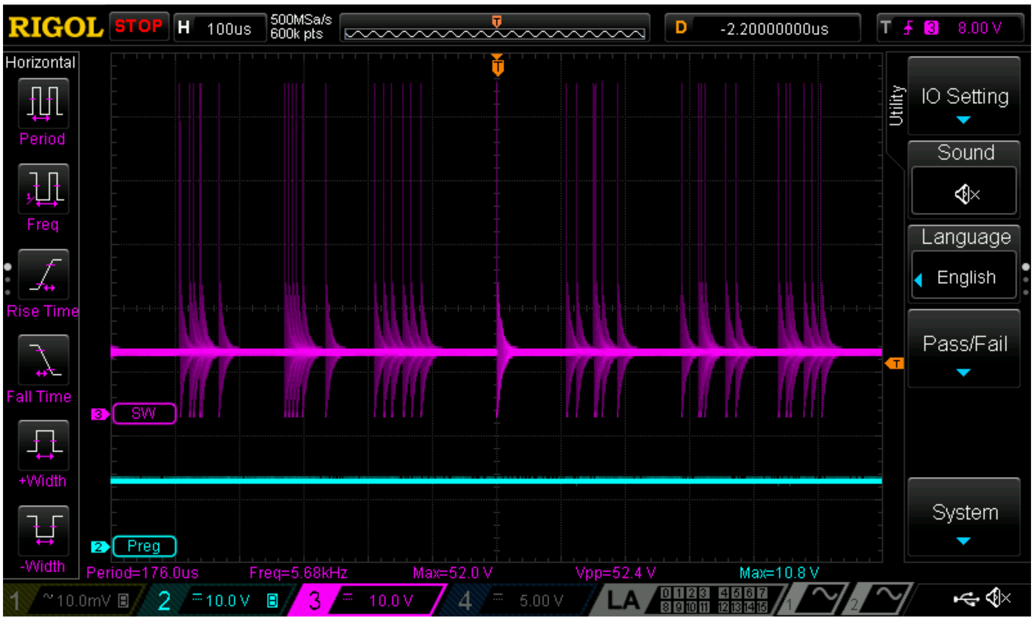

Fig. 8: Pre-regulator measurements, +VOUT = 5 V, no load is connected

Please note that in both cases the SW signal looks erratic but that is only because no load is connected. On Fig. 9 is shown characteristic switching pattern when load is connected (e.g. power resistor of 10 Ω).

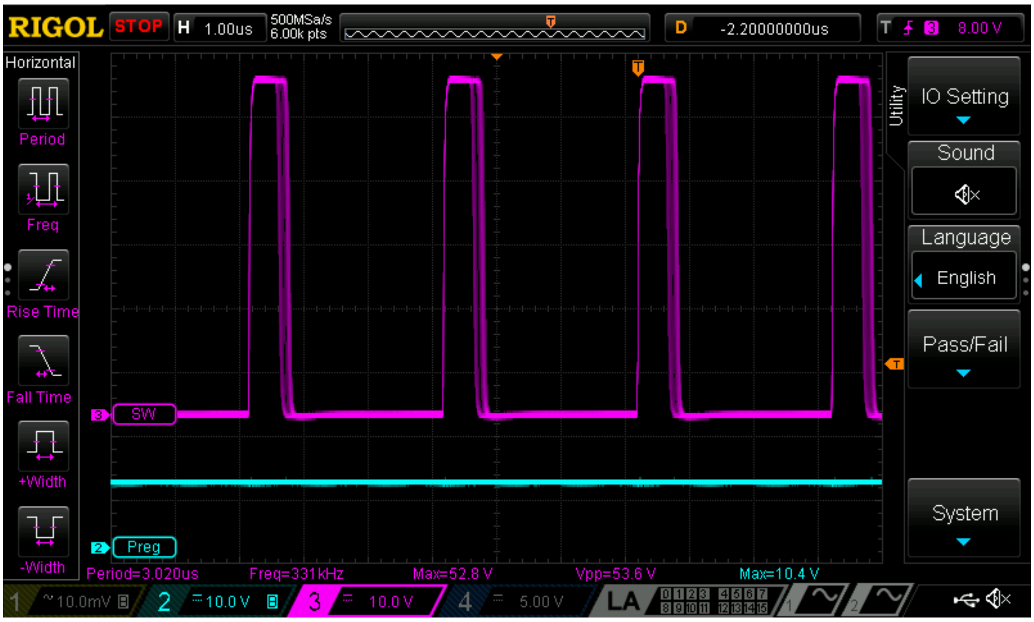

Fig. 9: Pre-regulator measurements, +VOUT = 5 V with load connected

Finally on Fig. 10 is shown what will happen if Q3 is not turned on. The IC1 will enter 100% duty cycle because chosen values for voltage divider in feedback loop (R13, R16) are set well over pre-regulator input voltage to +56 V while input voltage is not higher then +50 V.

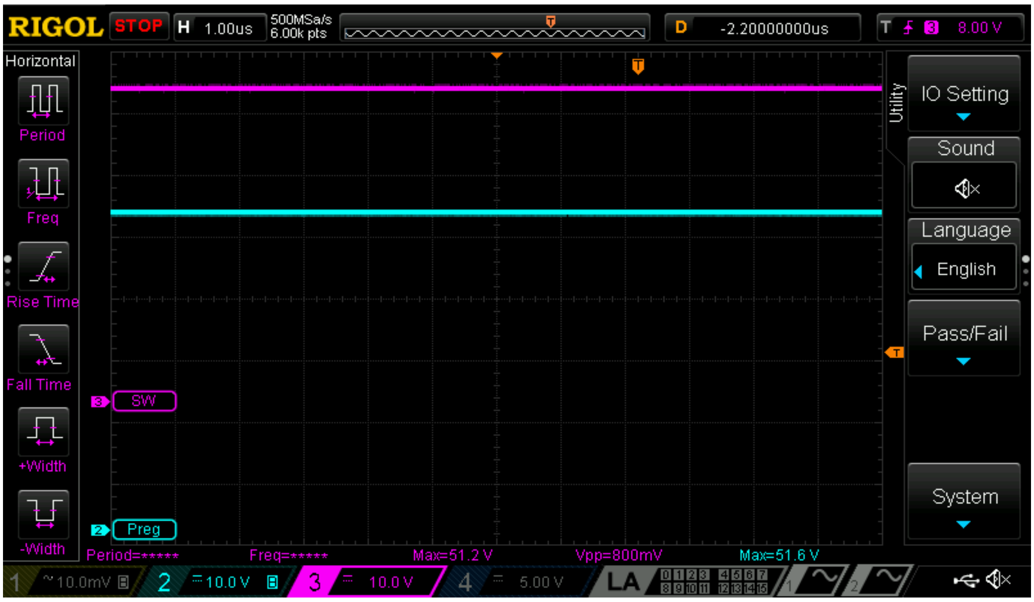

Fig. 10: Pre-regulator in 100% duty cycle, without load connected

The so-called Low noise mode of operation when pre-regulator works with 100% duty cycle will remove from channel's output hard to filter switching noise. But working in that mode puts real limitation in output power and if the PSU will be used for application where higher output ripple can be easily tolerated it's possible to disable that option by removing Q3 and mount permanently zero-ohm resistor on position R14.

Bias power supply

The sheet 2/12 has to be followed for bias power supply circuit assembling. Again, like in case of power pre-regulator start with IC4 that has exposed thermal pad and require hot air soldering station.

When all other parts are in place we can proceed with basic testing. Please note that power pre-regulator will be now also powered and make sure that no load remains connected on the *PREG_OUT* from the previous testing. Obviously, the first thing that we can test are output voltages on points *+V* and *-V* (IC4 outputs) and *+5V* (IC2 output). They should to be within 5 % tolerance +5 V, -5 V and +5 V.

Another point of interest is *PWRGOOD* (IC2). It's interesting to see how the *PWRGOOD* relates to change of input voltage. That signal will be actively monitored with MCU and it will push the PSU into Stand-by mode as soon as possible when its changed from high to low level. Fig. 11 and Fig. 12 shows transition while input power is applied and removed.

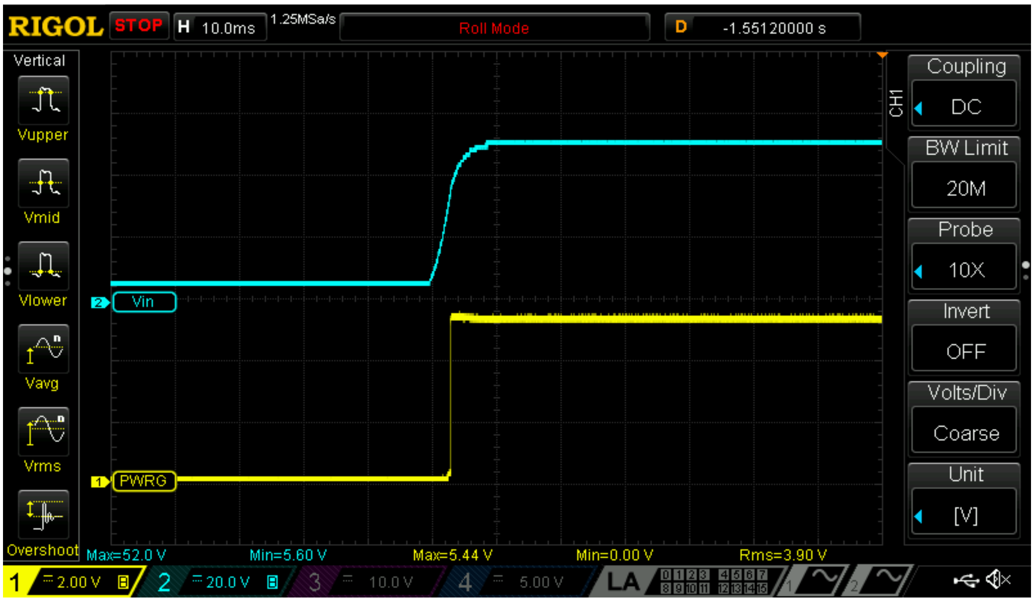

Fig. 11: PWRGOOD signal transition (Power on)

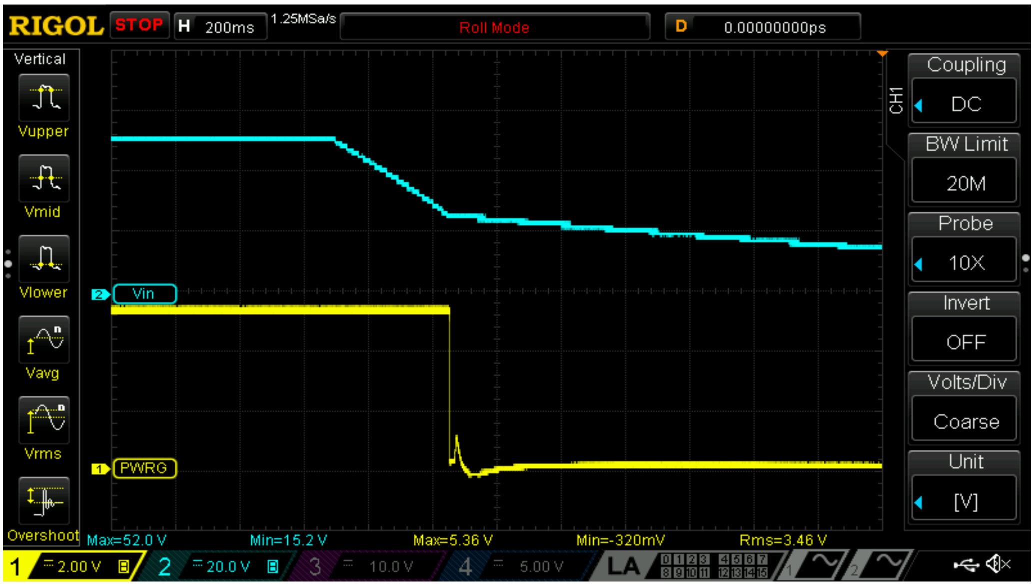

Fig. 12: PWRGOOD signal transition (Power off)

Post-regulator

The post-regulator circuits can be found on Sheets 3/12, 4/12 and 5/12. The power mosfet (Q4) require special attention. It's THT part and before soldering we have to bend its terminals as is shown on Fig. 13.

