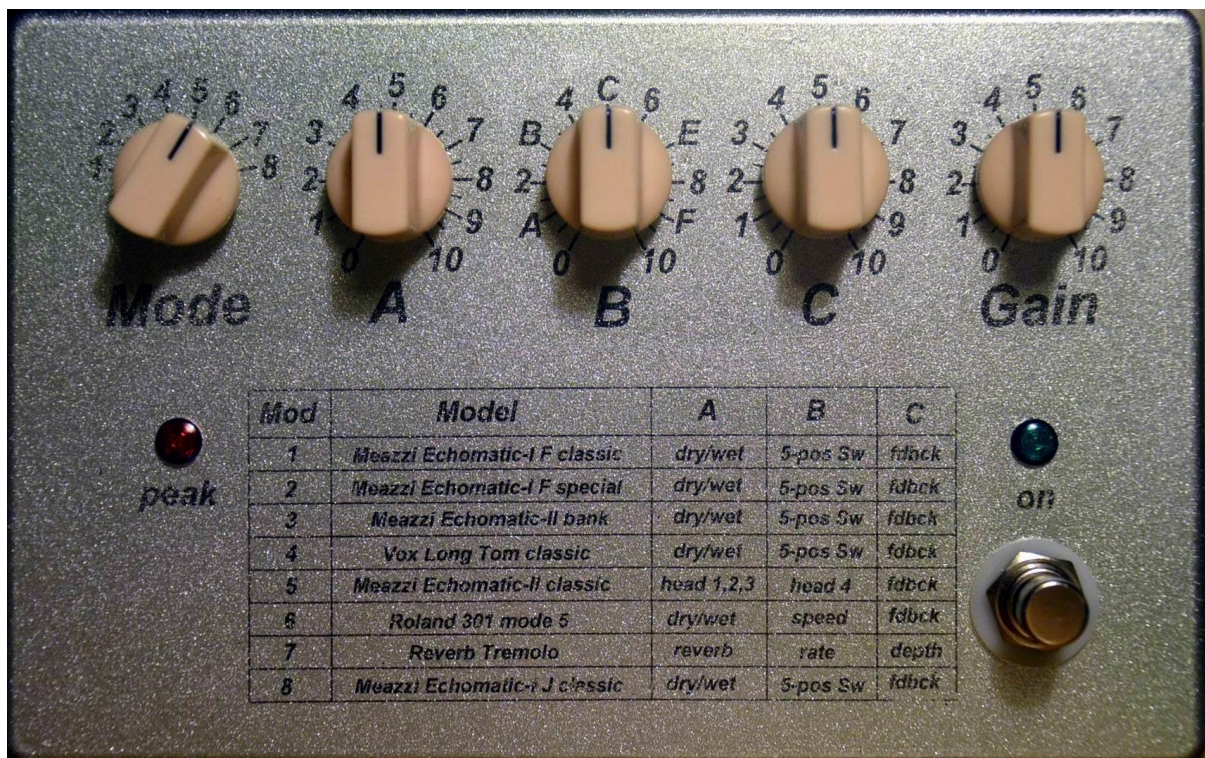


# eTap2HW

*Vintage echo emulation DIY kit*



## ***BUILDING INSTRUCTIONS***

### ***Newtone Versiion***

©May 2013

*P. Verbruggen, Newtone-online.nl, G.J. van Helden, M. Snijders, M. Boesewinkel*

*English Translation by S.C. Mitchell*

# **CONTENTS**

<b>Introduction</b>	<b>3</b>
<b>Chapter 1</b>	<b>4</b>
1.1 ESD warning	4
1.2 Considerations	4
1.3 Further supplies	4
1.4 Bill of materials	5
<b>Chapter 2 Building Instructions</b>	<b>6</b>
2.1 Points to note	6
2.2 Layout of the Motherboard components	7
2.3 Wiring	19
<b>Chapter 3 Testing and reduce</b>	<b>24</b>
3.1 Testing the unit	24
3.2 Drilling	29
3.3 Functions and potentiometers BCD switch	30
<b>Chapter 4 Modifications</b>	<b>31</b>
4.1 Leds	31
4.2 FET installation in a nutshell	33
4.3 5-way switch instead of potentiometer B	34
4.4 Modifications to the power supply	35
4.5 Schedule	36
<b>Decal example 1:1</b>	<b>37</b>

# Introduction

The eTap2HW was developed by Peter Verbruggen and is distributed via Newtone-online. It is a guitar effect that provides emulations of the first tape and disk echoes equipment. These effects are an integral part of the sound of the guitarist (Hank B. Marvin) of the band The Shadows. Peter Verbruggen has responded to the demand for a compact and affordable DIY solution to capture the sound without the maintenance requirements of the expensive and sensitive originals. The result is a fine set of disk and tape echo emulations that are a must for people who want to play, like The Shadows, but also for lovers of irregular timings that the old echoes made their own. The following echo equipment is emulated:

- Meazzi Echomatic-I model 'J' Classic
- Meazzi Echomatic-I model 'F' Classic
- Meazzi Echomatic-II Bank
- Meazzi Echomatic-II Classic
- Vox LongTom Classic
- Roland model 301 Mode 5
- There is also a separate program for Reverb and/or Tremolo

Peter Verbruggen first developed a worldwide leading software version for these emulations (a VST plug-in called eTap2) and now has developed a hardware version to use in a live situation. On the basis of extensive studies of recordings by The Shadows the timings of the echoes are accurately simulated along with the so-called wow and flutter that was a feature of the old equipment. For this he used a self-programmable chip, the FV-1. This is plug\_and\_play and is provided in the form of the SKRM-daughter board. To approximate the vintage equipment as accurately as possible and also to capture the very characteristic tube sounds, extra attention is paid to the pre-and post-amp sections. For example, use is made of the so-called Fetzer Valve to create specific and pleasant sounding 'tube-like' harmonics.

Detailed information about the creation can be found on the website of Peter Verbruggen: <http://www.echotapper.nl>

The eTap2HW is constantly evolving, and we set ourselves the task of putting together and supplying a complete working package from Newtone.

It is not recommended as a project for the novice builder, because some basic knowledge and skills are required. Please adhere to this advice. The package is designed to be built by those builders who are a little more advanced using standard tools such as a soldering iron with a sharp tip, (stripping) pliers, screwdriver, a digital multimeter, drills (6.5mm, 7mm, 8mm, 9mm, 12mm), etc.

With this construction description we offer an alternative to the original which is available for download on the [echotapper website](http://www.echotapper.nl)

Here there is also information about programming the SKRM FV-1 board

We wish you much fun with the construction and use of the eTap2HW!

# **Chapter 1**

## **1.1 ESD Precautions**

The SKRM-daughterboard is packed in anti-static material. A static discharge could induce sparks to the board and damage it beyond repair. Leave the board in the special package until it's going to be used. The following rules should be applied while handling the board:

- Avoid carpets in cool, dry areas. Leave PC cards and memory modules in their anti-static packaging until ready to be installed.
- Dissipate static electricity before handling any system components (PC cards, memory modules) by touching a grounded metal object, such as the system unit unpainted metal chassis.
- If possible, use antistatic devices, such as wrist straps and floor mats.
- Always hold a PC card or memory module by its edges. Avoid touching the contacts and components on the memory module.
- Take care when connecting or disconnecting cables. A damaged cable can cause a short in the electrical circuit.
- Prevent damage to the connectors by aligning connector pins before you connect the cable. Misaligned connector pins can cause damage to system components at power-on.

## **1.2 Considerations**

This instruction is geared toward the standard build of the eTap2hw unit. There are many additional ways the unit can be extended, some examples are in Chapter 4, we suggest to read this section before starting the project.

- Some examples items covered in chapter 4:
- You require using different LED's or LED connectors.
- You need or consider a different FET settings (bias)

## **1.3 Further Supplies**

In addition to the parts you've received you need to use some additional parts like Tools and solder equipment including solder. You also need:

- A 12 Volt mains adaptor with a minimum of 100mA (see chapter 4.4 for 9V modification)
- Some wires advisable to be in different colours as you need to connect the main board to the outside world.
- As wires are also used with a pin/socket connection it is also recommended to use heat shrink tubing to insulate.
- Decal and paint to finish the enclosure

## 1.4 Bill of Materials

- 1 Aluminium enclosure type DD
- 1 PCB eTap2HW motherboard
- 1 PCB (populated) SKRM daughterboard from OCT with eTap2hw pre-programmed
- 1 3PDT switch
- 2 Mono jacks
- 1 DC connector
- 3 10k linear potentiometers
- 1 1M log potentiometer
- 1 Lorlin BCD-switch
- 2 Bright LED's transparent
- 1 Fresnel lens red (peak)
- 1 Fresnel lens green (on)
- 4 Miniature spacers (15mm)
- 1 set solder pins
- 1 2-pins connector
- 3 IC socket, 8-pins
- 1 2x8-pins header
- 1 Transistor socket, 20-pin in line
- 3 NE5532p dual OPAMP's
- 1 7809 voltage regulator
- 1 Heatsink 7809 Voltage regulator
- 1 1N4004 diode
- 2 J201 FET's
- 1 470 $\mu$ F/25V elco
- 2 100 $\mu$ F/16V elco's
- 1 10 $\mu$ F/16V elco
- 9 1 $\mu$ F MKT capacitors
- 2 100nF MKT capacitors
- 2 15nF MKT capacitors
- 2 1nF MKT capacitors
- 3 220pF ceramic capacitors
- 2 120pF ceramic capacitors
- 1 10 $\Omega$  resistor
- 4 1k $\Omega$  resistors
- 1 1,8k $\Omega$  resistor (peak LED)
- 2 2,2k $\Omega$  resistors (FET source resistor)
- 2 6,8k $\Omega$  resistors (FET drain resistor)
- 8 10k $\Omega$  resistors
- 1 22k $\Omega$  resistor (on/bypass LED)
- 1 33k $\Omega$  resistor
- 2 68k $\Omega$  resistors
- 4 100k $\Omega$  resistors
- 2 2,2M $\Omega$  resistors



Check contents of kit before you start.



# Chapter 2 Building Instructions

## 2.1 Points to Note

- Read chapter 1 carefully before starting the project build
- This instruction manual is written in a way that it provides you with all the information required to successfully build the unit. It is therefore highly recommended to follow the instructions provided step by step
- The placement of the components is structured from 'low profile' parts to higher profile parts, the first parts installed are therefore the resistors then the IC sockets and gradually to the higher electrolytic capacitors. By using this method the board can be put upside down assuring components are all flush with the board when soldering takes place.
- The printed circuit board does not have silk-screening so extra care is required to assure the parts are inserted correctly before the actual soldering takes place. It is also recommended to use an AVO meter to measure each resistor before inserting as this guarantees the correct values. (Colour bands are difficult to read)

Points requiring special attention are marked in the build instructions with:



! Extra attention, part could be placed wrongly



\* A part possible affected by alternative build requirements (Chapter 4)



**NB** Remarks on parts not part of the kit



This is a connection. .

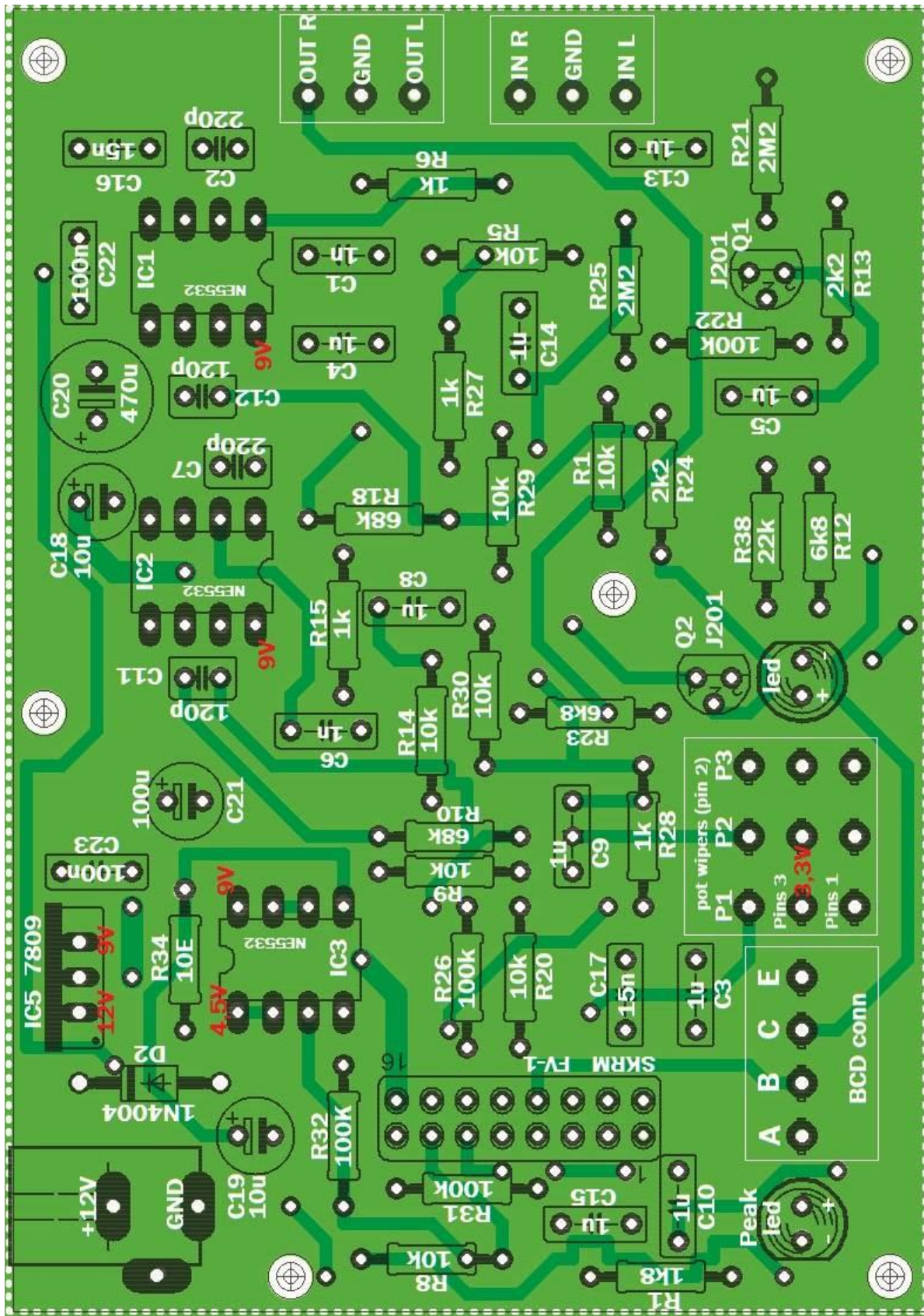
-Use solder iron with sharp point at correct temperature and use solder for electronics.

- Never apply more solder than required. With too much solder applied, bridging between two solder joints can happen creating shorts. Also, note that soldering time of a solder pad that's not connected to the earth shield takes shorter than a solder pad connected to the earth shield.