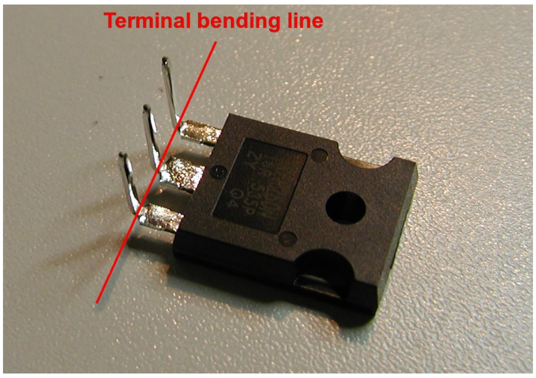

Fig. 13: Q4 preparation for mounting

The Q4 has to be mounted on PCB’s bottom layer, and if EEZ H24005 customized enclosure is used, please ensure that distance between its surface area and PCB bottom side is 5 mm. The easiest way to accomplish that is to mount PCB on the heatsink (Fig. 14) using spacers, then mount Q4 (do not forget insulation!), fix it with M3 screw and then start with soldering.

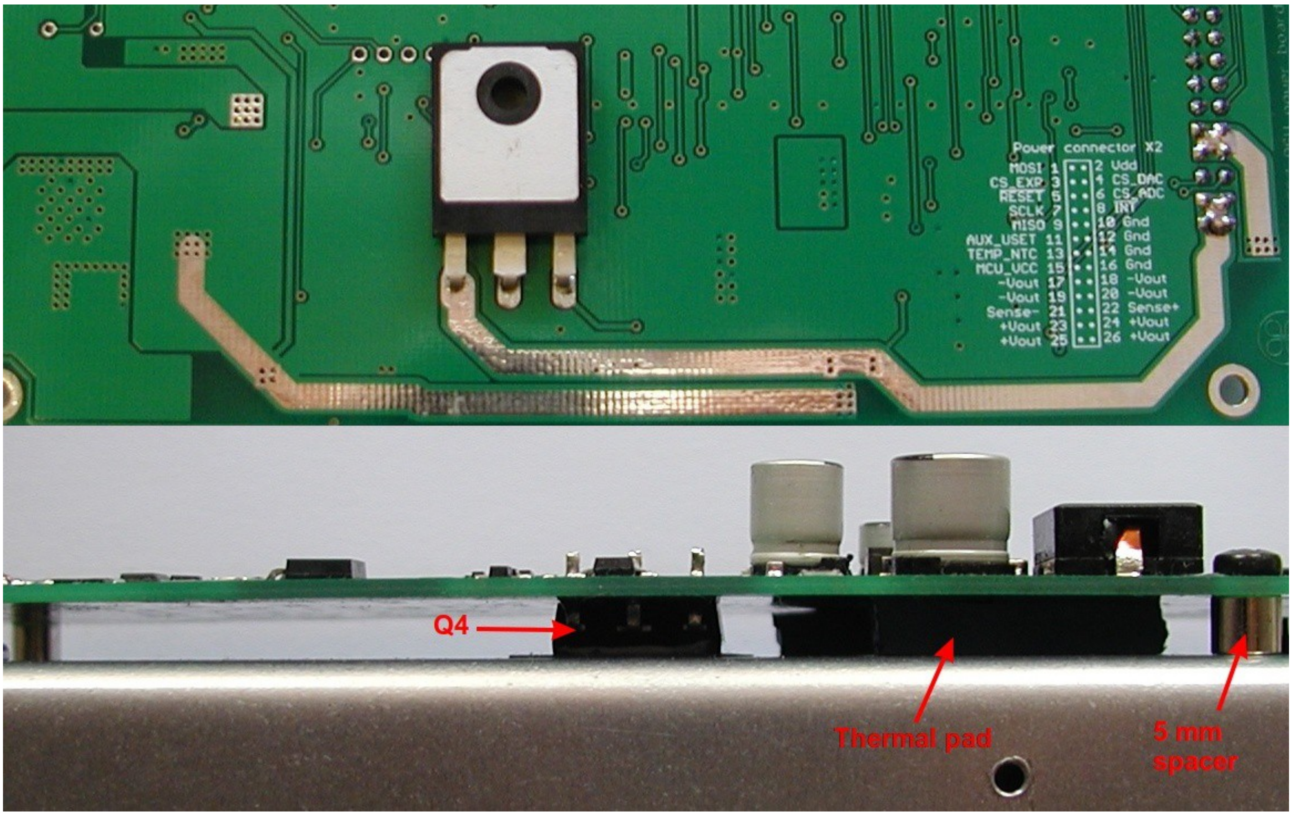

Fig. 14: PCB side view with Q4 mounted on heatsink

It is highly recommended that below power pre-regulator area (IC1, Q1, D1) and high voltage LDO (IC3) a piece of 5 mm thick silicone thermal pad is mounted (search e.g. AliExpress) as shown on Fig. 15). Channel temperature that is measured with NTC1 can be lowered in that way significantly. Lowering temperature will also affect cooling fan operation which will work with lower speed and makes it quieter.

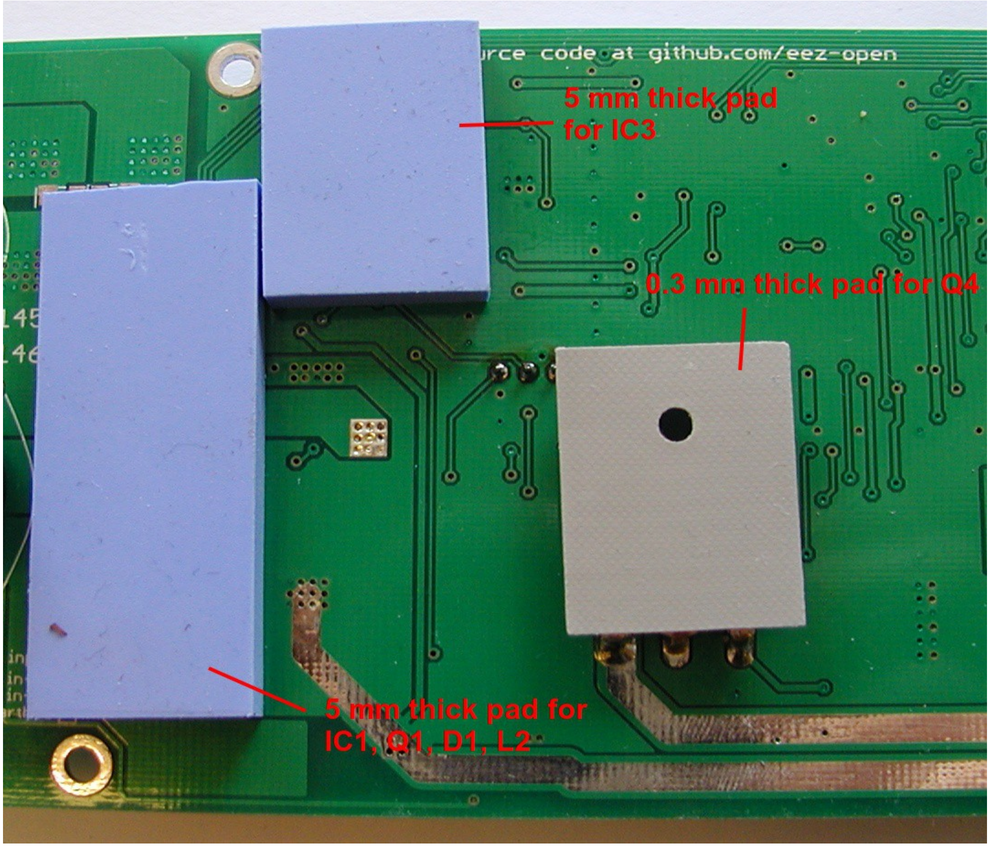

Fig. 15: Silicone thermal pads

Do not use e.g. aluminium block (with PCB side insulated) as an alternative to silicone thermal pad since it could increase EMI.

When everything is mounted, you can check few points before connecting it with the Arduino Shield that is required for controlling the Power board:

Signal	**Value**	**Position**	**Comment**
+VREF	*+2.5 V*	*IC9*	Define overall precision
+OUT	*around zero*	*Pin 23-26, X2*	
CV_ACTIVE	+5.1 V (logical high)	R84/R88	CV mode active (if U_SERVO is negative)
CC_ACTIVE	+0.6 V (logical low)	R97/R99	
POST_OE	+4.67 V (logical high)	R55	Output is disabled
U_SERVO	-1.1 V	D7 cathode	
I_SERVO	+0.28 V	D9 cathode	

2. Enclosure and wire harness

The next step when all PCB modules are assembled and tested should be wiring and placing it into enclosure. The customized metal enclosure manufactured by Varisom has the following elements (Fig. 16):

- Top and bottom plate with ventilation holes
- Front panel for TFT touch-screen display, binding posts, push-in connectors, light pipes (example) for LEDs, etc.
- Rear panel for AC inlet, DOUT2, USB, Ethernet, cooling fan and AUX PS module mounting holes
- Two set of three U-shaped aluminum heatsink with Power module mounting holes
- Mounting rail for AC/DC power modules and
- Mounting rail for TFT touch-screen display

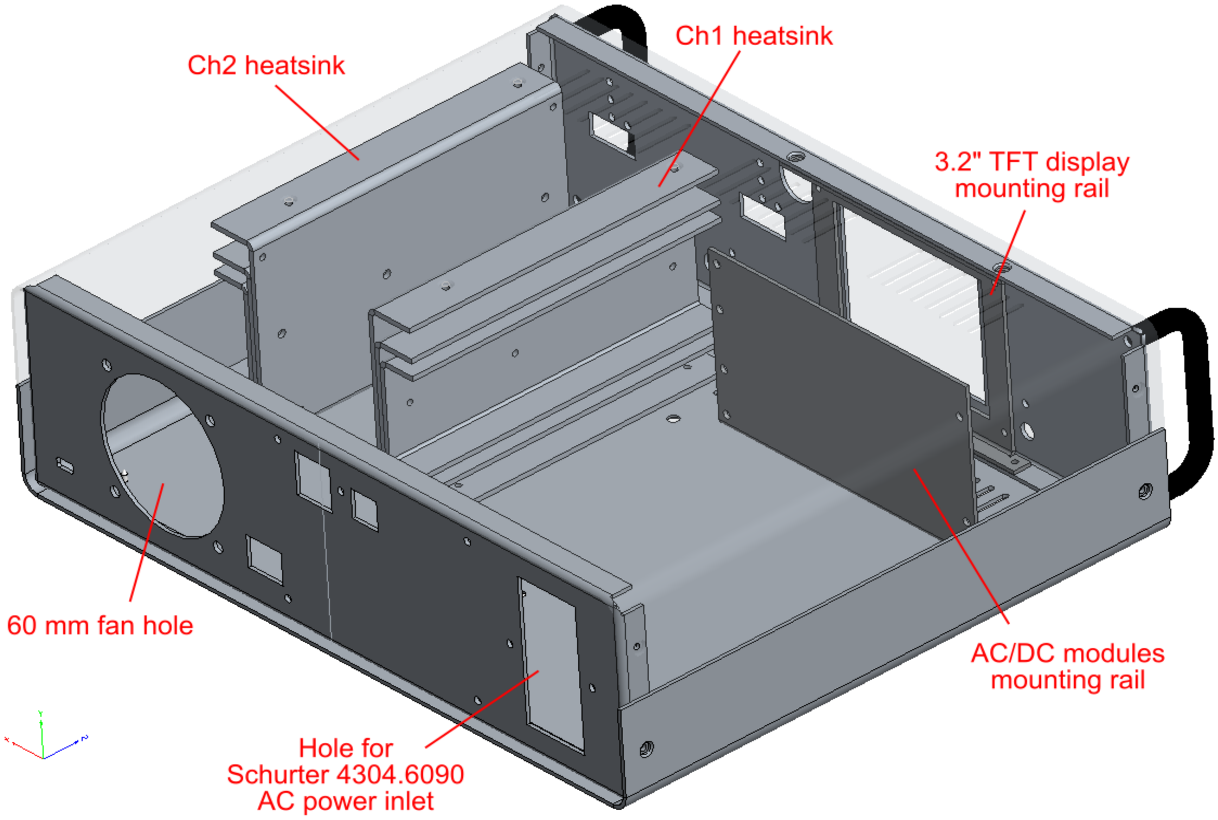

Fig. 16: Customized metal enclosure (Varisom)

The wire harness specification is available in separate section (link) that also include BOM. That specification is just an example how wire harness can be done, and one is free to use wires of different type and length but with equal or bigger diameter and voltage rating for power paths.