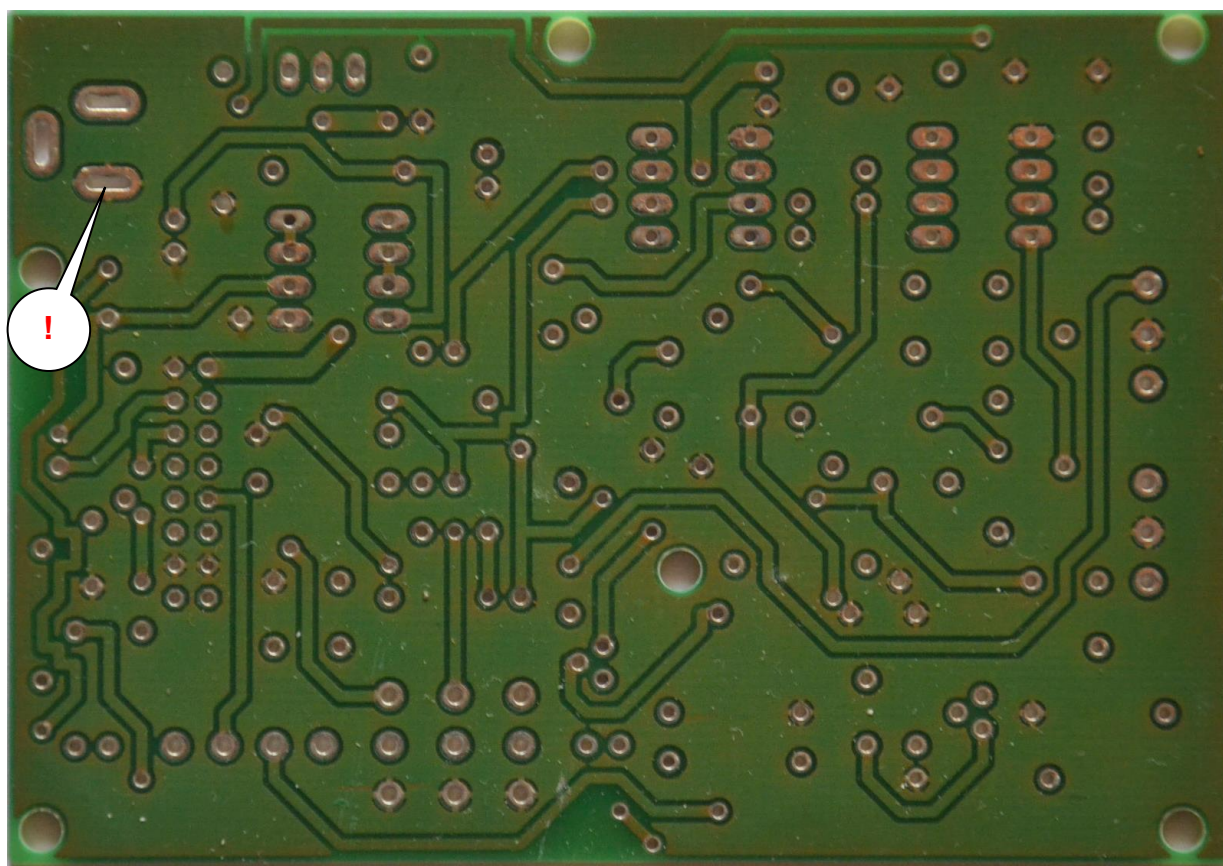


- The NE5532 OPAMP's, the J201 FETs including the SKRM daughterboard are installed in Chapter 3: 'Testing and final build'

*As there is no silkscreen available this layout provided can be used as a guide. Voltages required for tests (later in paragraph 3) are given already here in red.*



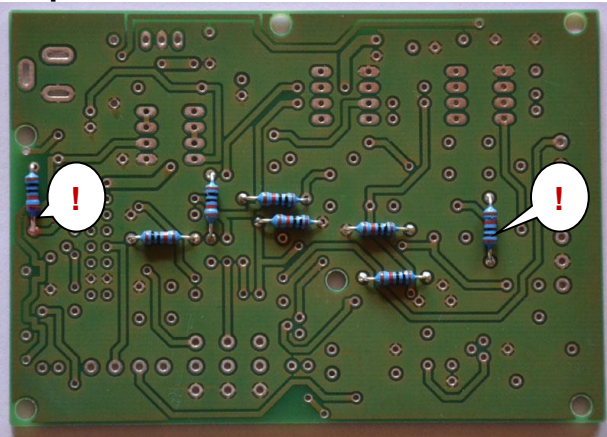




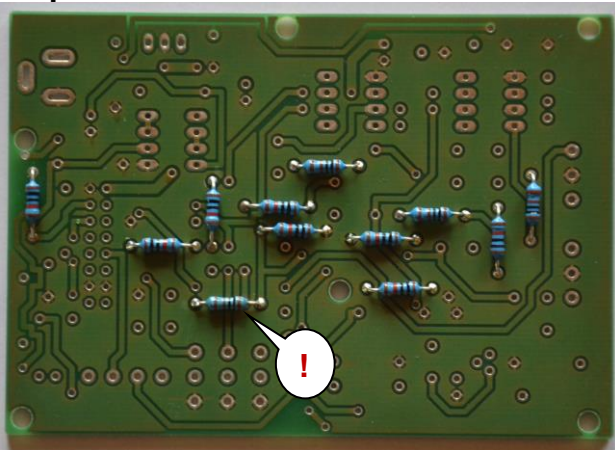
*This is the position of the board*

! The DC connection is in the upper left corner.

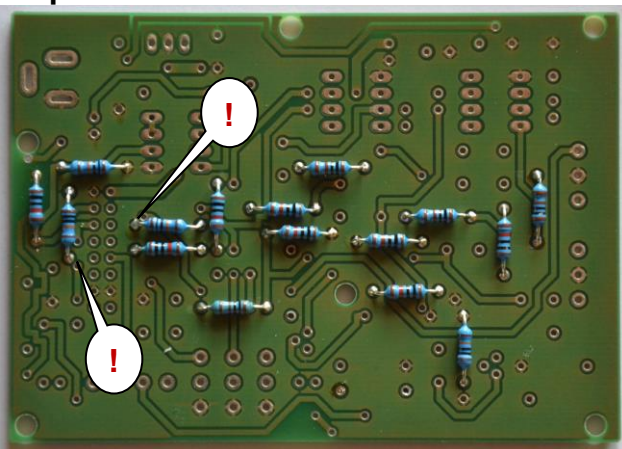


**Step1**

Place 8x 10k resistor. (R5, R8, R9, R14, R17, R20, R29, R30)  
! Warning, via's (holes) under the resistor.

**Step 2**

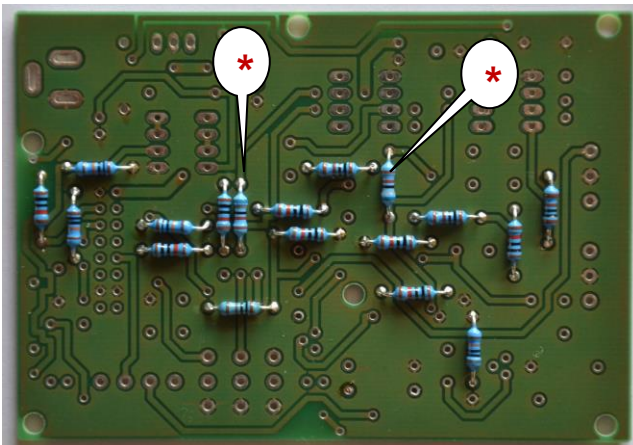
Place 4x 1k resistor. (R6, R15, R27, R28)  
! Warning, via's (holes) under the resistor.

**Step 3:**

Place 4x 100k resistor (R22, R26, R31, R32)  
! Warning via's in the vicinity of the resistors.



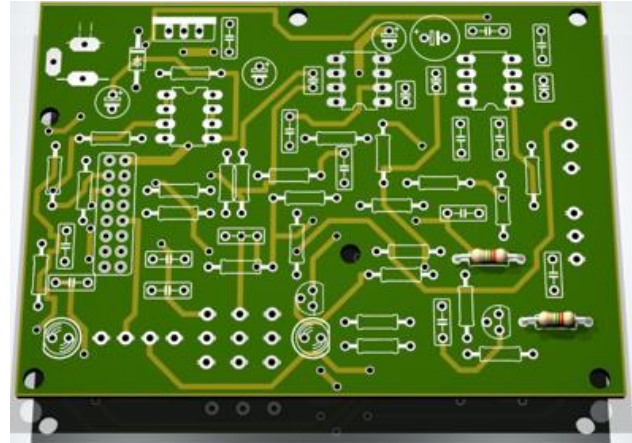
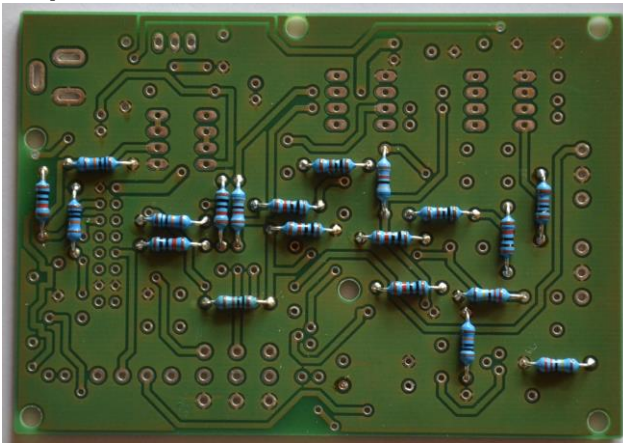
**Step 4:**



Place 2x 68k resistor (R10, R18)

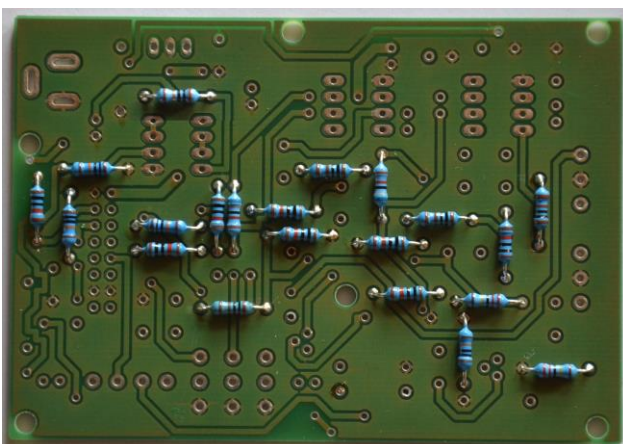
\* Also see chapter 4.2 '*FET optimisation*'...

**step 5:**



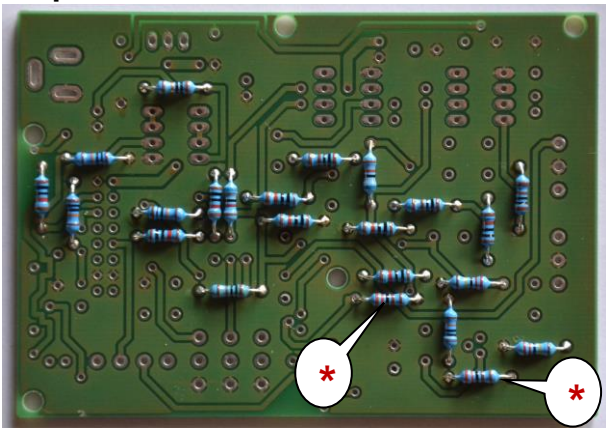
Place 2x 2M2 resistor(R21, R25)

**step 6:**



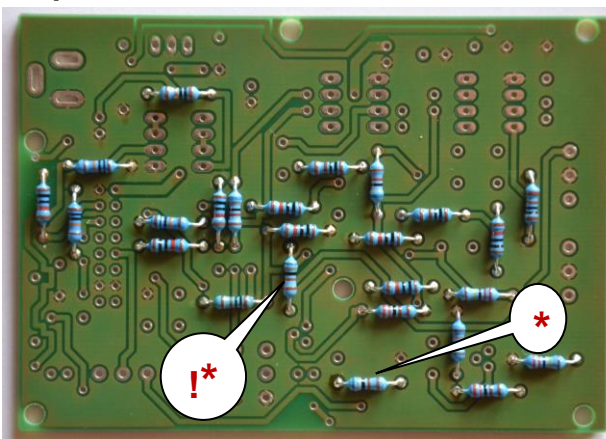
Place the 10R resistor (R34)



**Step 7:**

Place 2x 2k2 resistors (R13, R24)

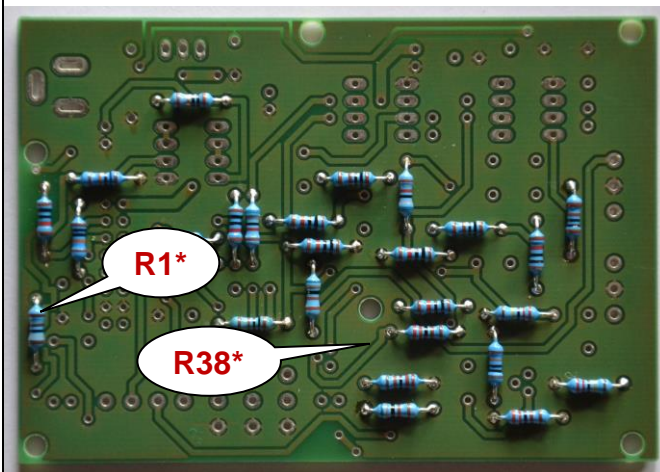
\* These values are general use values for the J201 FET transistor source. The value chosen does have a measurable impact on the working of the FET. Chapter 4.2 FET settings: provides detailed information.

**Step 8:**

Place 2x 6k8 resistors (R12, R23)

! Warning, via's under the resistors

\* These values are general use values for the J201 FET transistor drain. The value chosen does have a measurable impact on the working of the FET. Chapter 4.2 FET settings provides detailed information.

**Step 9a:**

Install the 22K resistor (R38) (Resistor for the "power on" LED for ~0.28mA at 9V))

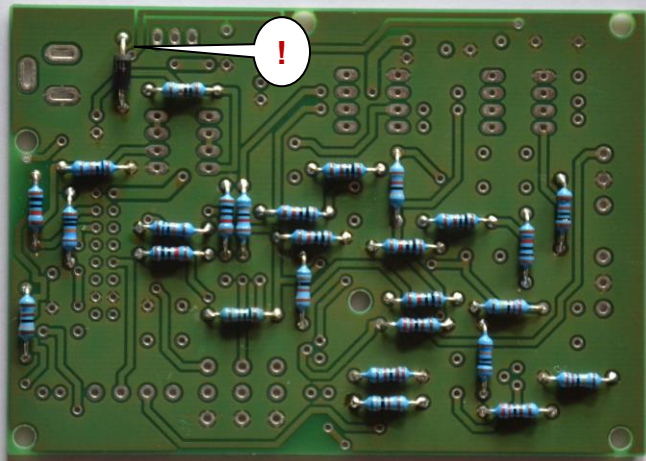
**Step 9b:**

Install 1K8 resistor (R1)  
(LED resistor for the "peak" LED approx. 0.33 mA at 3.3V)

\* These are the recommended values provided by us. Chapter 4.1 'LEDs' provides formulas to calculate values for other LED's.



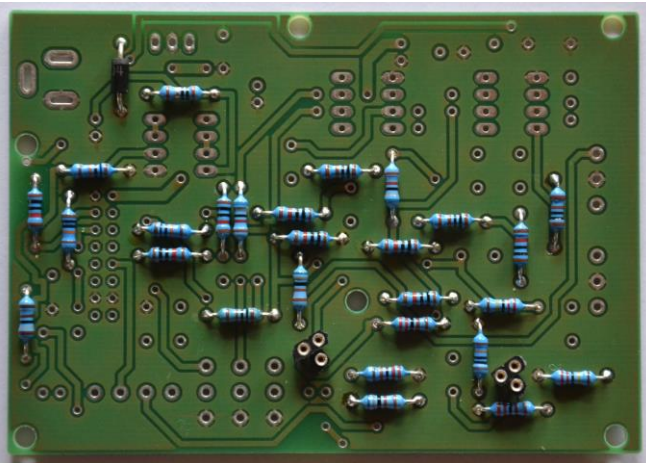
**Step 11:**



1N4004 diode (D2) ! Warning, take note of polarity

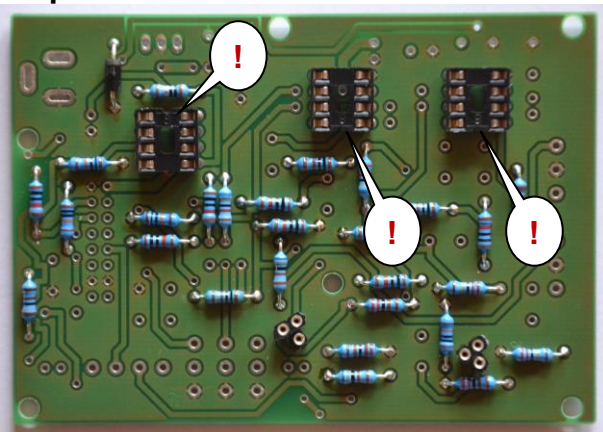


**Step 12: (optional)**

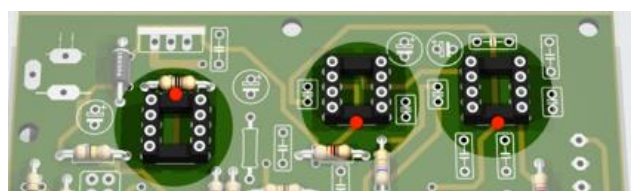
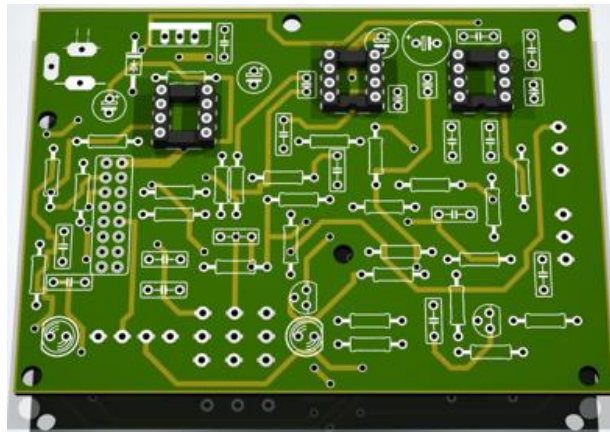