Two cables, one for USB connection between Arduino board and AUX PS module, and another IDC 10-pin flat cable for powering Arduino shield are possible to get as already assembled. All other cables has to be cut to the size and equip with appropriate connectors. Genuine crimping and wire cutting tools are expensive and if you already don't have them a good alternative could be the following:

- Jokari No. 20050
- Engineer Crimper with No.12 and No.13

Fig. 17 shows block diagram that could help to get an idea how wiring looks like.

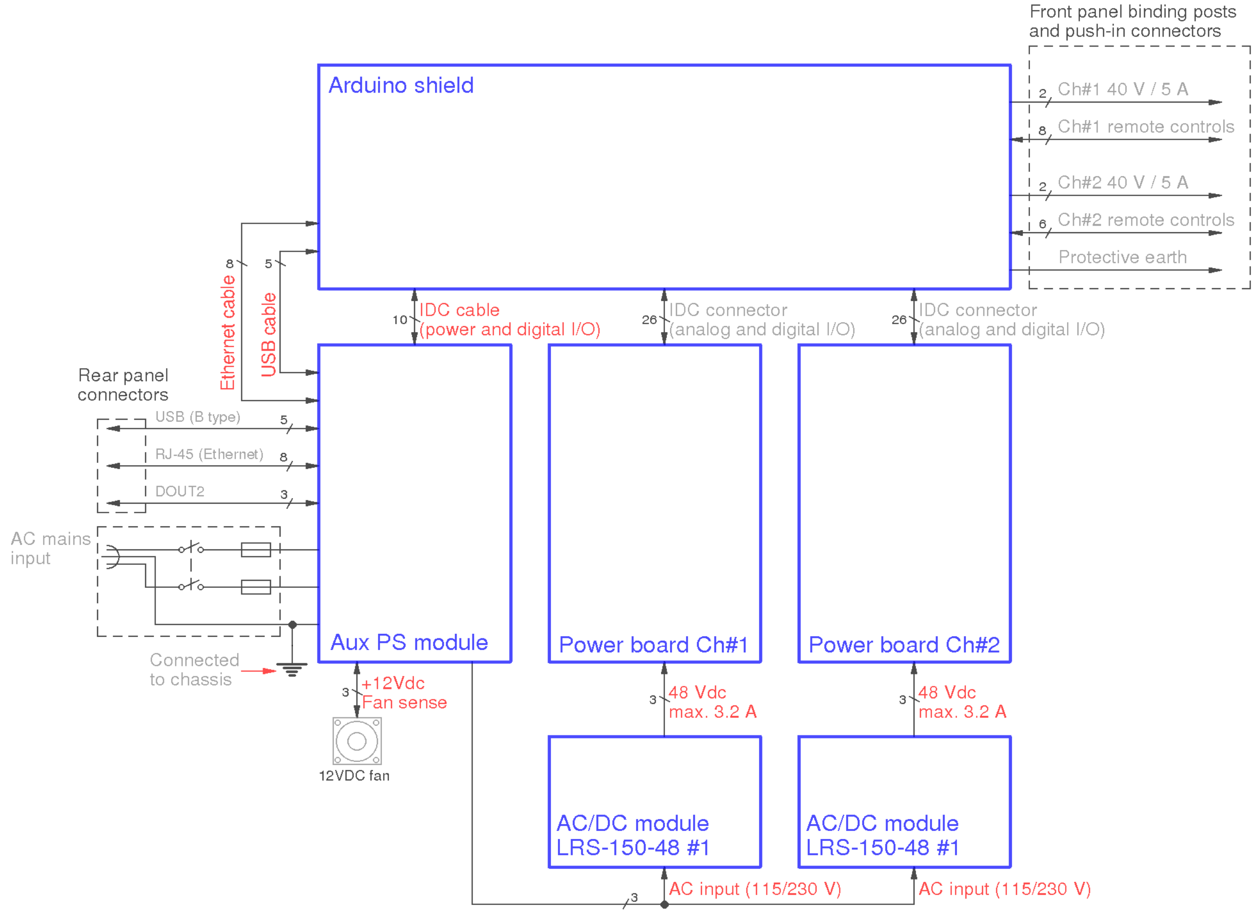

Fig. 17: Block diagram

2.1. Front panel mounting

The Arduino Shield is mounted directly on the enclosure front panel and it's fixed on six positions: five binding posts and 40-pin TFT touch-screen display connector.

The 3.2" TFT touch screen display has to be mounted first on the display mounting rail as shown on Fig. 18.

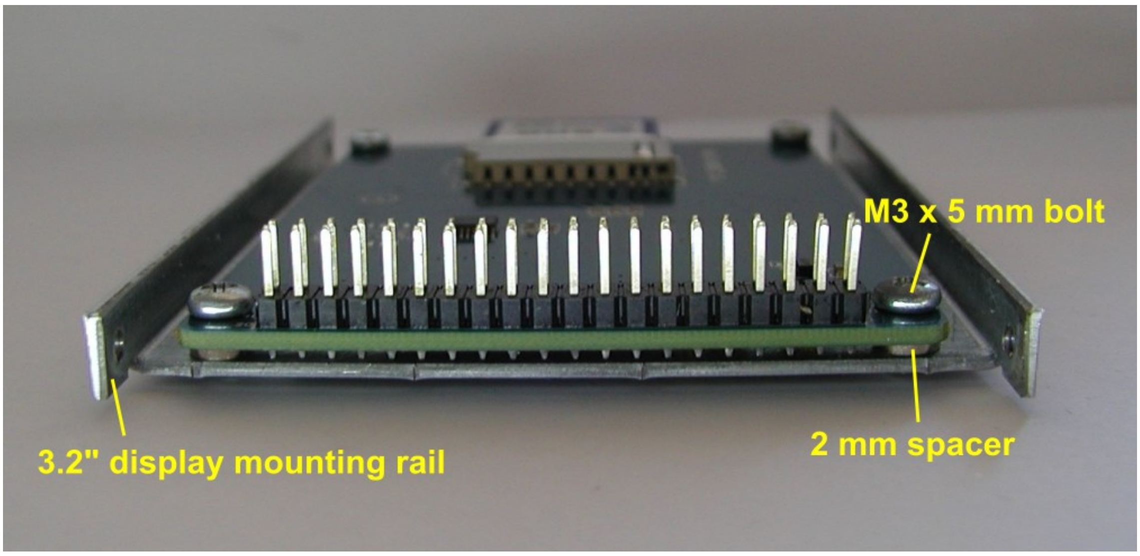

Fig. 18: 3.2" TFT display mounting

Before mounting display rail equipped with TFT display, a SD card has to be inserted into socket (Fig. 19).

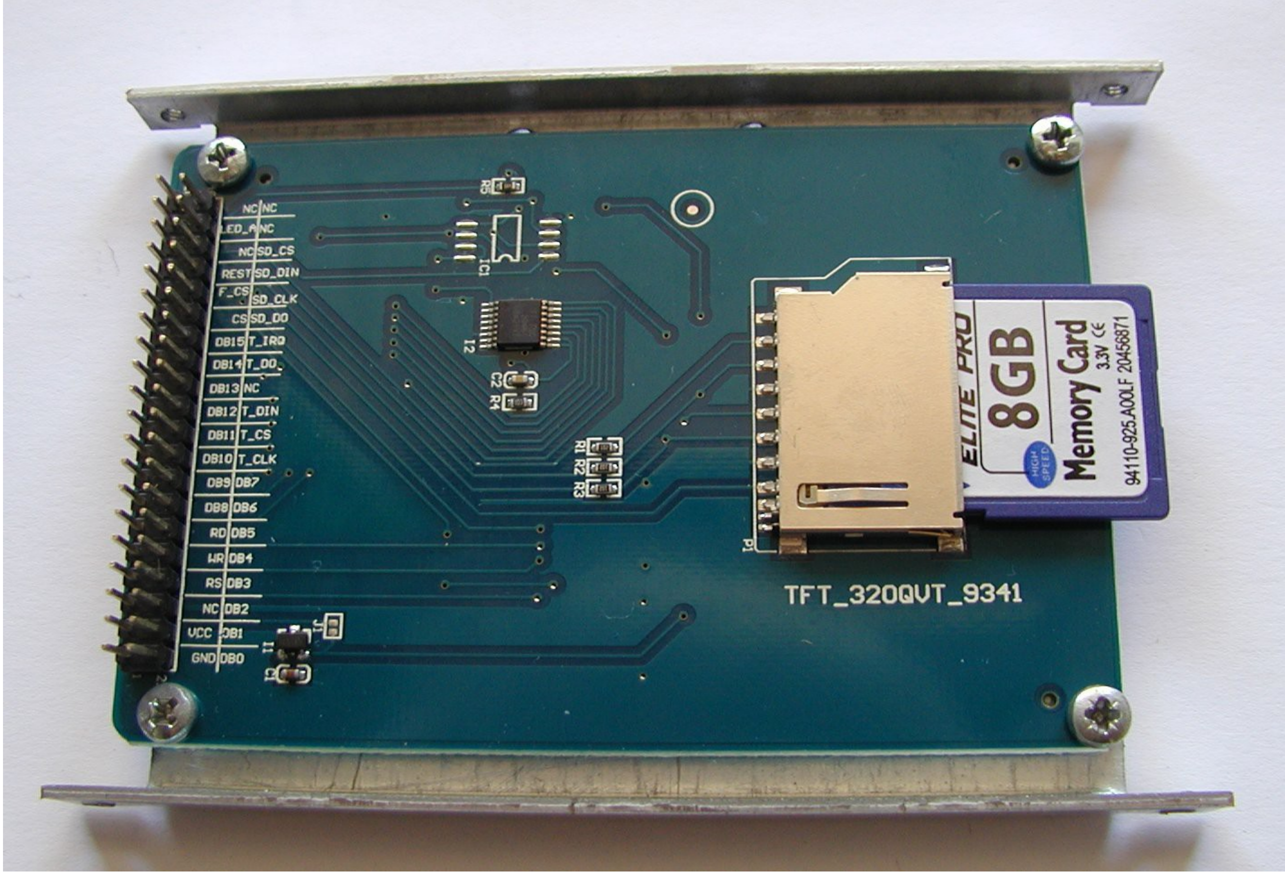

Fig. 19: TFT display mounted with inserted SD card

All LED indicators are mounted directly on the PCB bottom layer and light pipes has to be added that their lights can be visible on the surface of the front panel. Since no separators between LEDs exists for better effect we masks each light pipe with black heat shrink tube as shown in Fig. 20.