Install 2x sockets for the JFET's (optional.)  
We recommend to use the SIL connectors,  
part of the kit to create this

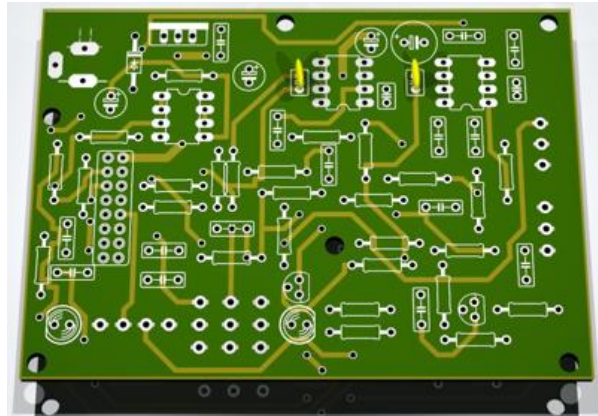
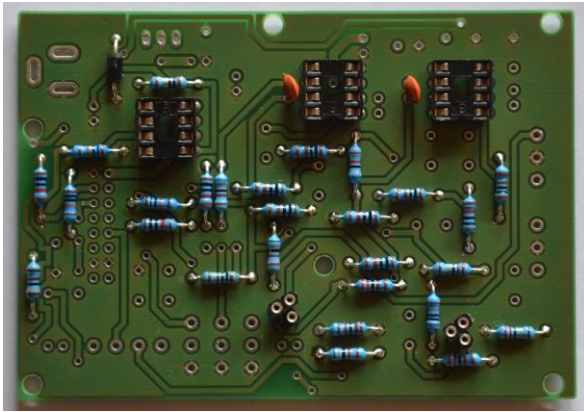
**Step 13:**



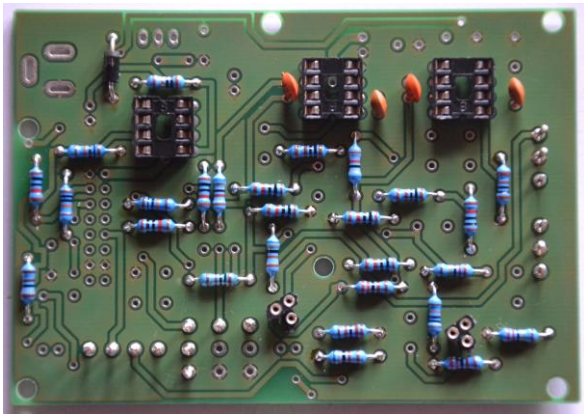
Install the 3 DIL sockets.  
! Watch orientation, on the picture on the  
right, the orientation is given by red dots.



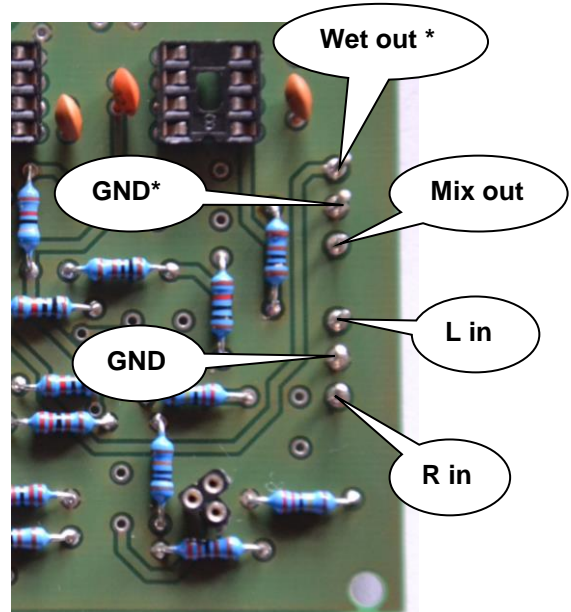
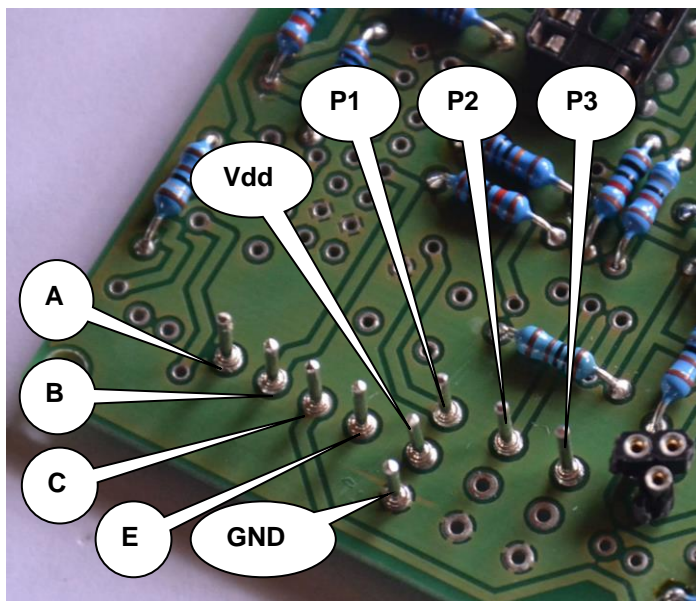


**Step 14:**

Install 2x 120pF ceramic capacitors (C11,C12)

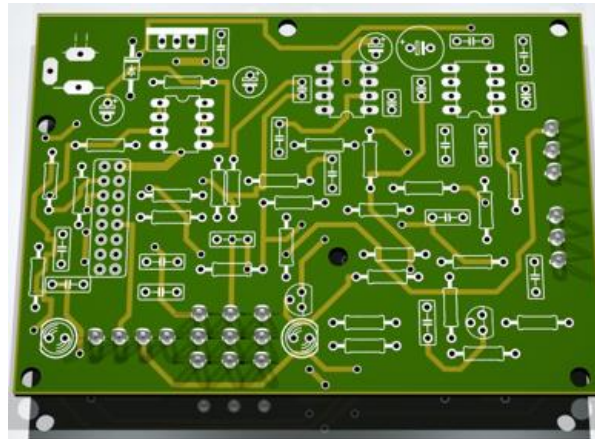
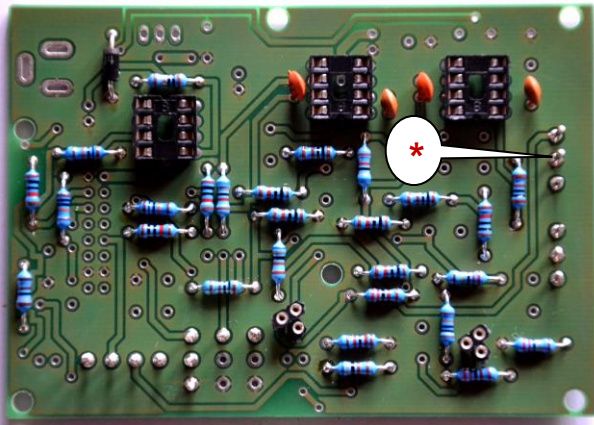
**Step 15:**

Install 2x 220pF ceramic capacitors (C2,C7)

**Step 16a:** Study connector pinout, not all solder pins are required



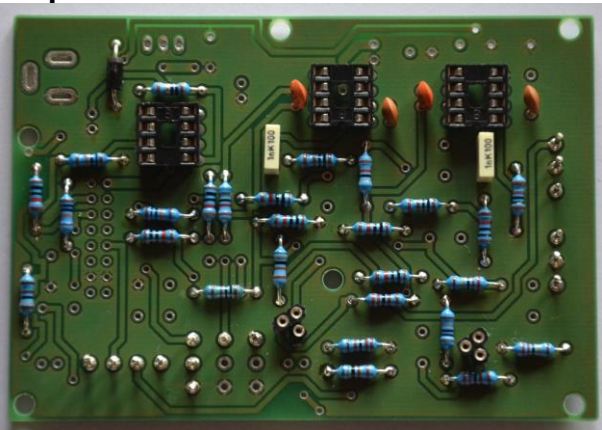
**Step 16b:**



Insert all 14 to 15 solder pins.

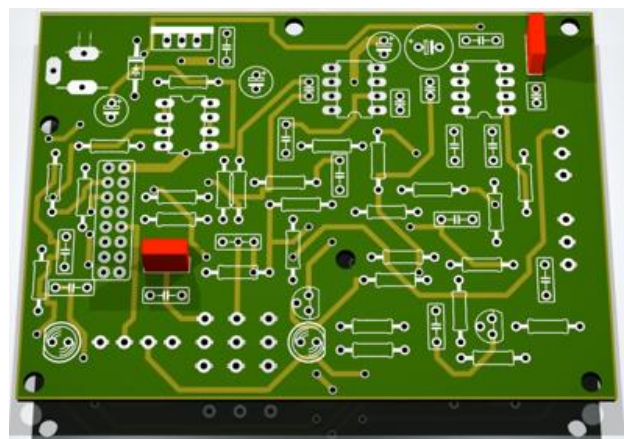
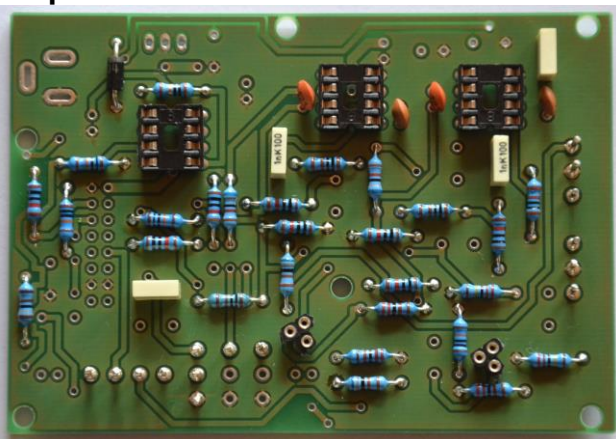
\* One GND connection is sufficient. A second GND pin above is not required.

**Step 17:**



Install 2x 1nF mkt capacitor (C1, C6)

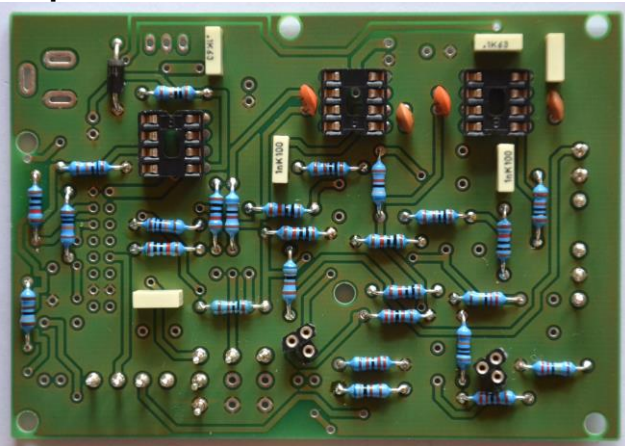
**Step 18:**



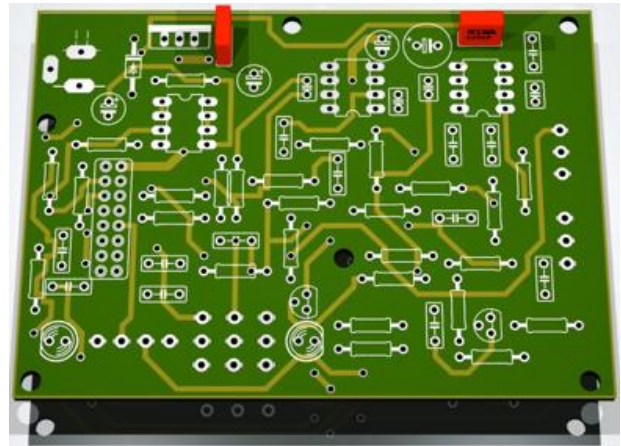
Install 2x 15nF mkt capacitor (C16, C17)



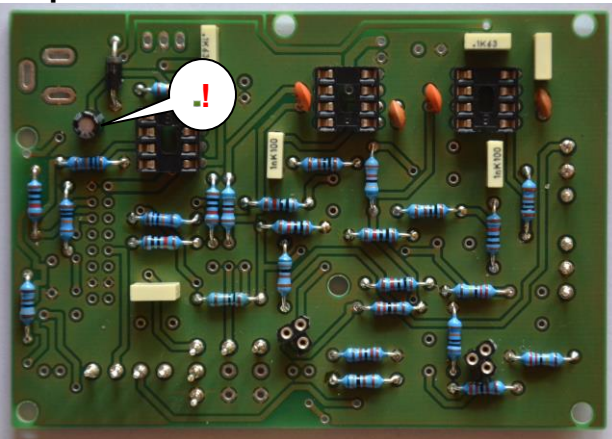
**Step 19:**



Install 2x 100nF mkt capacitor (C22, C23)



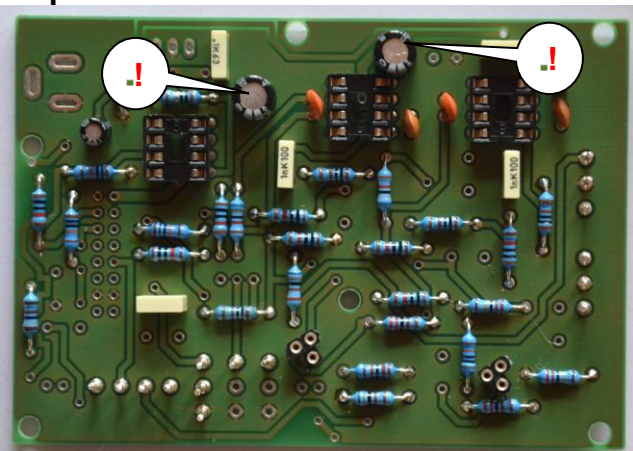
**Step 20:**



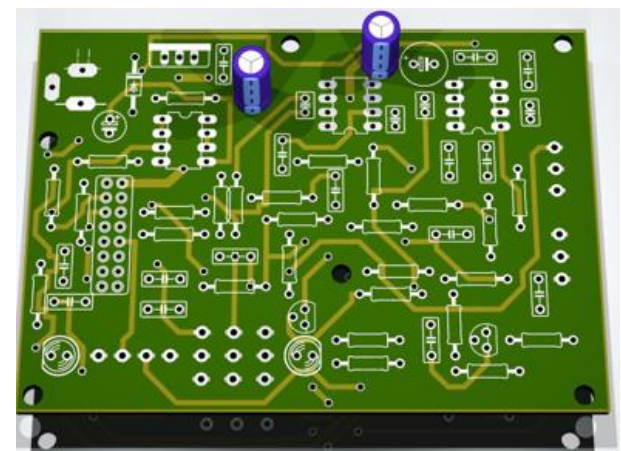
Install 1x 10  $\mu$ F elco (C19)  
! warning, polarity markings + up, - down



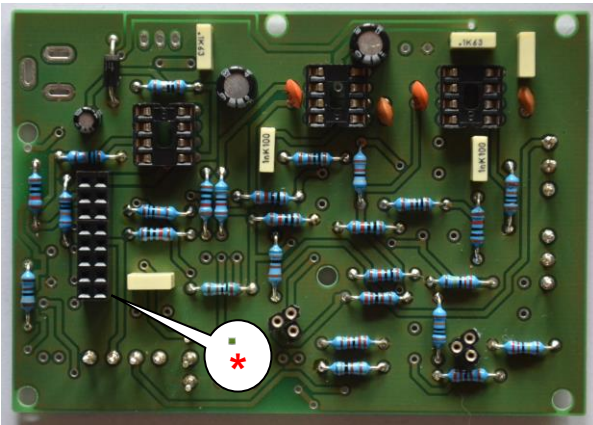
**Step 21:**



Install 2x 100  $\mu$ F elco (C18, C21)  
! Warning, polarity markings + up, - down.

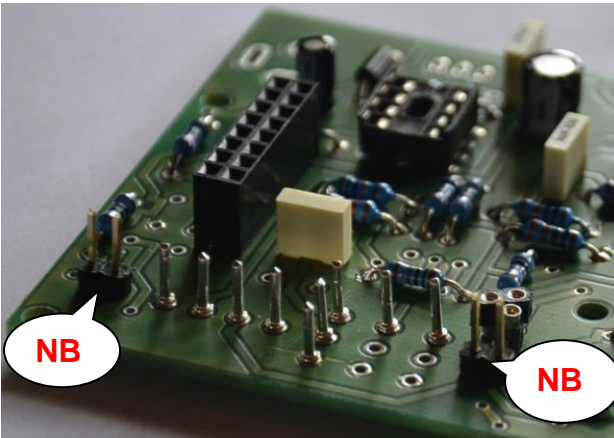




**Step 22:**

Install 1x Female header 2x8 pins.

\* The best method is to insert the header and solder diagonally the outer pins, then check if the header is flush with the board. If not heat-up soldered area and apply some pressure to the header to reseat correctly. Then solder all other pins.

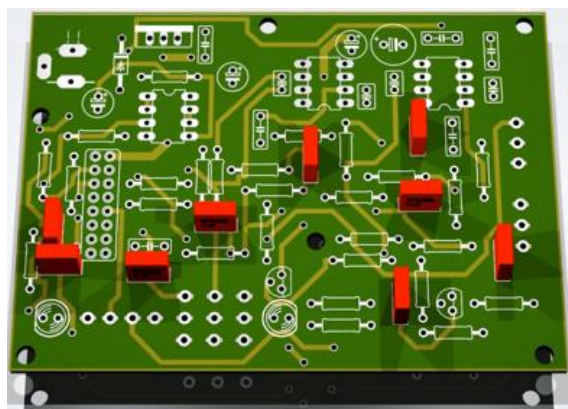
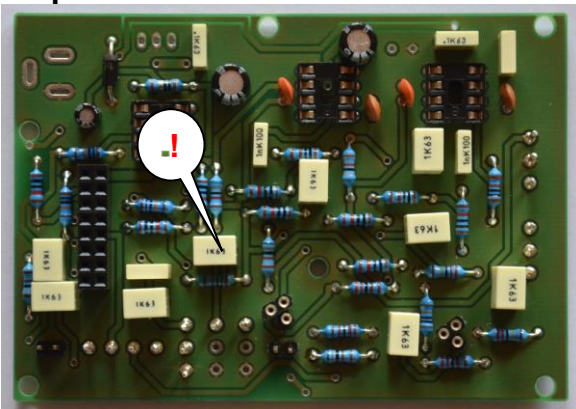
**Step 23: (optional)**

Install 2x Male header 2-pins for the LEDs

**NB:** These parts are not part of the kit! Chapter 4.1 '*indication leds*' provides an alternative so this step is possibly not required.

If required, the male 1x20 header can be ordered at Newtone-online:

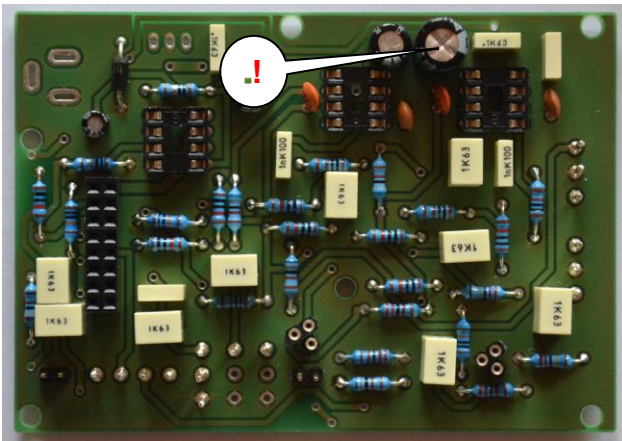
Product link: [http://www.newtone-online.nl/catalog/product\\_info.php?products\\_id=1648](http://www.newtone-online.nl/catalog/product_info.php?products_id=1648)

**Step 24:**

Install 9x 1 $\mu$ F mkt capacitor. (C3, C4, C5, C8, C9, C10, C11, C13, C15)

! Warning Via under component.



**Step 25:**

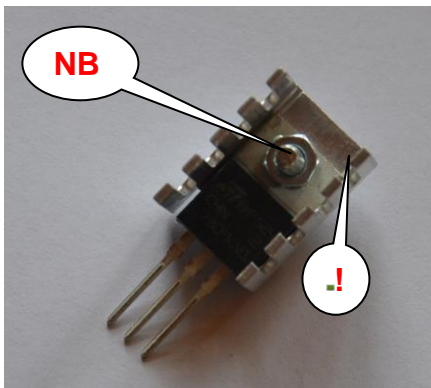
Install 470  $\mu$ F elco (C20)

! Warning, polarity markings + left, - right

Apply some pressure while soldering to assure component is flush with board

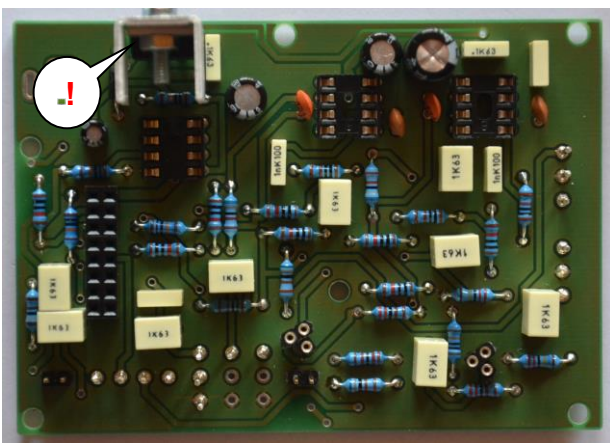
**Step 26a:**

Install heatsink on 7809 9V regulator before entering step 26b.



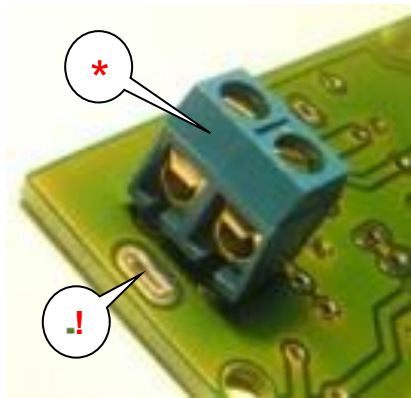
! The heatsink is slightly larger than the voltage regulator and requires to be trimmed. It is recommended to mount the heatsink on the regulator and insert into board before soldering it to check clearances.

**NB:** The M3 screw, nut and washer are not part of the kit and can be obtained from hardware stores.

**Step 26b:**

Install the 7809 regulator. (IC5)

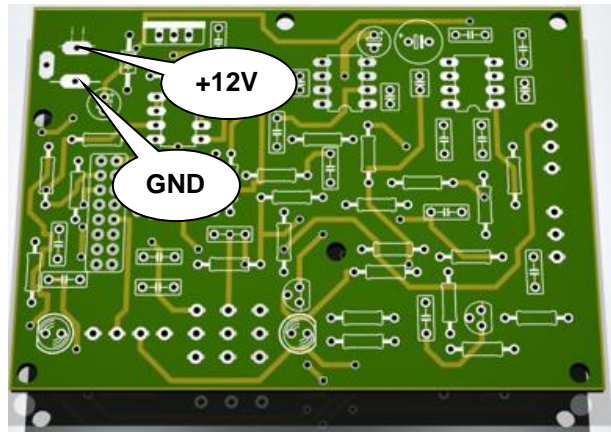
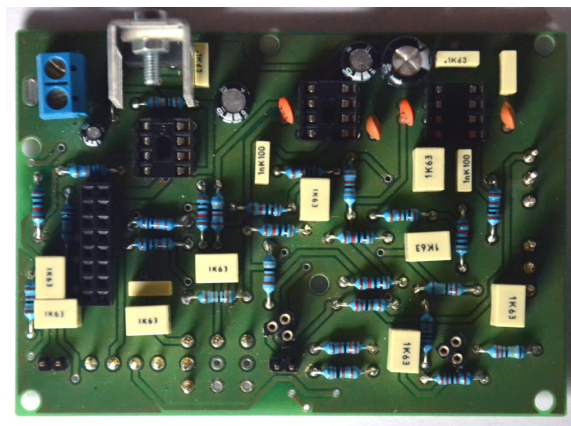
! Do not cut the regulator's legs too short as you need extra solder area to assure a perfect solder joint.

**Step 27a:**

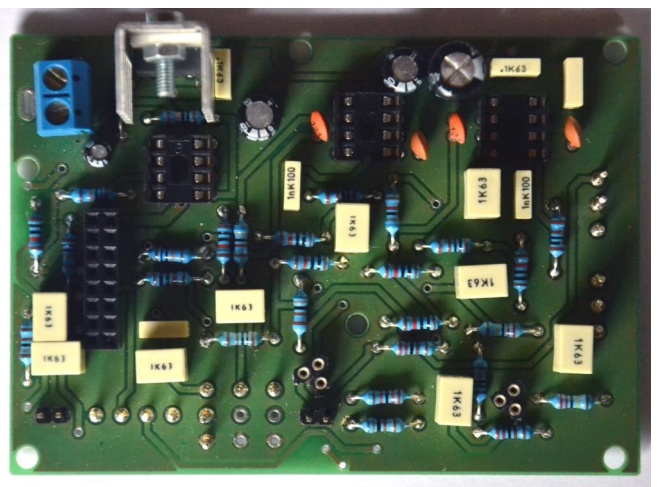
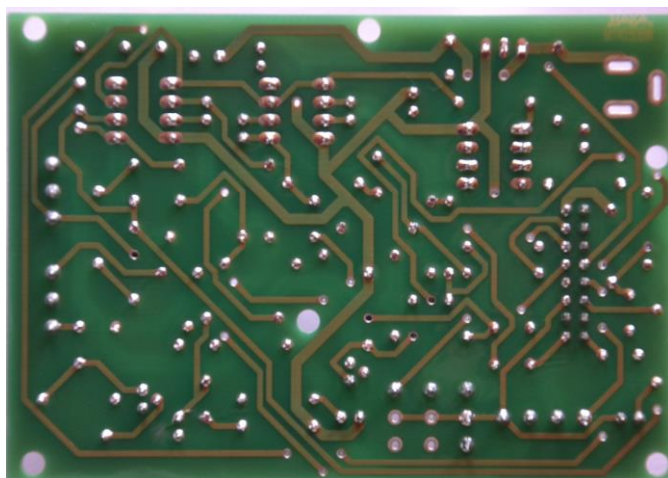
The version 1 board connector requires a small rework to fit the board. This is due to tolerances. Slightly bend the two pins out, ~0.5 mm is sufficient to fit the connector. A small clearance with the board of less than 1mm is acceptable.

! Version 1 boards are recognisable by elongated holes while the version 2 are with round holes.

\* See Chapter 4.4 when a different supply is used.

**Step 27b:**

Install the 12V connector block

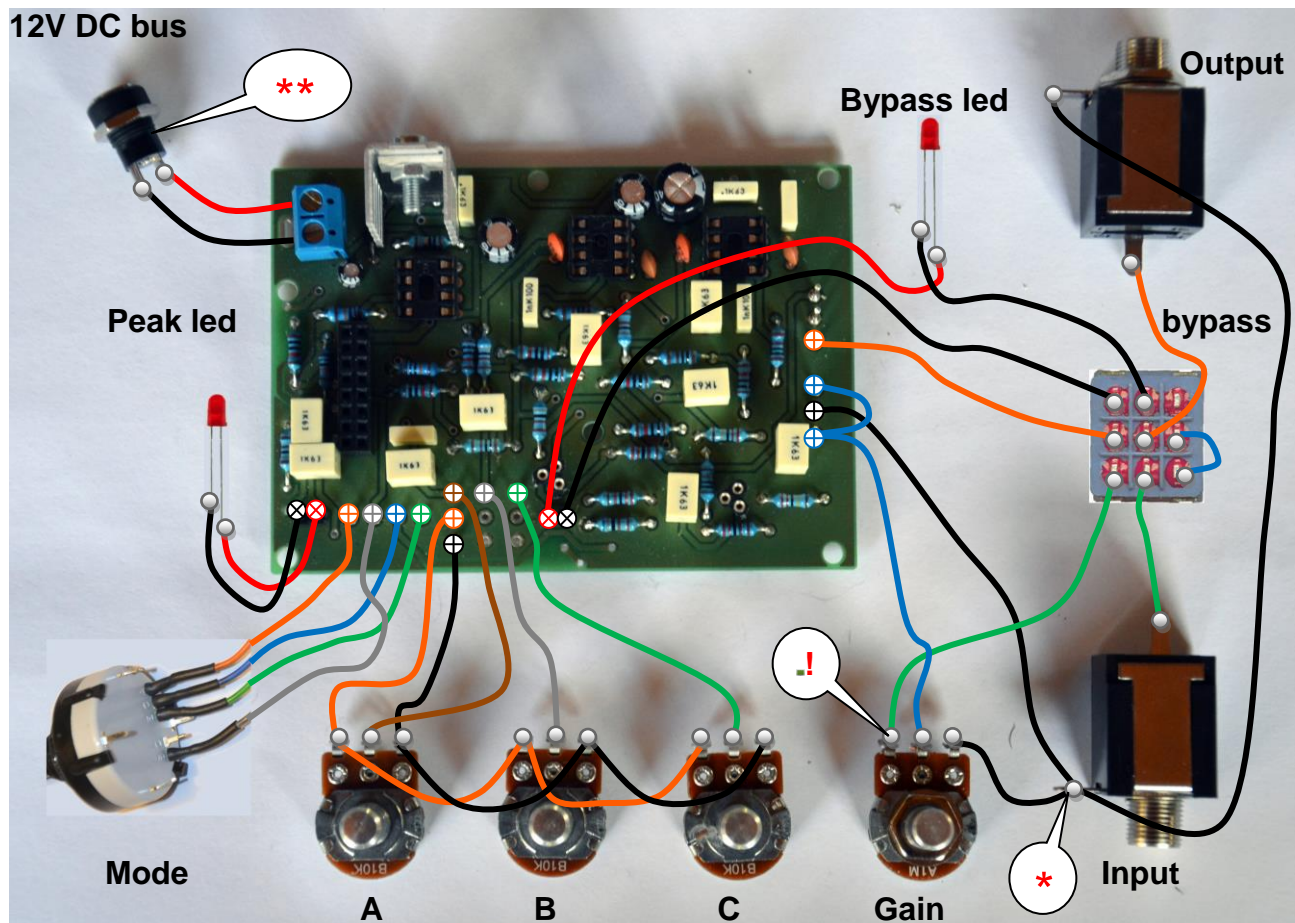
**Step 28:**

The final result of your work should look like this. Use a magnifying glass to check each area for short circuits. Recheck exact placement of all components.



## 2.3 Wiring

The picture below shows all required wiring that will be discussed step-by-step. Firstly, keep all wires to 25cm so they can be shorted later-on. It is also possible to use the drilling information of your enclosure to pre-cut the wires at exact length.



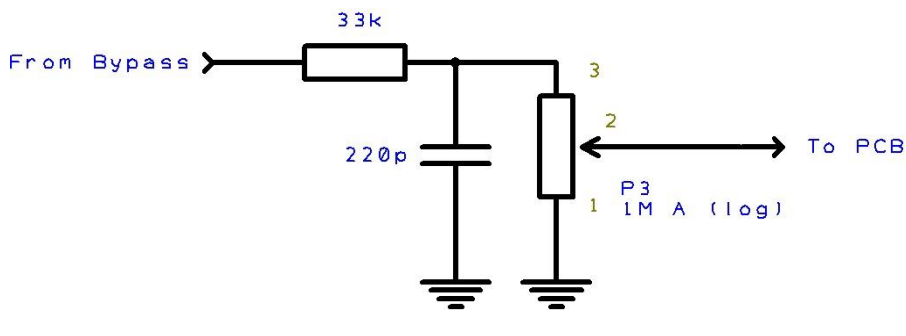
! The gain (level) potentiometer will have its resistor and capacitor installed later in the process. (See wiring: step 1a and 1b)

\* GND connections on input jack sleeve.