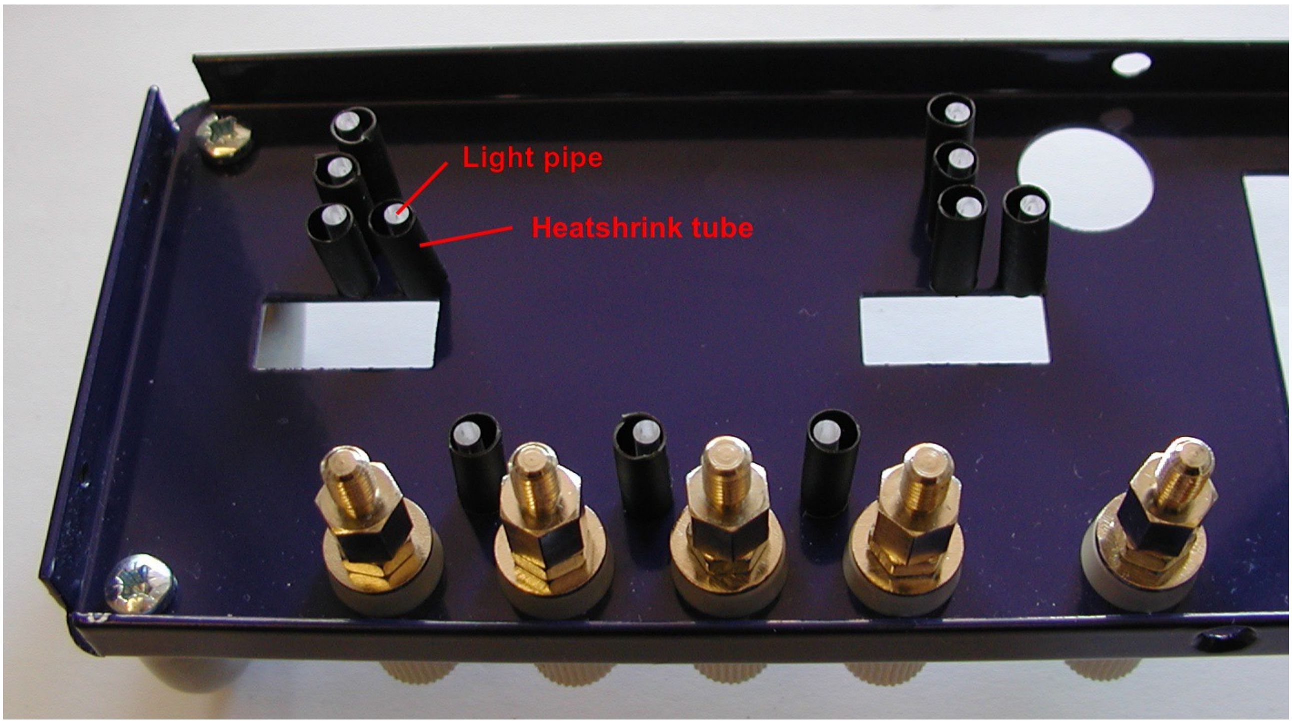

Fig. 20: LEDs light pipes with heat shrink tubes (not heated yet)

Selected biding posts are in the same time too short and too long that can be used to expose PCB's power outputs and protective earth to the front panel. It's too short that M5 nut with washer can be mounted on PCB's top side. Therefore some kind of spacer is needed. But, if spacer of e.g. 8 mm is used it become too long and it has to be cut as shown on Fig. 21.

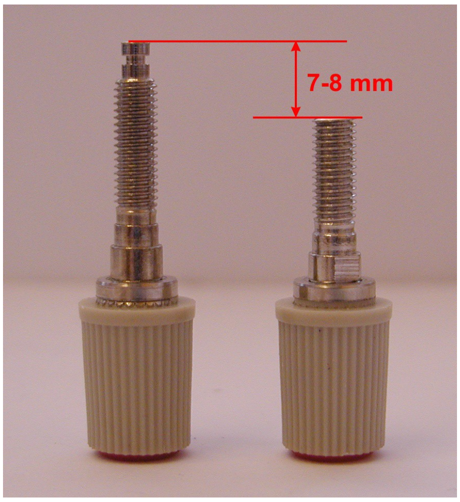

Fig. 21: Binding post cut

Binding post comes with two M5 nuts, Ø10 mm and Ø12 mm washers that can be used while mounting it on front panel (Fig. 22) and PCB (Fig. 23).

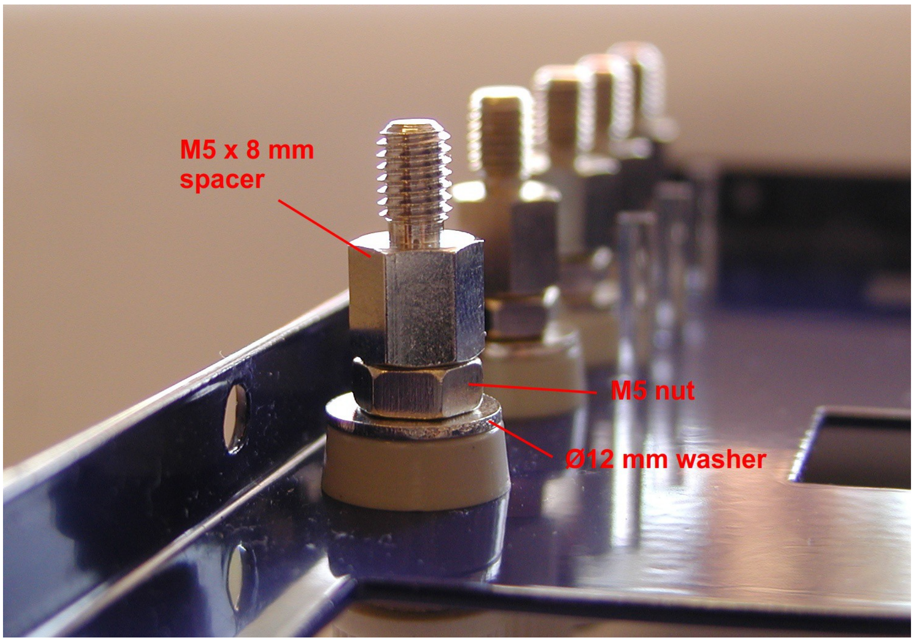

Fig. 22: Binding posts mounted on front panel and extended with spacers

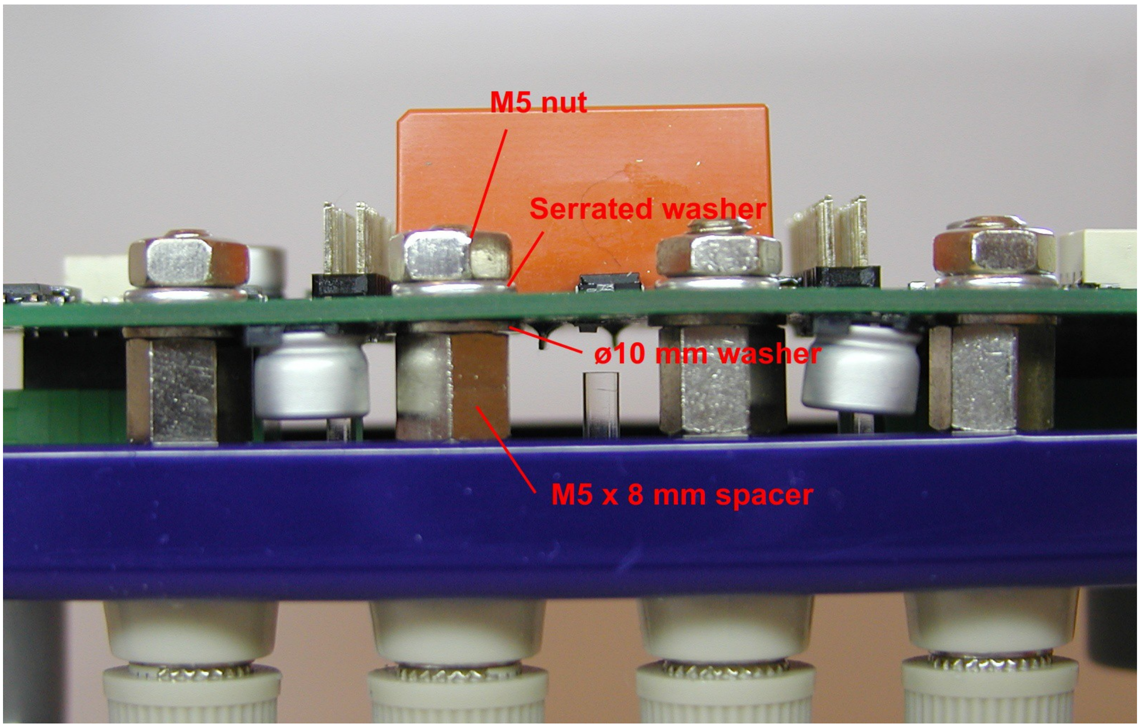

Fig. 23: Biding posts mounted (side view)

The Arduino Due board should be inserted before Arduino shield is mounted on the front panel. Cables as shown on Fig. 24 has to be inserted before front panel is fixed on the enclosure's bottom plate.

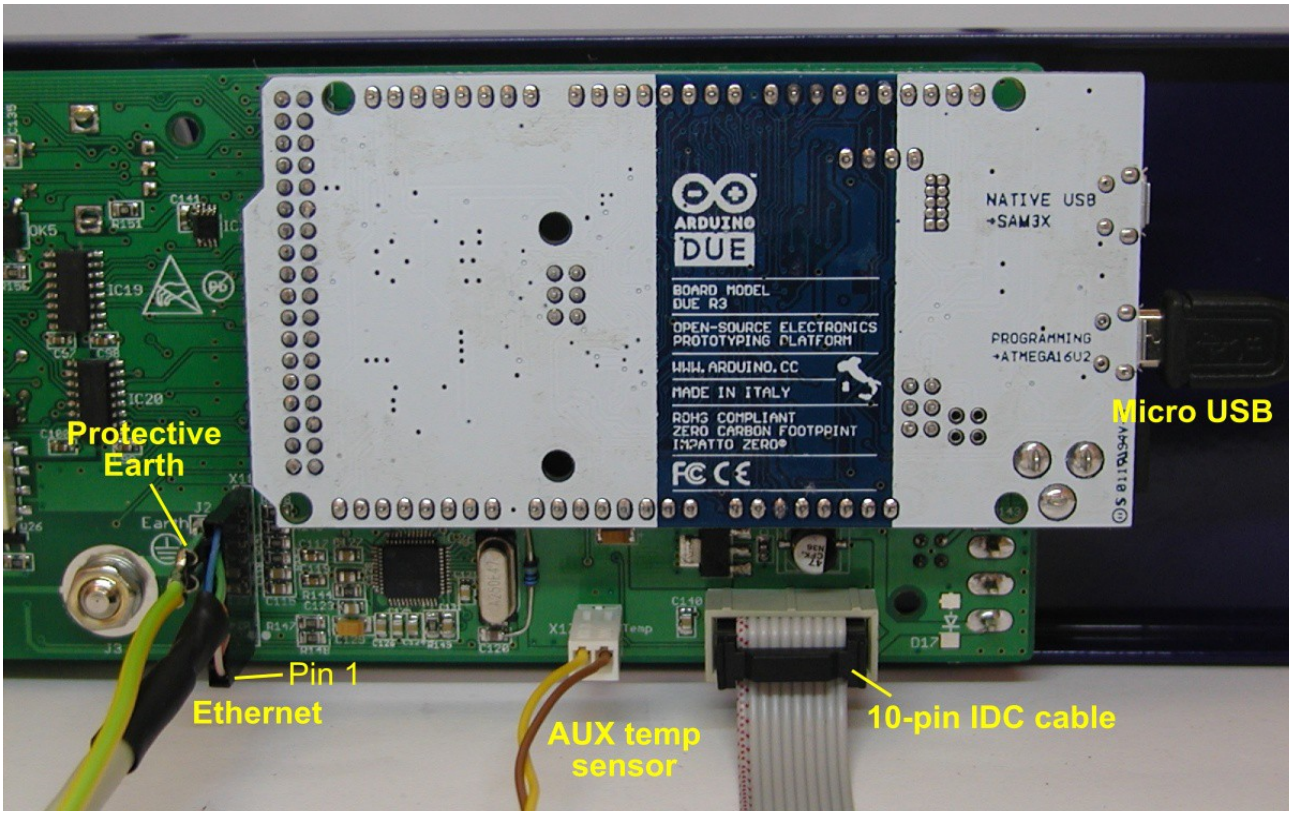

Fig. 24: Arduino shield wiring (r5B12)

2.2. Rear panel mounting

AUX PS board when assembled is mounted on the enclosure rear panel using five M3 bolts and 14 mm spacers. It's connected with other parts and modules with five cables as shown on Fig. 25.