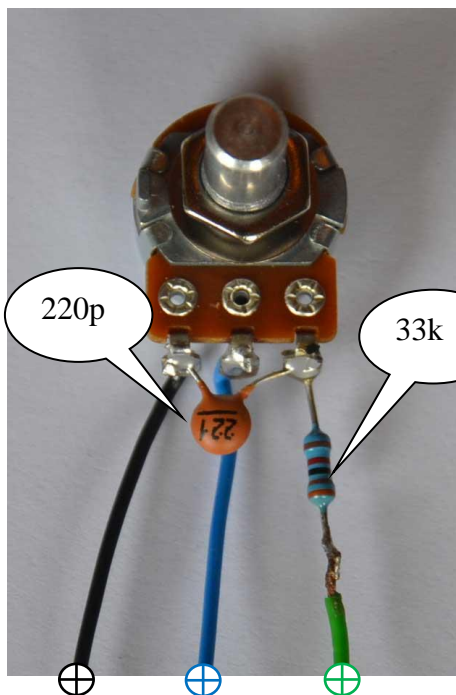
\*\* DC bus wiring is made for a positive tip adapter (See also wiring step 4 and chapter 4.5)

⊕ Connections made with solder pins..

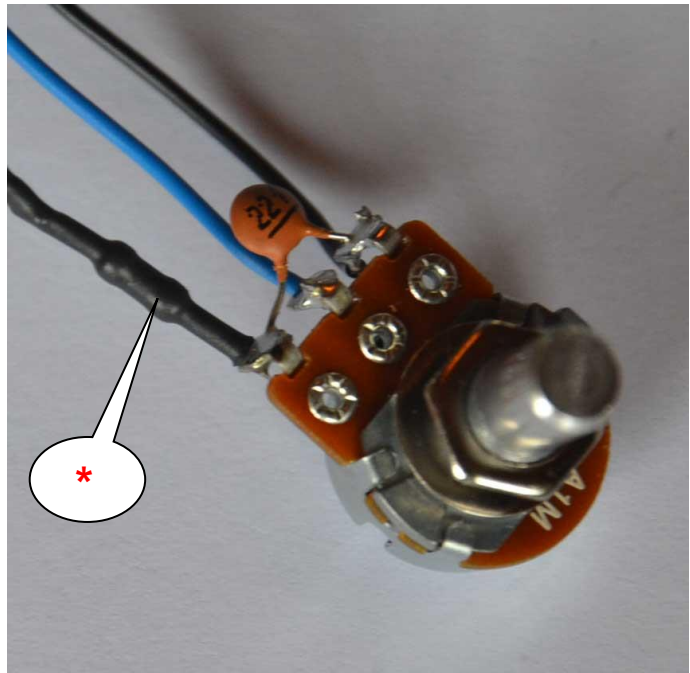
○ Normal soldered connections

**Step 1a: Preparation of the 1M gain potentiometer.**

Picture above shows the gain/level input configuration required to connect the guitar to the eTap2hw module potentiometer.

**Step 1b:**

GND PCB in L+R From Input

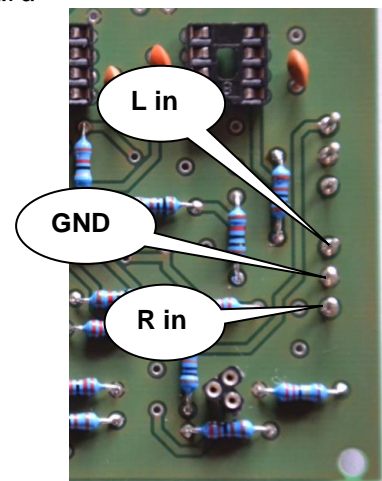
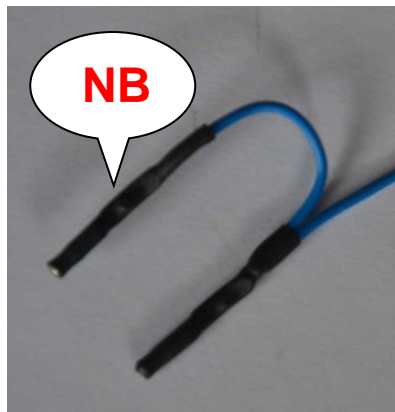
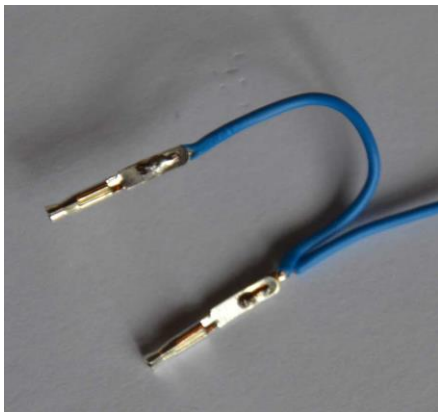


Assembly point-to-point

- Solder blue wire to pin 2 of the 1M potentiometer
- Solder the black wire and the 220pf capacitor to pin 1 of the 1 M potentiometer
- Solder the other side of the 220pf capacitor and the 33K resistor to pin 3 of the 1 M potentiometer
- Solder the green wire on the 33K resistor

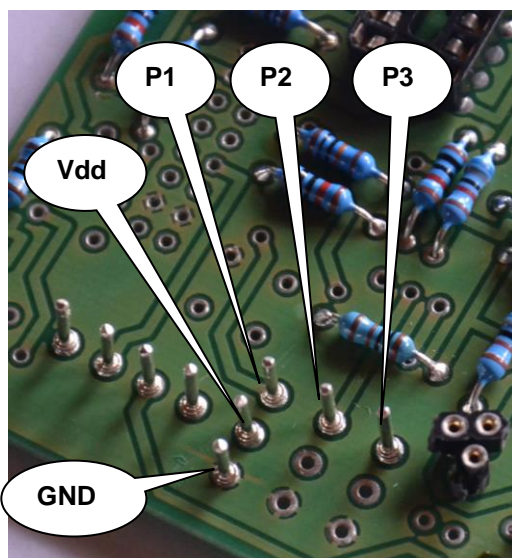
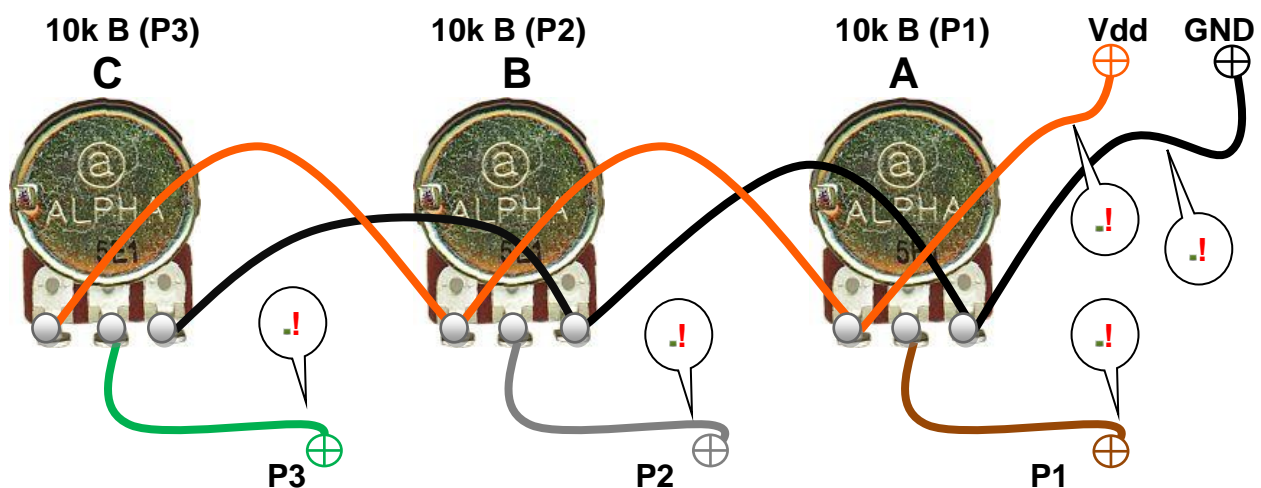
\* It is recommended to apply a heat-shrink sleeve over the 33K resistor and solder-join to provide insulation and also extra mechanical strength.



**Step 1c: Preparation of the gain potentiometer on motherboard**

The blue wire requires to be split into two parts to allow connection to the 2 input channels (*L in* and *R in*) on the motherboard.

**NB** Usage of heat-shrink tubing is highly advisable.

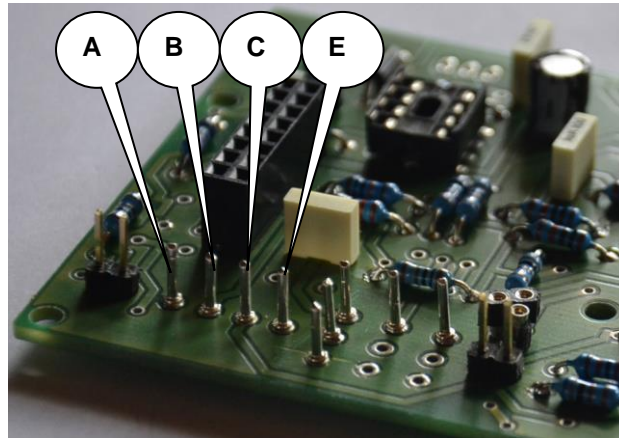
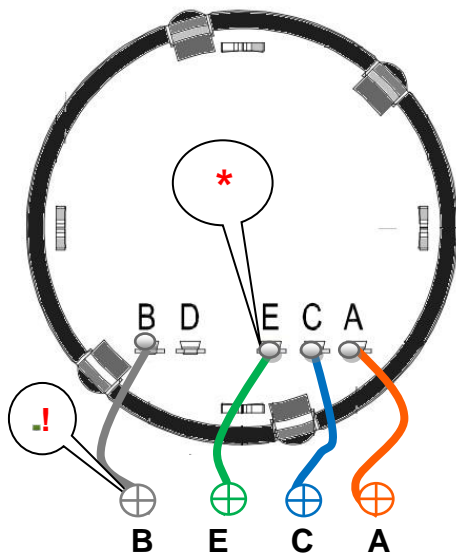
**Step 2: Preparations of A, B and C potentiometers**

3x 10k B (lin) potentiometer (P0, P1, P2)

Join 3x pin 3.

Join 3x pin 1.

! Assure sufficient length is kept to later connect those wires to the motherboard. Those wires will be soldered to the connectors for the solder pins.

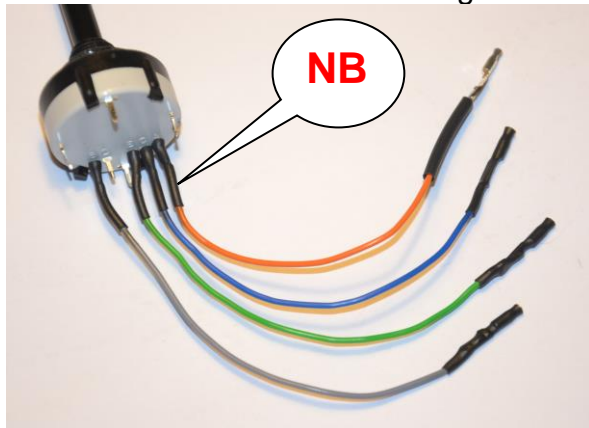
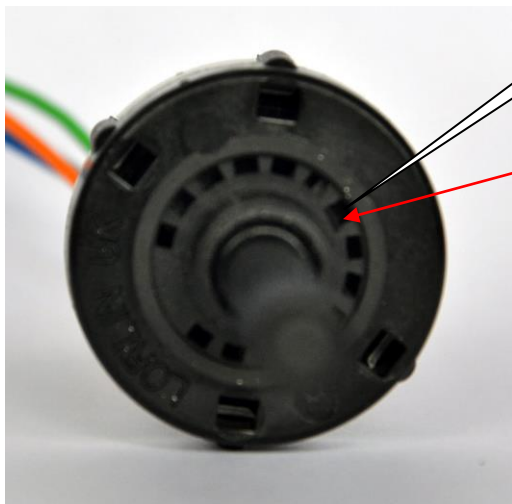
**Step 3a: Preparation of BCD-switch**

Solder the four wires to the BCD switch and give them solder-pin connectors.

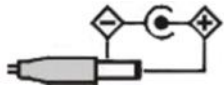
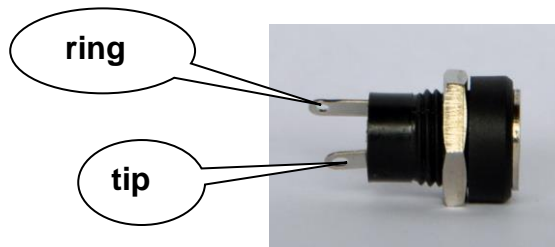
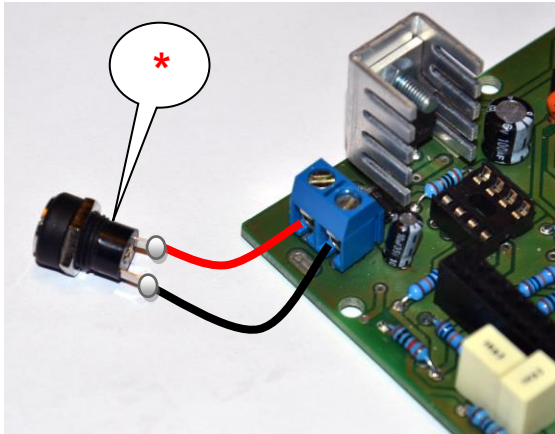
! Keep sufficient length to make connections to the PCB in the housing

\* It is easiest when soldering that wire and pin are tinned well in advance.

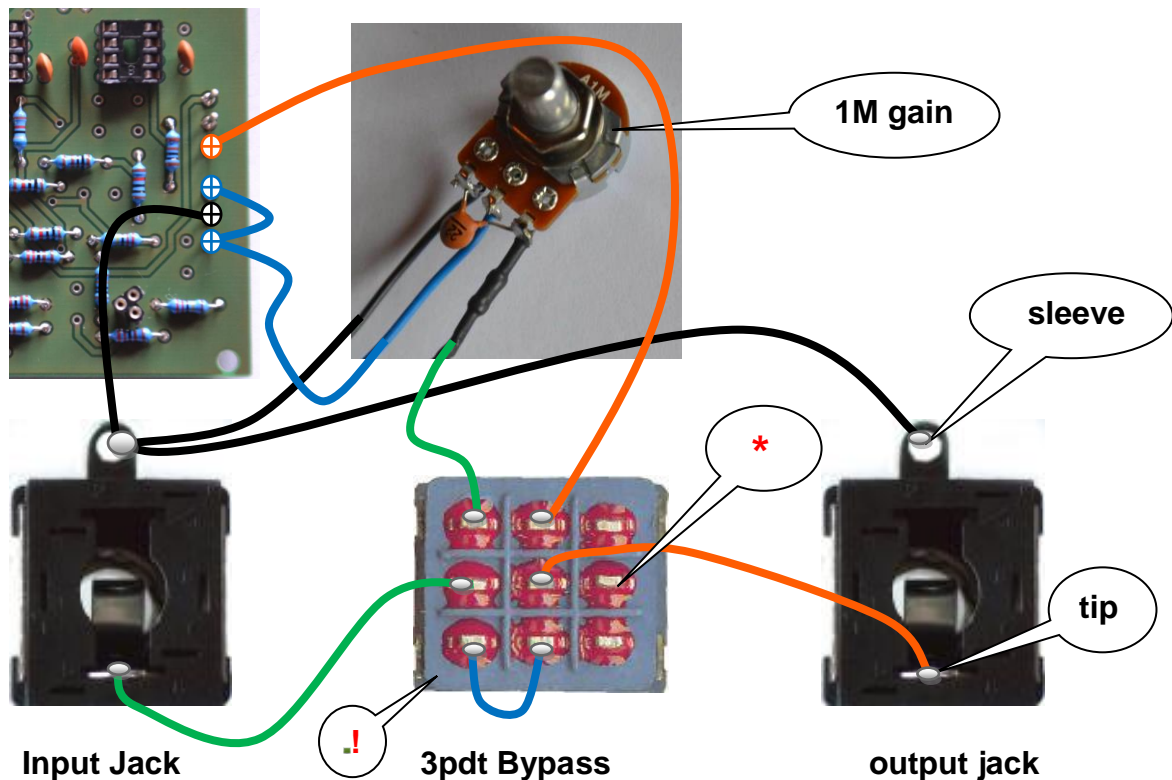
**NB** The use of heat shrink tubing is recommended.

**Step 3b: The Switch stop Positioning**

\* There are only 8 (of 16) BCD switch positions needed. Under the nut is a stopper, a ring with a hole in which a pin can fall. To limit this to 8 positions, turn the switch first fully counter clockwise and then insert the stopper in the 7th (!) Hole.

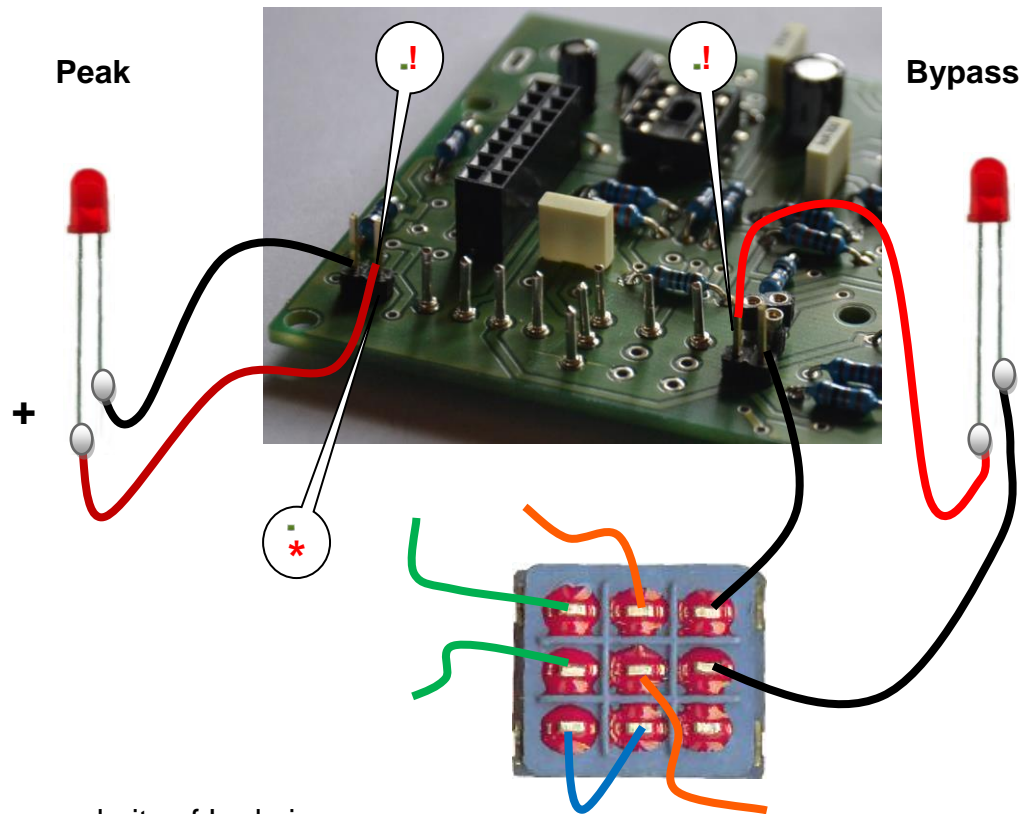
**Step 4 : Preparation of the DC Bus**

\* **12V DC** Utilises a positive tip connection. See section 4.4 for alternatives

**Step 5: Connect instrument jacks and true bypass switch**

! Ensure correct orientation of the 3pdt footswitch.

\* This row left free for the bypass indicator LED. See next step.

**Step 6:** Wiring *bypass led* and *peak led*.

! Observe polarity of Led pins.

The inner pins are positive and with (red) are connected to the longest legs of the leds.

\*You can also wire the leds directly, without soldering on the PCB connector.

See section 4.1 for use of alternative leds and/or connectors.



# Chapter 3 Testing and Final build

## 3.1 Testen van de unit

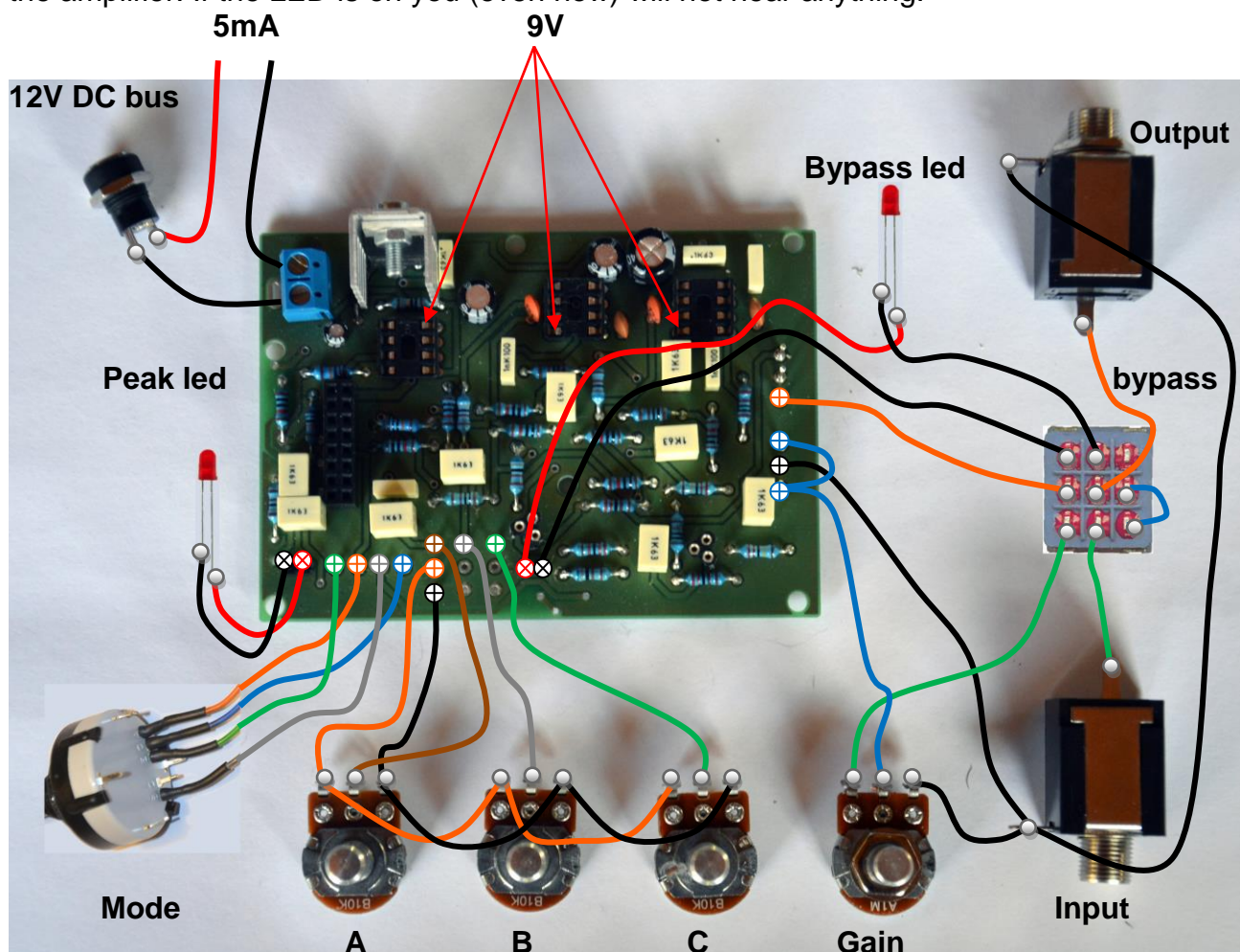
- Put no-ICs, FETs or SKRM daughter board in the sockets.
- There is a need to check current and voltages using a Multimeter.
- There is a sound source (guitar) and an amplifier needed. You can also use an oscilloscope and. signal generator

**Test 1:** Connect a 12V DC adapter, but leave the positive connection (red) open and measure the current between the DC bus and the connector block. This should measure around 5mA.

**Test 2:** Connect the +12 V now directly to the connector block and then check 9V voltage at the points indicated. (3x pin 8 of the IC sockets)

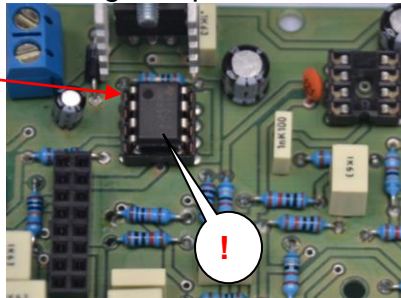
**Test 3:** Connect the bypass switch, jack sockets, gain potentiometer and indicator LED to the bypass switch. Connect an instrument to the input jack and connect the output to an amplifier.

You can now check the bypass. If the LED is off, you should hear the instrument through the amplifier. If the LED is on you (even now) will not hear anything.



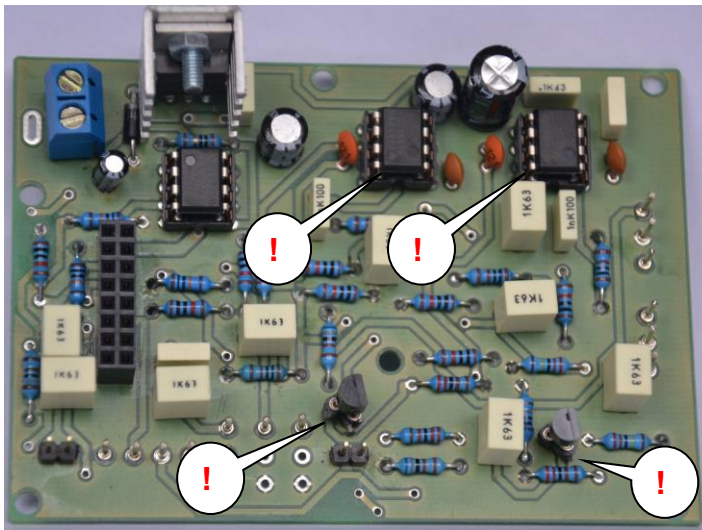
**Test 4:** Insert IC 3 and check the voltage on pin 1 is 4.5V

4,5V



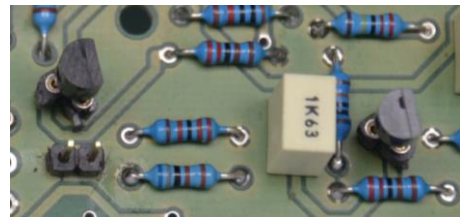
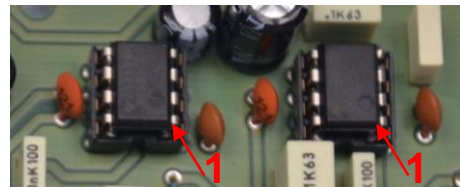
! Note the correct orientation of the IC. Pin 1 in the upper left corner.

**Test 5:**



Install remaining 2x NE5532 IC's (IC2, IC3)

Install 2x J201 FET.

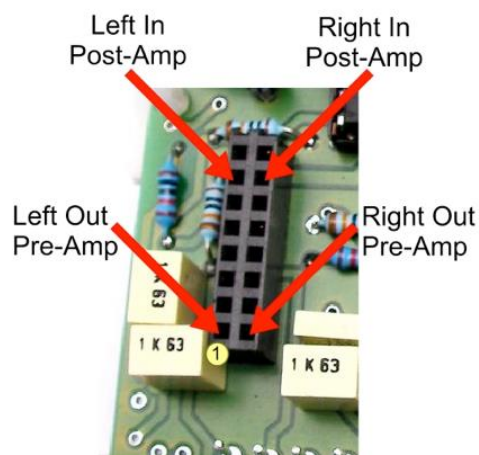


Repeat the current measurement of test 1 and you should now measure around 25mA.

! Note correct orientation

Later you need to cut the legs of the FETs relatively short to fit under the SKRM daughterboard.

**Test 6a:** Test of the pre- and post-amp



Before installing the daughterboard the preamp should be tested. You need to connect the terminals indicated by the two jumpers. These are easy to make by tinning, e.g. a stripped cable.

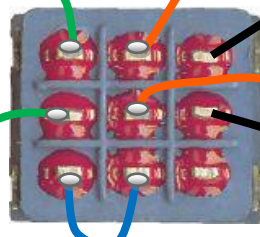
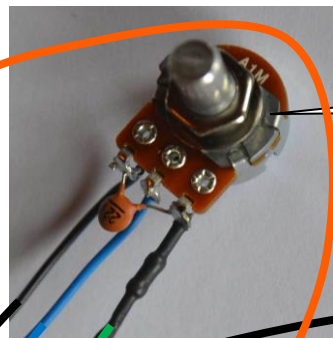
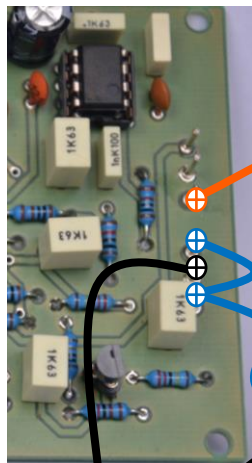
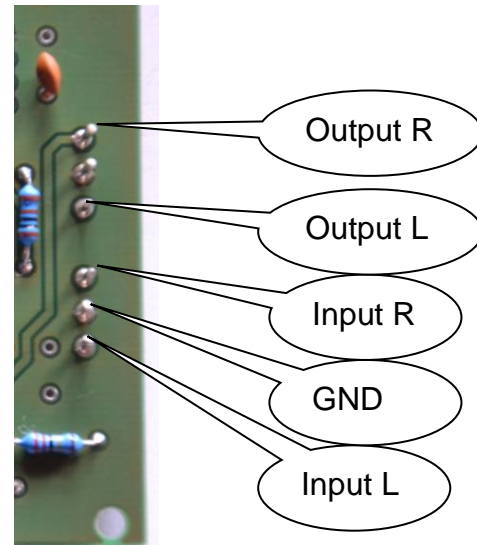
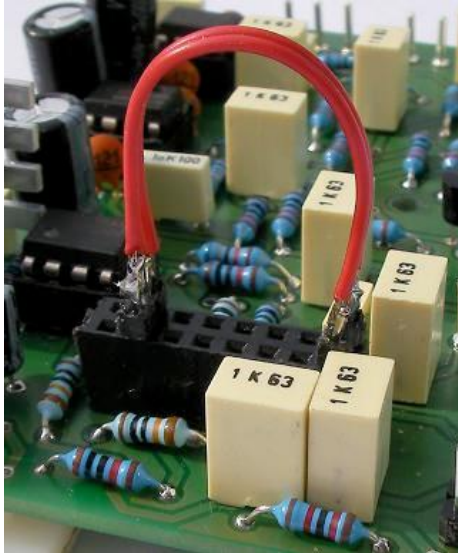




**Test 6b:** Testing of the pre- and post-amp

Make the jumper links as shown and connect the adapter.

Connect an instrument (or functiongenerator ca 100mV) to both inputs (Input) and an amplifier (or scope) on the left output (Output L)

**Input Jack****3pdt Bypass****(led)****output jack**

-Check the bypass signal and the signal with the effect on.

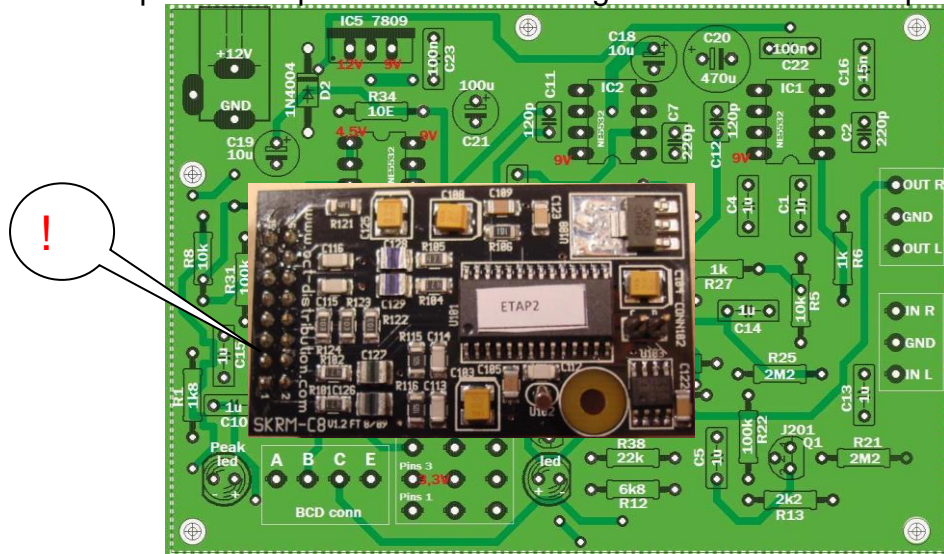
-As you turn up the gain-potentiometer with the effect on the signal will give an audible boost. (On an oscilloscope with maximum gain employed distortion will be visible.)