WARNING: pay special attention when connect Ethernet cable. Wrong direction will permanently damage RJ-45 modular jack (X7).

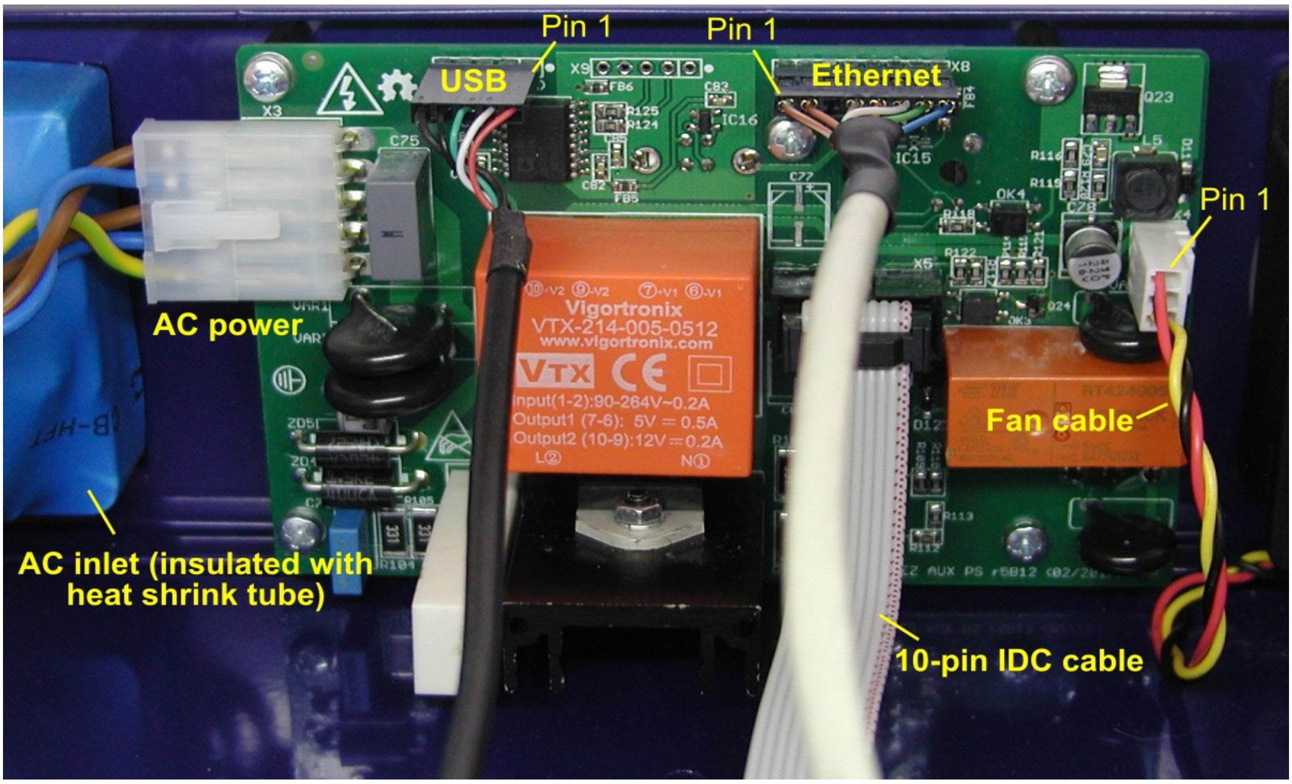

Fig. 25: AUX PS module wiring (r5B12)

The protective earth has to be connected to the single point on the metal enclosure as shown in Fig. 26. Make sure that ring terminal has good electric contact with enclosure by using externally serrated washer (example). Earthing requirement (that could also define mounting details) could vary from country to country therefore please check what is applicable in your case.

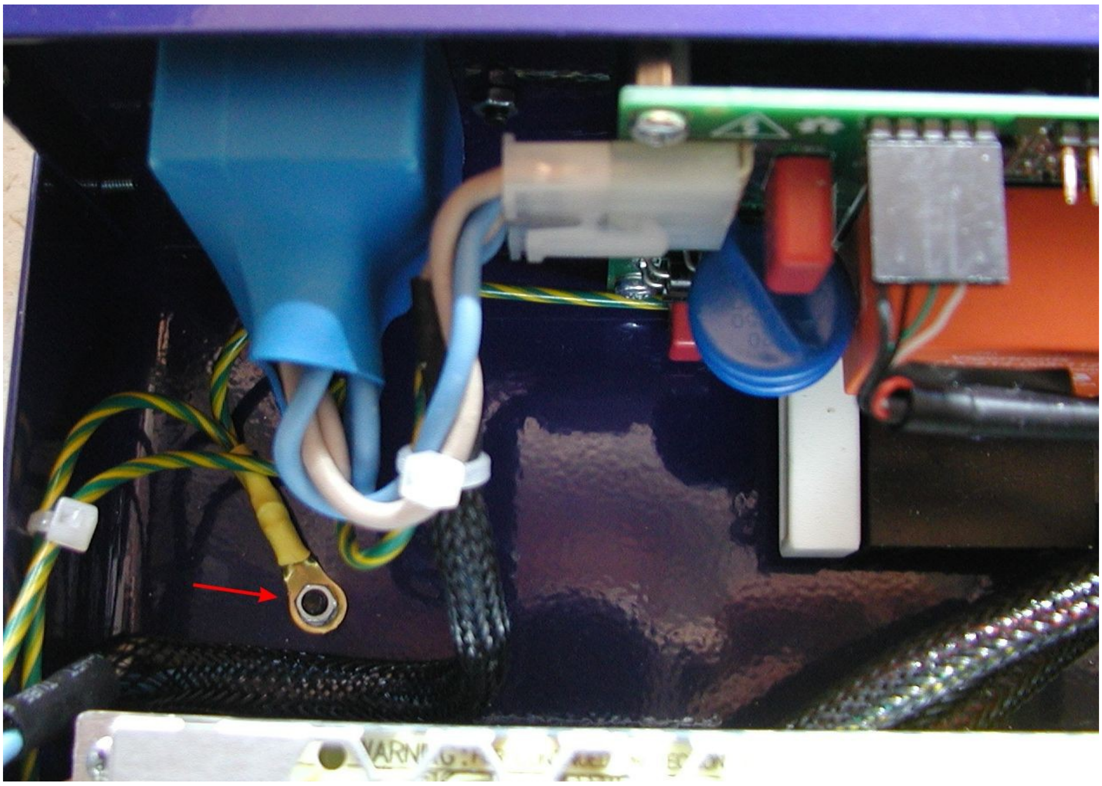

Fig. 26: Protection earth connection to enclosure

2.3. AC input selection

WARNING: take required precautions when working with AC mains voltage.

AC mains voltage defines what type of AC/DC modules or mains power transformer can be used for powering the Power boards. That also dictate fuse ratings and you can experiment with different fuse types and ratings depends of selected AC/DC module or mains transformer. In general the fuse current rating has to be twice higher for 115 Vac then 230 Vac. We are using 4 A (example) for two LRS-150-48 modules (max. combined output power is 320 W and with listed 90 % power efficiency min. 3.13 A is required). Two fuses are needed when AC IEC inlet such as Schurter 4304.6090 is used.

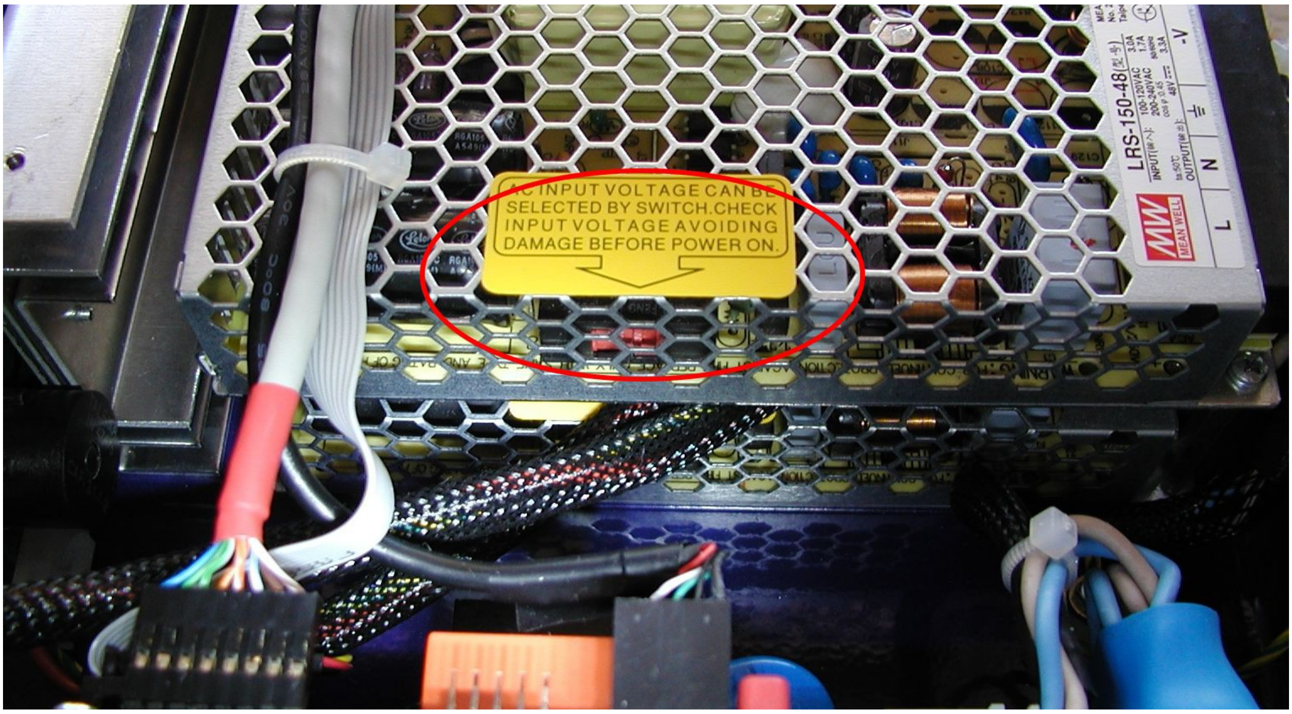

Fig. 27: AC input voltage switch for LRS-150-48

AC/DC power modules comes with and without so-called auto-range or auto-select feature. When manufacturer offer two models with the same output ratings, model without that feature is usually a little bit cheaper. For example, mentioned LRS-150-48 comes without auto-range and has a switch (Fig. 27.). Switch is not easily accessible when modules are mounted inside enclosure so make sure that switch is in proper position before mounting and AC mains voltage is applied.
Cable set #1 and *Cable set #2* are connected to the both AC/DC modules following the wire colors as shown on Fig. 28:

Name	**Position**	**Color**
AC line input	L	Brown
AC neutral input	N	Blue
Protective earth		Yellow-green
DC output negative	-V	Black
DC output positive	+V	Red

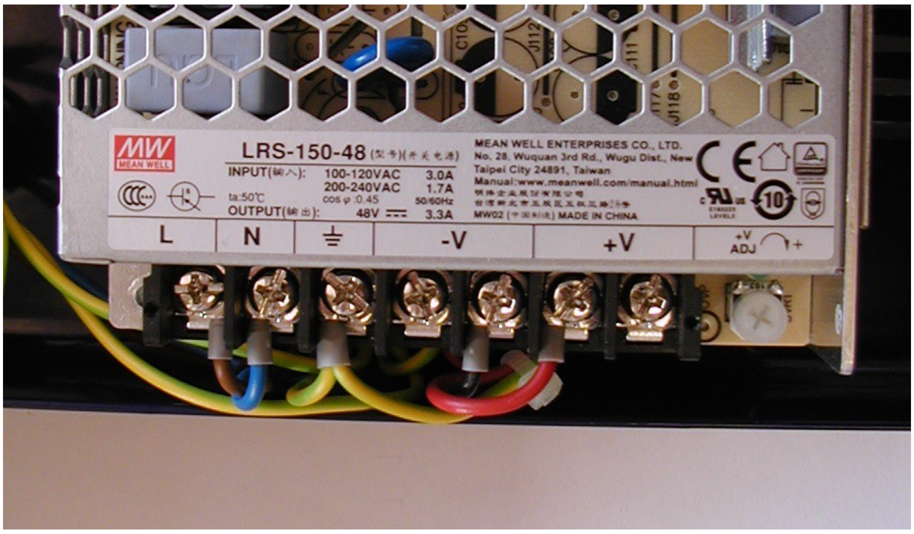

Fig. 28: AC/DC module terminals

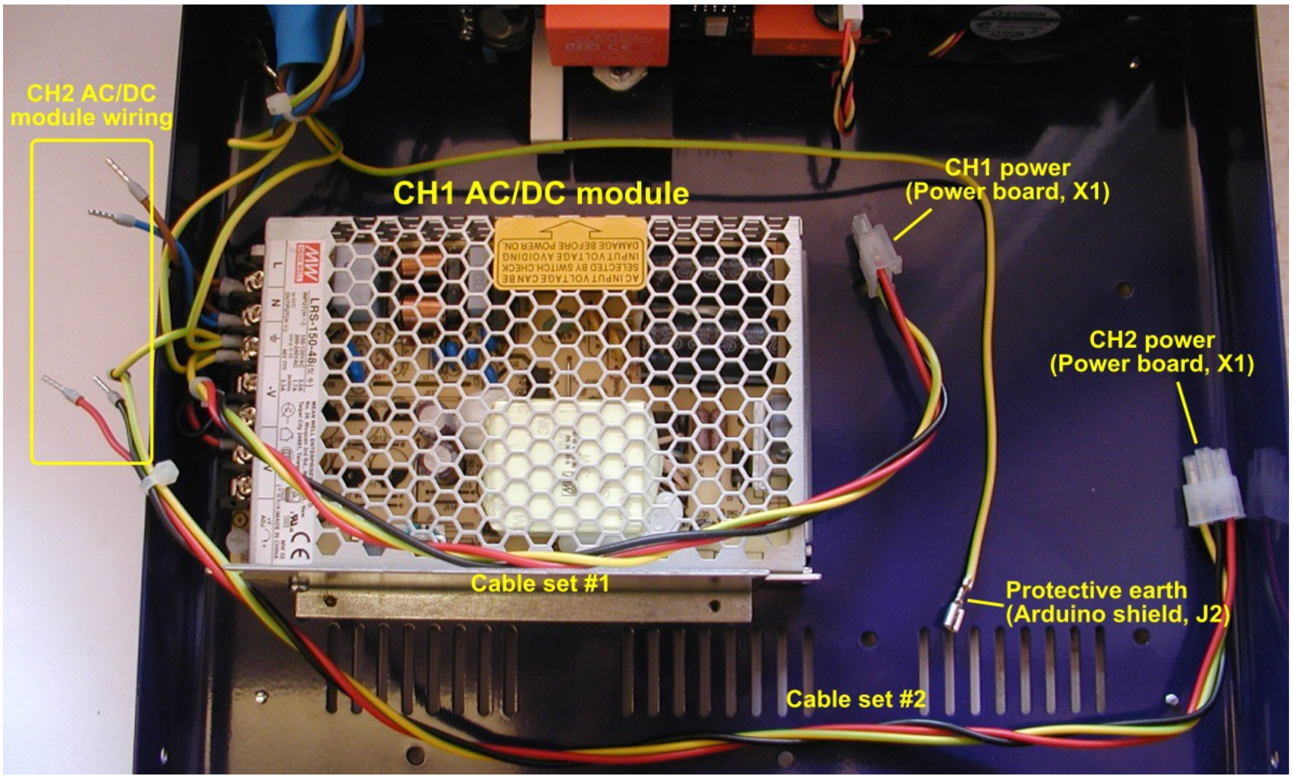

Fig. 29: Bottom (CH1) AC/DC module with Cable set #1 mounted

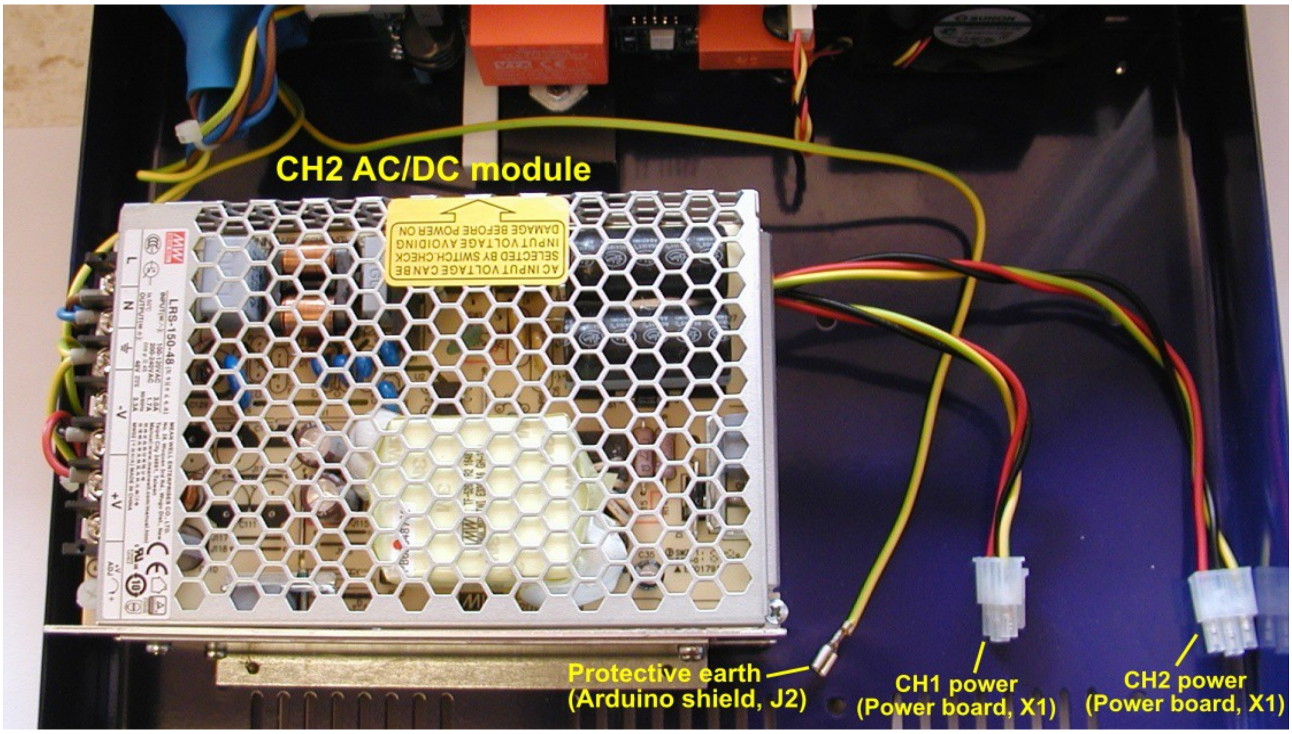

Fig. 30: Top (CH2) AC/DC module with Cable set #2 mounted (without terminal cover)

Two AC/DC modules has to be stacked one on top of another and fixed on one side with AC/DC mounting rails (Fig. 31). Additional two 40 mm long spacers are used for better mechanical strength.

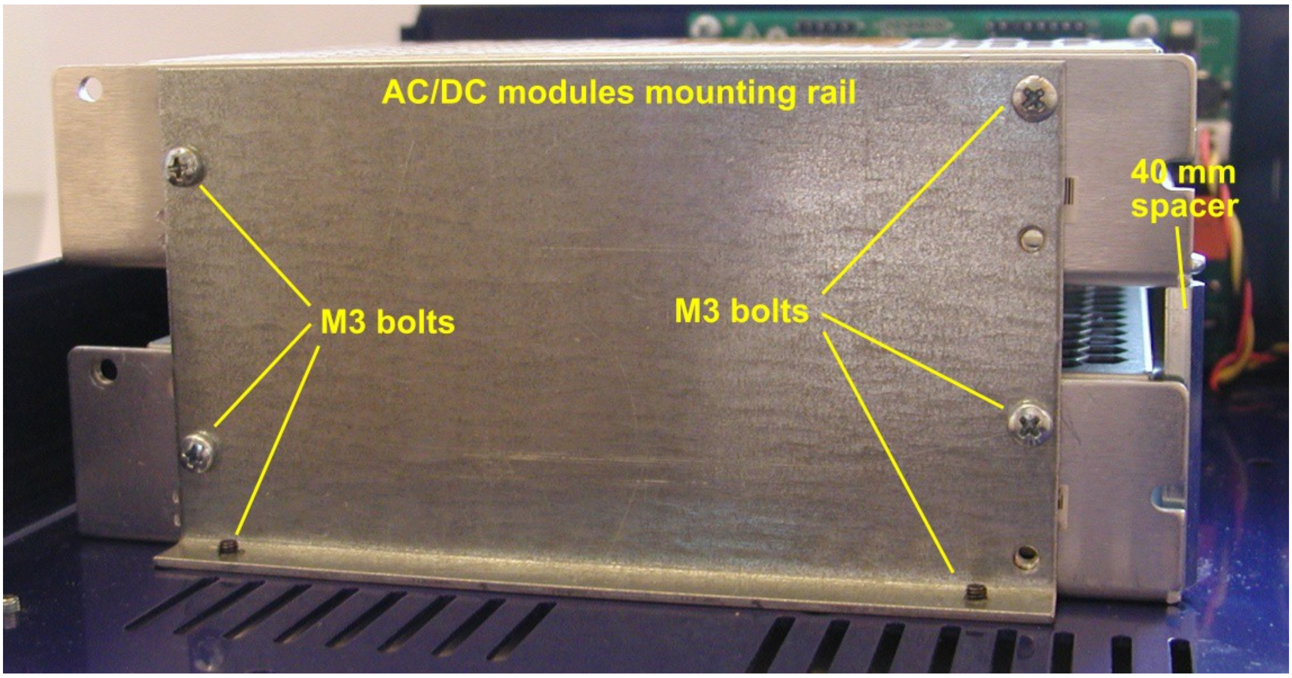

Fig. 31: AC/DC power modules mounting rail

Finally, an AC/DC module terminal cover should be mounted as additional protective measure (Fig. 32).

Fig. 32: AC/DC module terminal cover

3. Firmware uploading

The PSU firmware is an Arduino sketch. Therefore for its compilation and uploading to the Arduino Due board that resides on the Arduino Shield you have to install and run cross-platform Arduino IDE which can be downloaded here.
Installing Arduino IDE alone is not enough in our case because support for ARM boards such as Due is not installed by default. Therefore you have to add ARM Cortex-M3 board support. Navigate to **Tools… Board:… Boards manager** and select for installation *Arduino SAM Boards package*. When it's successfully installed, you can check that as shown in the Fig. 33.

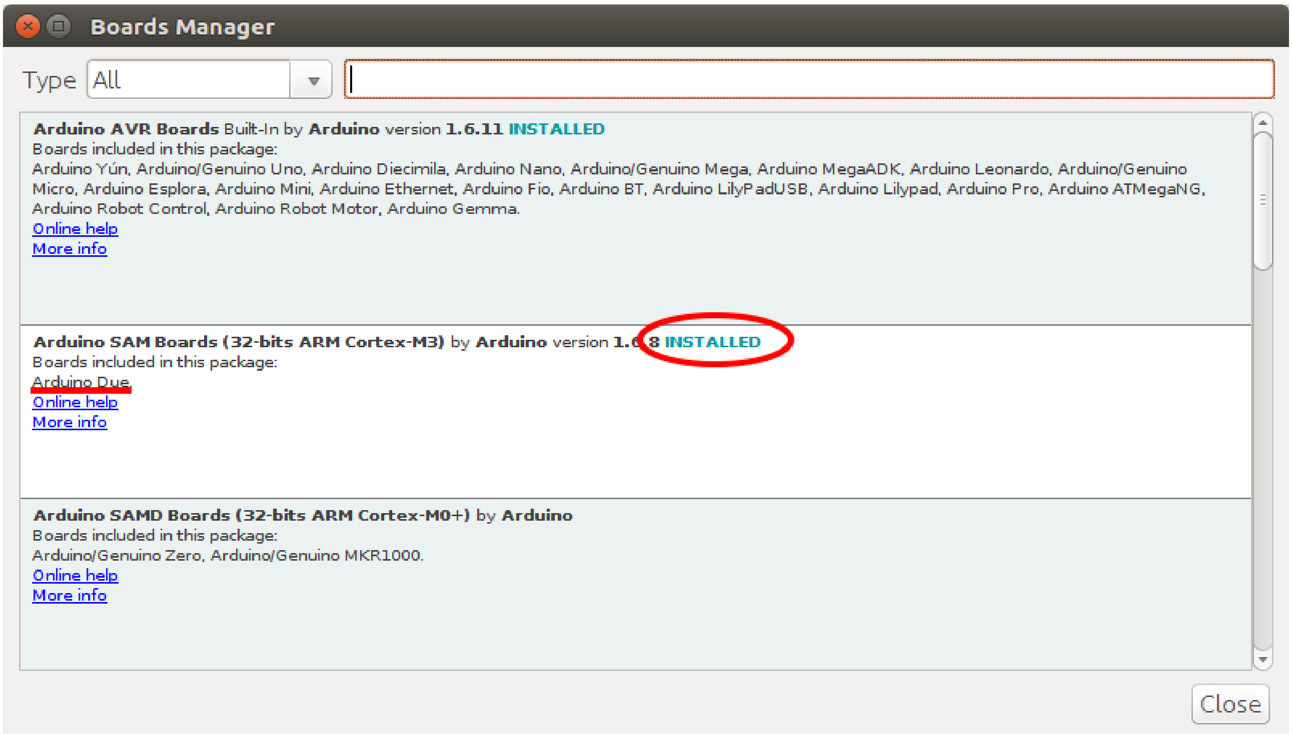

Fig. 33: Arduino boards manager with installed support for Due

Next step is selection of Board and USB since Arduino Due board has two USB ports. We'll use *Programming Port* as shown on Fig. 24 and Fig. 34.

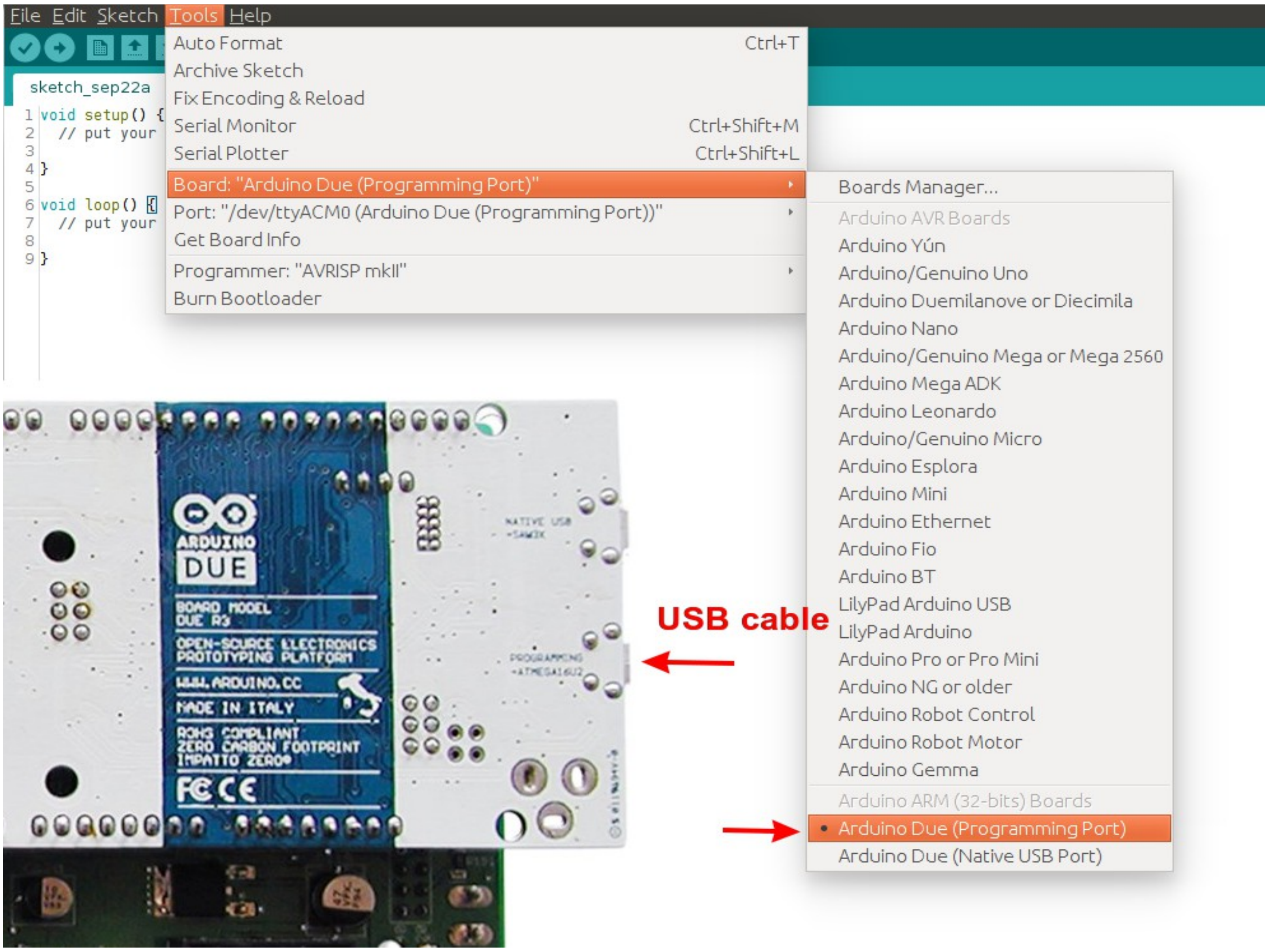

Fig. 34: Arduino DUE USB port selection

The Arduino IDE is now ready for work and we can continue with downloading firmware from the GitHub repository. The r5B12 boards versions require firmware revision *M4* or newer that can be found in Releases section.
Another possibility is to use the most recent build that can be found in *Master* branch. In that case move to **Clone or download** option and select **Download ZIP** option to save the latest build on your PC as shown on Fig. 35.

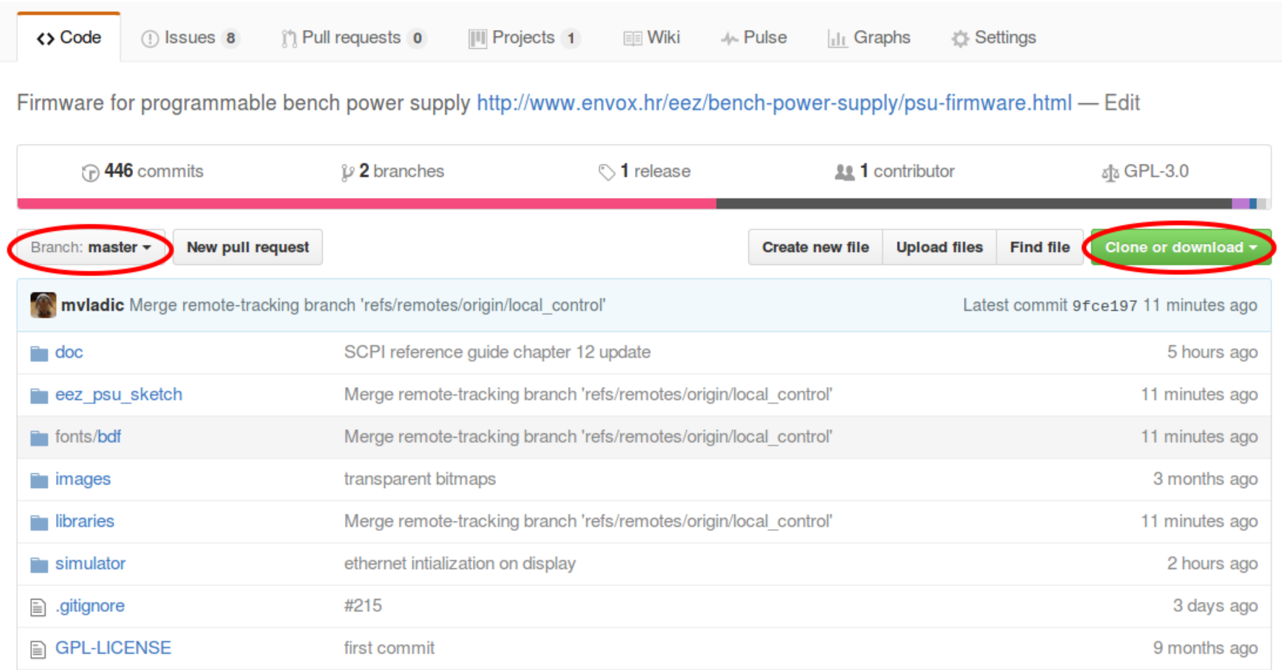

Fig. 35: Firmware download

When downloaded `master.zip` package is extracted, the following folder's tree should be created:

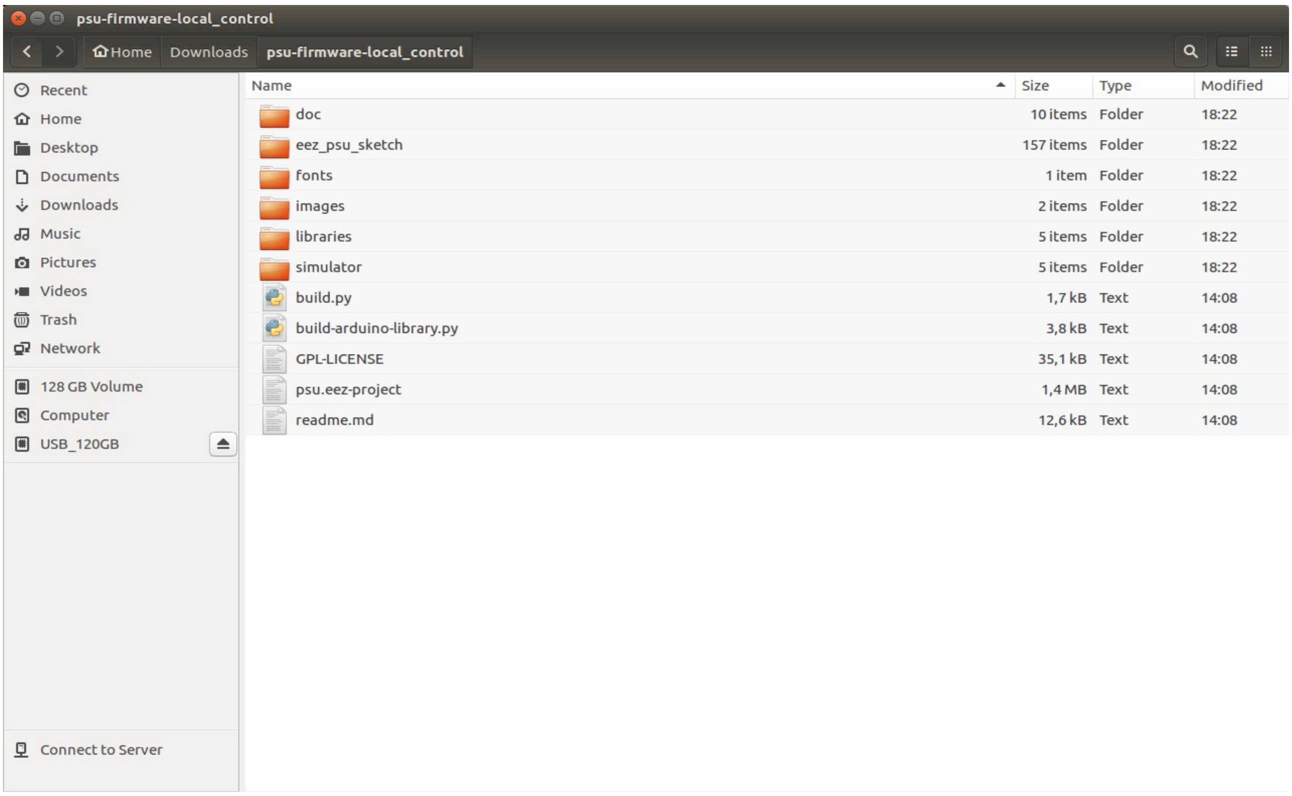

Fig. 36: Firmware folders (Ubuntu Linux)

Contents of two folders will be used for firmware compiling within the Arduino IDE: `eez_psu_sketch` and `libraries`. The contents of later one has to be copied into Arduino folder that contain 3rd party libraries. That could be e.g. `/home/denis/Arduino/libraries` on Linux or `My Documents\Arduino\libraries` on Windows. See also this article.
The firmware upload require the following steps:

- Power up EEZ H24005
- (Re)start Arduino IDE

- Check if proper USB port is selected under **Tools… Port:**
- Load firmware sketch `eez_psu_sketch.ino` (**File… Open**)
- Make sure that *Verify after code upload* option is not selected (**File… Preferences**) to speed up upload that is even without verification frustratingly slow
- Start Upload (**Sketch… Upload** or select the Upload icon on toolbar)
- While upload procedure started a progress bar will appear after compiling (Fig. 37).

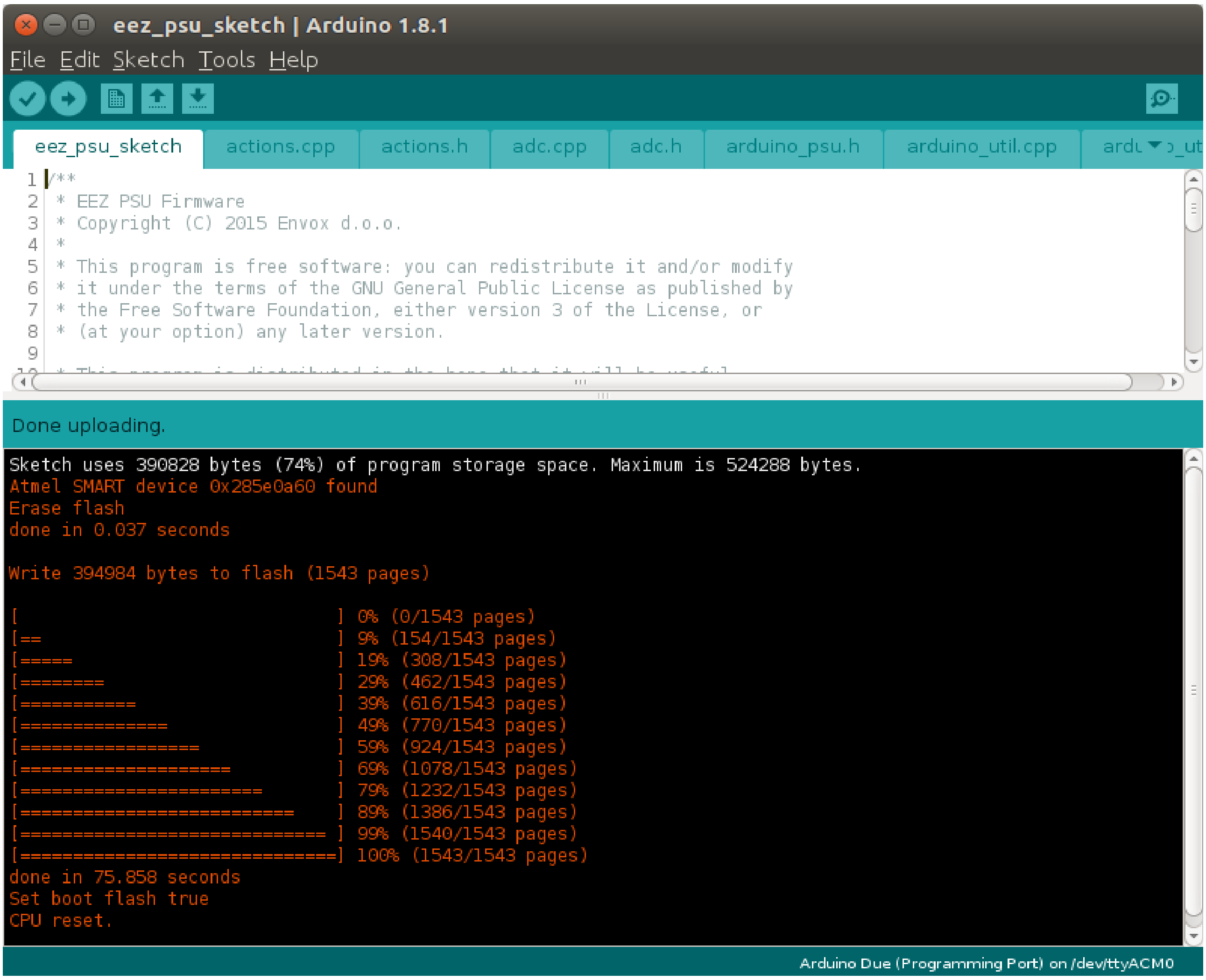

Fig. 37: Arduino sketch is successfully uploaded

On first start the initial touchscreen calibration page (Fig. 38) will be displayed otherwise a welcome page (Fig. 39) will be displayed for few seconds. You cannot skip this test since without calibrated touchscreen interaction could be erratic or even impossible.

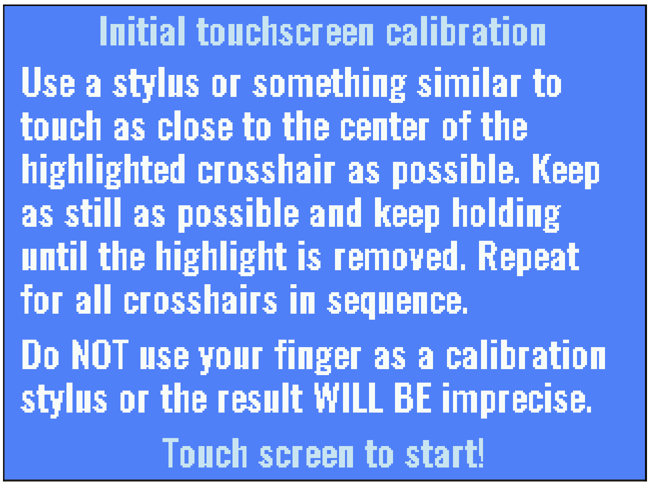

Fig. 38: Initial screen calibration on first power on

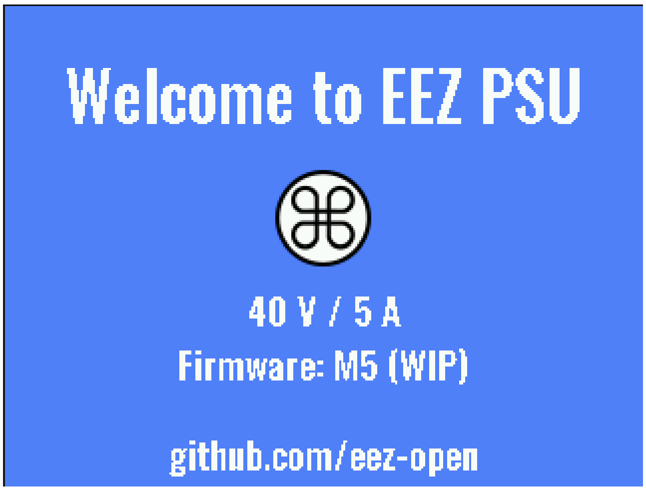

Fig. 39: Welcome page

When initial touchscreen calibration is finished, the self-test will be initiated and it is a good indicator that all control over all circuits can be established. If self-test is passed, no additional message will be displayed, and firmware will proceed with the main page (Fig. 40).

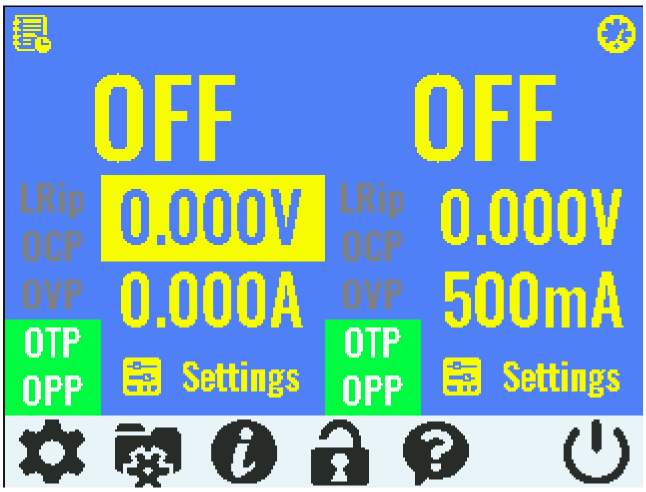

Fig. 40: Main page

Work in progress