-The place the orange wire on the top output to test the right channel (Output R). After the test put it back on output L.

-Check that the indicator LED is lit.

**Test 7:**

Turn off the power and place the SKRM daughterboard in the 2x8-pin socket.



Set all potentiometers in the middle, and the BCD switch to the first position.

Check that the echo is working correctly.

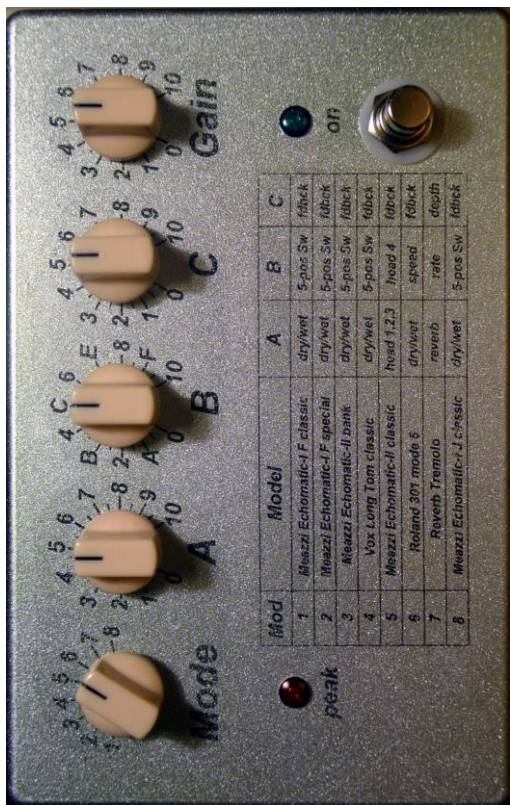
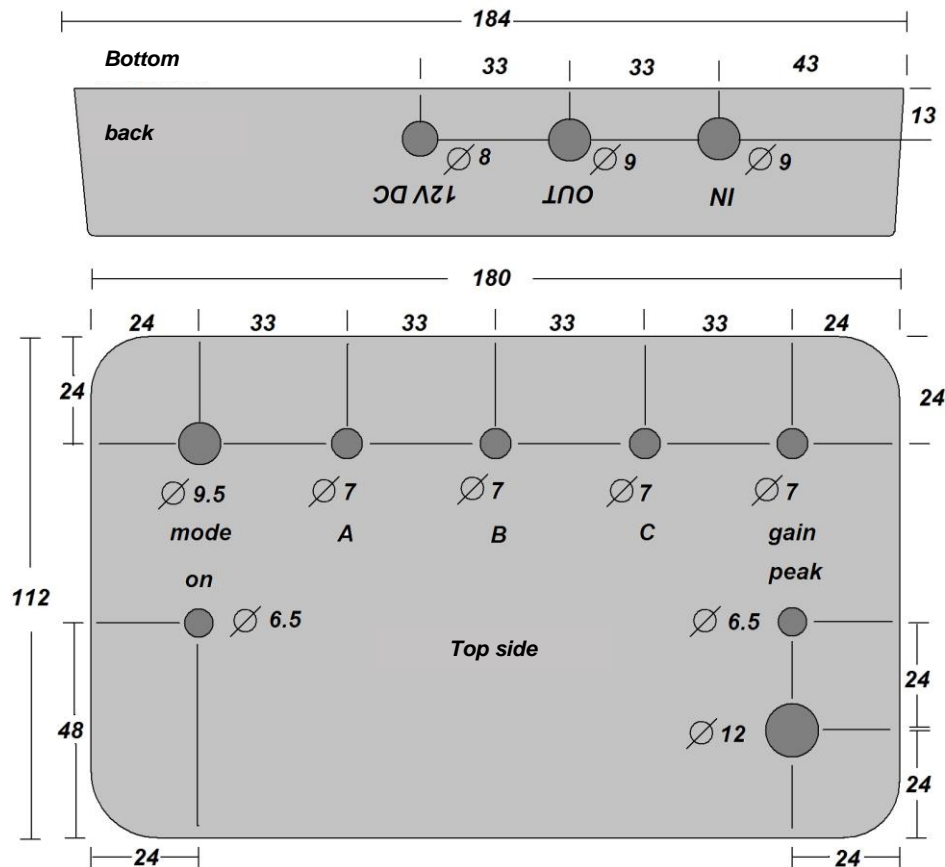
**! Take care when handling the daughter board and use the anti-static measures (Chapter 1.1).**

Now everything works the effect can be built into an enclosure.

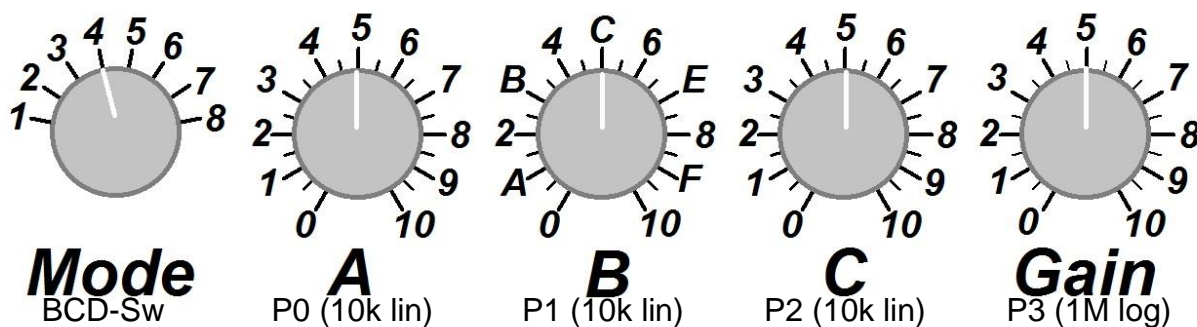


3.2 Drilling plan

Shown here, is an example for an enclosure. All measurements are in mm and based on the use of an aluminium box, Type DD. You will find a full size decal image on page 37 of this PDF manual.



### 3.3 Functions of potentiometers and bcd-switch



Mode (bcd)	Model <i>emulation</i>	A (P0)	B (P1)	C (P2)
1	Meazzi Echomatic-I model 'F' Classic	dry/wet	5 pos. Sw	feedback
2	Meazzi Echomatic-I model 'F' Special	dry/wet	5 pos. Sw	feedback
3	Meazzi Echomatic-II Bank	dry/wet	5 pos. Sw	feedback
4	Vox LongTom Classic	dry/wet	5 pos. Sw	feedback
5	Meazzi Echomatic-II Classic	dry/wet head 1/2/3	dry/wet head 4	feedback
6	Roland model 301 Mode 5	dry/wet	varispeed	feedback
7	Reverb en/of Tremolo	reverb mix	tremolo rate	Tremolo depth
8	Meazzi Echomatic-I model 'J' Classic	dry/wet	5 pos. Sw	feedback

**Mode** A three 'bits' code is generated by switching to one of the first eight positions of the BCD switch and used to select an Echo Emulation. This (Lorlin switch) starts with the digital one (001) and position 8 is then (000).

**A** This potentiometer determines in all cases the Dry / Wet mix, i.e. The extent to which the raw signal is mixed with the delayed signal or reverb..

**B** This potentiometer works in most cases as a 5-way switch, as is also found on the original echo units. The positions are always located at the centre of rotation of the potentiometer 20% increments. Only the positions A, B, C, E and F are emulated as D mode adds little. See also section 4.3..

**C** This potentiometer controls the amount of feedback from all but the tremolo emulation (Tremolo Depth in that case). The echo is attenuated and returned to the input to give several repetitions.

**Gain** This potentiometer is used to adjust either for the FET triode emulation 'sweet spot' of the circuit, or for maximum signal. Input volume: Please keep the 'Peak' LED in mind. If this LED is lit you are introducing unwanted distortion that should be avoided. See also section 4.2.



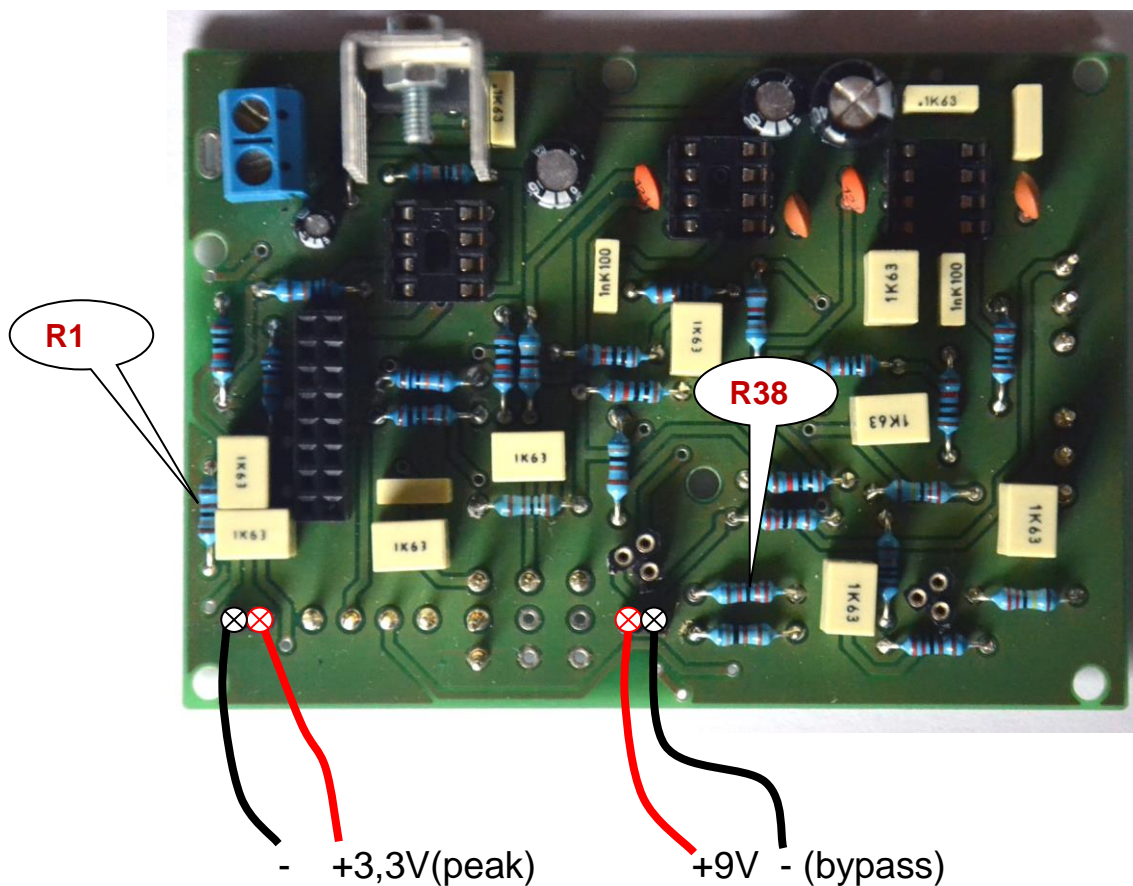
# Chapter 4

# Modifications

## 4.1 LEDs

You can also solder wires for the LEDs directly to the PCB. It will be easier if the studs are first tinned. If you want to use other 2-pin connectors they should be of 2.54mm spacing.

In this kit bright white LEDs are supplied. You can also use other LEDs, if this is the case, the resistor, R1 (1k8) needs to be changed for the 'Peak' LED and also the resistor R38 (22k) for the 'bypass indicator'-LED.



Each LED has a nominal voltage  $U_f$ . For one approach to this voltage see the table below:

Colour	$U_f$ 3mm led	$U_f$ 5mm led
Red	2,2V	2,2V
Yellow	2,1V	2,2V
Green	3,4V	3,5V
Blue	3,5V	3,5V
White	3,5V	3,4V

*With a diode tester the actual voltage drop can be measured.*

**Adjusting R1:**

Voltage at the peak indicator LED is 3.3 V ( $U_s$ ). It is important to know that  $U_f$  of the LED used is lower than 3.3V otherwise there is no power output! The above table shows that only the red and yellow LEDs are standard fit at this point. The 'bright' LEDs usually have a lower voltage drop and can then be used.

It is also important to know how much current / amps (I) it takes the LED to light up sufficiently. This differs per LED type and is often between 0.5mA and 20mA. Too much current also shortens the life of the LED.

Suppose we take a red LED with an  $U_f$  of 2.2V. These are bright enough with 5mA. The calculation for R1 is as follows:

$$R1 = \frac{U_s - U_f}{I}$$

$$R1 = \frac{3,3V - 2,2V}{0,005A}$$

$$R1 = 220 \text{ ohm.}$$

**Adjusting R38:**

This is the same as with R1, except, the supply voltage ( $U_s$ ) is 9V here. Here, the use of any LED is possible.

Same red LED with  $U_f$  2.2 V gives the following series resistor at 5mA::

$$R38 = \frac{9V - 2,2V}{0,005A}$$

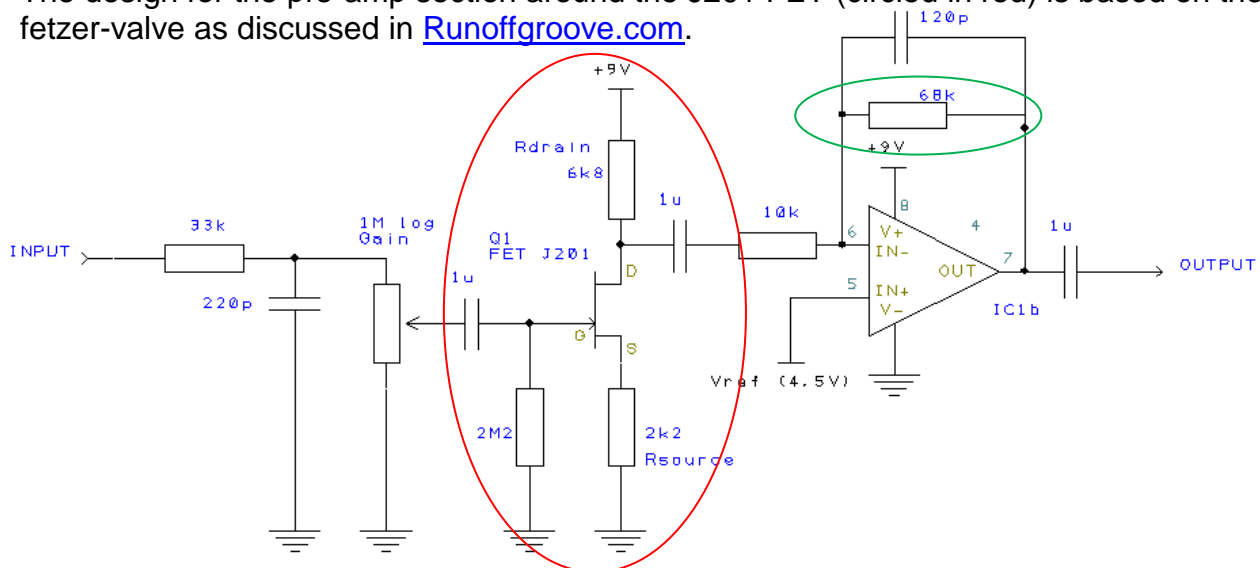
$$R38 = 1500 \text{ ohm (1.5k)}$$

A useful tool is an [online led calculator](#).



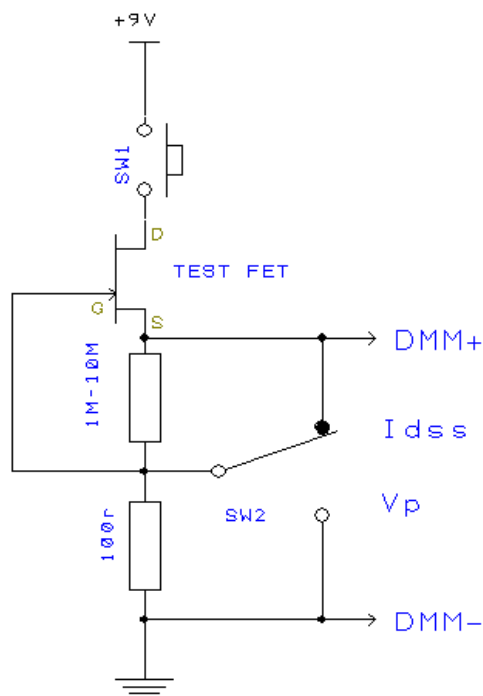
## 4.2 FET installation in a nutshell

The design for the pre-amp section around the J201 FET (circled in red) is based on the fetzer-valve as discussed in [Runoffgroove.com](http://Runoffgroove.com).



This FET circuit is intended to create harmonics which are similar to those produced by triode tubes and to provide some gain. This is followed by an inverting op-amp circuit with a gain of 6.8 (16.8 dB) to give what is a good S / N ratio signal of around 3Vpp to the FV-1 board. A well set up J201 has a gain around 4.8 (13.6 dB) and can handle an average of 1.5 Vpp. Coupled with the op-amp this can produce more than enough gain to develop 3Vpp. If it is difficult to produce a sweet sound from the FET by use of the Gain potentiometer then it may be that the output is too high (look at the peak LED). In that case, one can decrease the 68k resistors (R10, R18 circled in green above) of the opamps slightly in order to limit gain to an acceptable value.

- FETs of the same species have differences in the pinch-off voltage  $V_p$  and the maximum current  $I_{dss}$ . This makes for fairly unpredictable results. The supplied drain and source resistances in the test set generally give satisfactory results, but for an optimal setting these can be chosen based on FET measurements.



To measure  $V_p$  and  $I_{dss}$  of a particular FET this simple test can be used with voltages read by a digital multimeter (DMM). (Sw1 is optional):

1 Connect a DC voltmeter to + DMM and DMM-

2 Put Sw2 to  $I_{dss}$  measurement. Multiply the measured voltage by 10 to give  $I_{dss}$  in mA.

3 Put Sw2 to  $V_p$  measurement. The measured voltage is  $V_p$

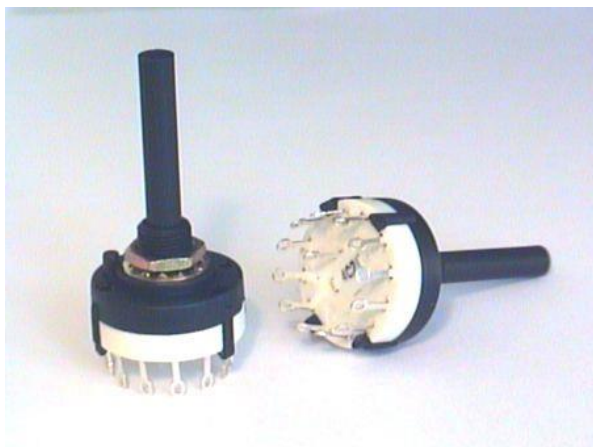
Now we know the voltage,  $V_p$  and  $I_{dss}$  (mA) we can fit the ideal drain and source resistances ( $R_s$  and  $R_d$ ).

A useful tool is this [online fet calculator](http://onlinefetcalculator.com).

[Here](#) deeper information about FET biasing.

### 4.3 5-position rotary switch instead of potentiometer B (P2)

As stated in section 3.3, potentiometer B (P2) acts in most of the emulations as a 5-way switch, (A, B, C, E and F). It is possible to add a rotary switch to the potentiometer. Newtonone offer the Lorlin 1X12 range which can be used very well. A further two 1k resistors and four 2k resistors are required.



There are three exceptions to the 5-position switch setting for emulations 5, 6 and 7.

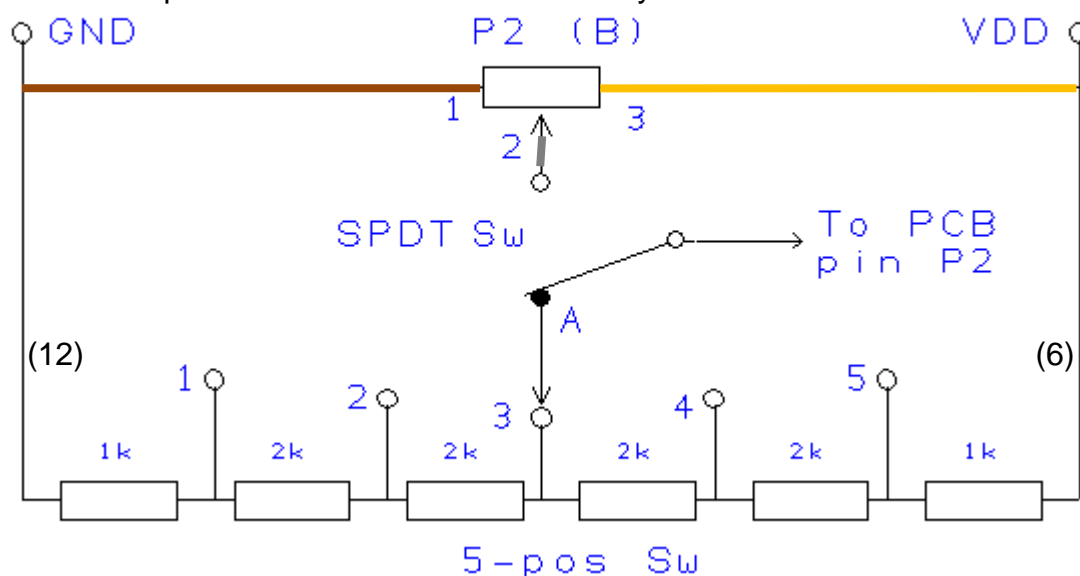
No. 5 sets P1 as Wet/Dry for Heads 1/2/3, P2 as volume for Head 4.

No. 6 sets P2 as a varispeed control.

No.7 sets P1 as Reverb Amount, P2 as Tremolo Speed and P3 as Tremolo depth.

It may suffice to have only 5 positions, but it is usually preferable to keep both options. In that case an additional SPDT switch is required. The SKRM FV-1 identifies the 5 positions by reading the voltage at 10%, 30%, 50%, 70% and 90% positions of the pot rotation (P2).

Below is a possible schematic to add a 5-way switch:



-A 5-way Lorlin indexed rotary switch can be used in conjunction with the 6 resistors shown in the schematic above. The outer 1k resistors are soldered between pins 1 and 12 and between pins 5 and 6. Rotation of the switch can continue until reaching the 5th position stop. To use in combination with a 10k linear potentiometer (P2) requires soldering as given below

-Pin 1 (GND / brown) of the potentiometer (P2) is soldered to pin 12 of the Lorlin rotary switch, and pin 3 (Vdd / orange) soldered to pin 6 of the rotary switch.

-Pin-A of the rotary switch goes to one of the outer pins of the SPDT switch.

-Pin 2 (wiper / grey) of the potentiometer (P2) is connected to the other outer pin of the SPDT switch.

-The middle pin (Common) of the SPDT switch now goes to the pin P2 connector on the PCB.

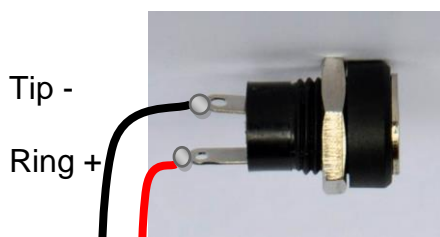


## 4.4 Modifications to the power supply.

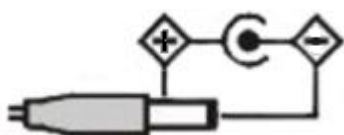
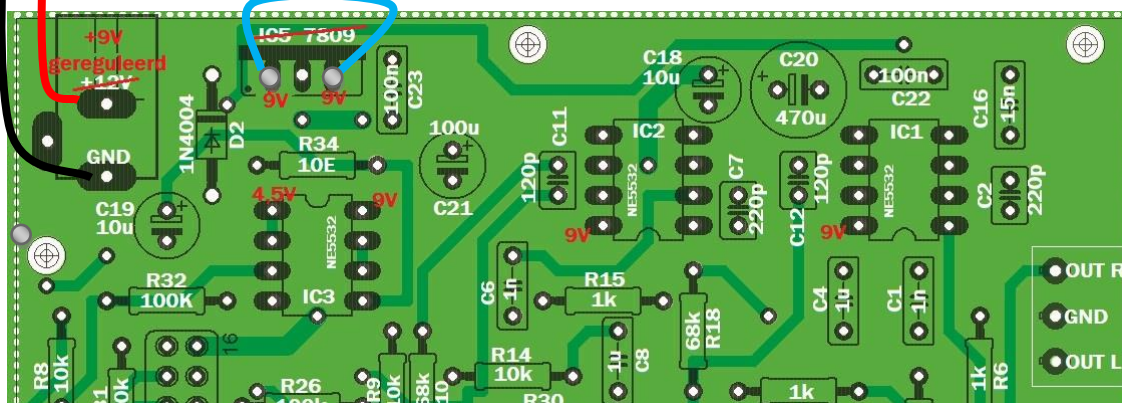
**Deze modificatie dient alleen bij voldoende kennis van zaken uitgevoerd te worden.**

The standard PCB is provided with a 7809 regulator for the 12V supply in order to convert to a stable 9V. (See also 4.5 'schedule') A regulated 9V voltage is important for the establishing the FET's bias and to limit any ripple that could be heard in the audio. The 7809 also provides some protection against high voltages on the [SKRM FV-1 daughterboard](#). The SKRM FV-1 daughter board has its own 3.3V regulator and can be powered with voltages between 5V and 12V.

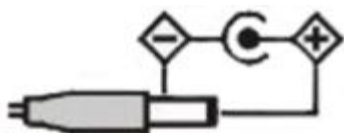
In the world of guitar effects, 9V adapters with positive ring are much more common than 12V adapters with negative ring and one can perform a modification to connect a **regulated** 9V (positive ring) adapter directly. Do **not** use unregulated power supplies because, although they are nominally 9V DC, they can often give up to 13V depending on voltage, load and power!!



- 1 To use a ring positive / negative tip the DC bus is reversed hence the +ve is connected to the 'Longest Leg' (see drawing)
- 2 IC5 (7809) is removed / not to be placed when a regulated 9V supply is used and pins 1 and 3 have to be bridged. ! Do not contact pin 2/GND



*Positive ring/negative tip (standard for guitar electronics)*

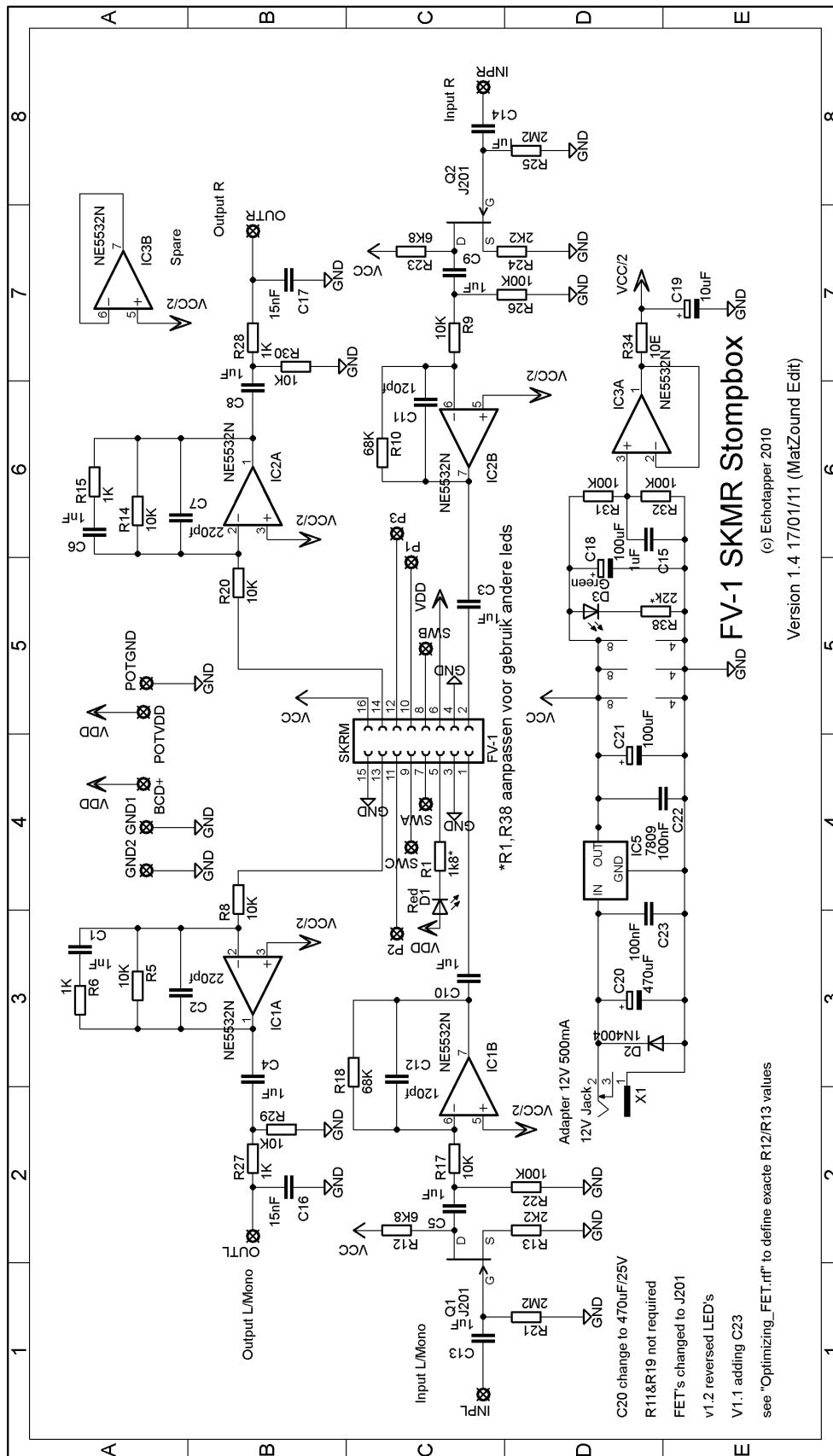


*Negative ring/positive tip (standard for consumer electronics)*

**Before placing ICs and FETs the voltages indicated in the red letters on the PCB should be checked.**

First measure the 9V points and then insert IC 3 in order to see if there is 4.5V generated at pin 1.

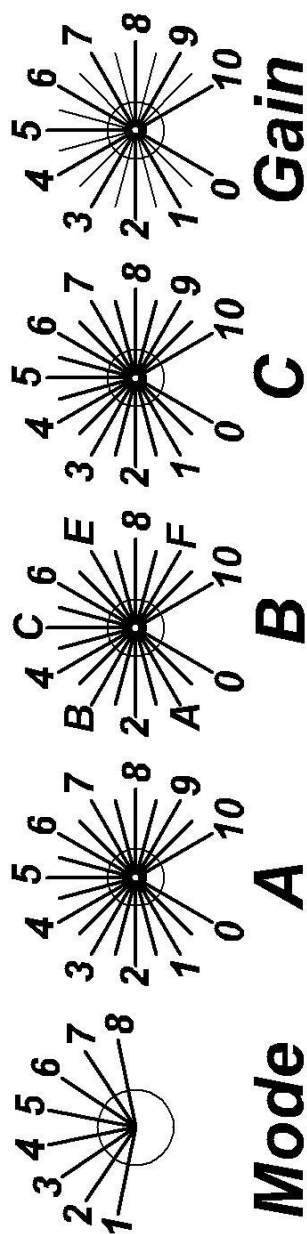
## 4.5 Schematic





Example for a decal

This can beprinted 1:1 (un-tick fit to page)  
And will then correspond to the drilling plan.  
Outside dimensions are 112mm x 180mm  
Always print a proof first!



Mod	Model	A	B	C
1	Meazzi Echomatic-I F classic	dry/wet	5-pos Sw	fdbck
2	Meazzi Echomatic-I F special	dry/wet	5-pos Sw	fdbck
3	Meazzi Echomatic-II bank	dry/wet	5-pos Sw	fdbck
4	Vox Long Tom classic	dry/wet	5-pos Sw	fdbck
5	Meazzi Echomatic-II classic	head 1,2,3	head 4	fdbck
6	Roland 301 mode 5	dry/wet	speed	fdbck
7	Reverb Tremolo	reverb	rate	depth
8	Meazzi Echomatic-I J classic	dry/wet	5-pos Sw	fdbck