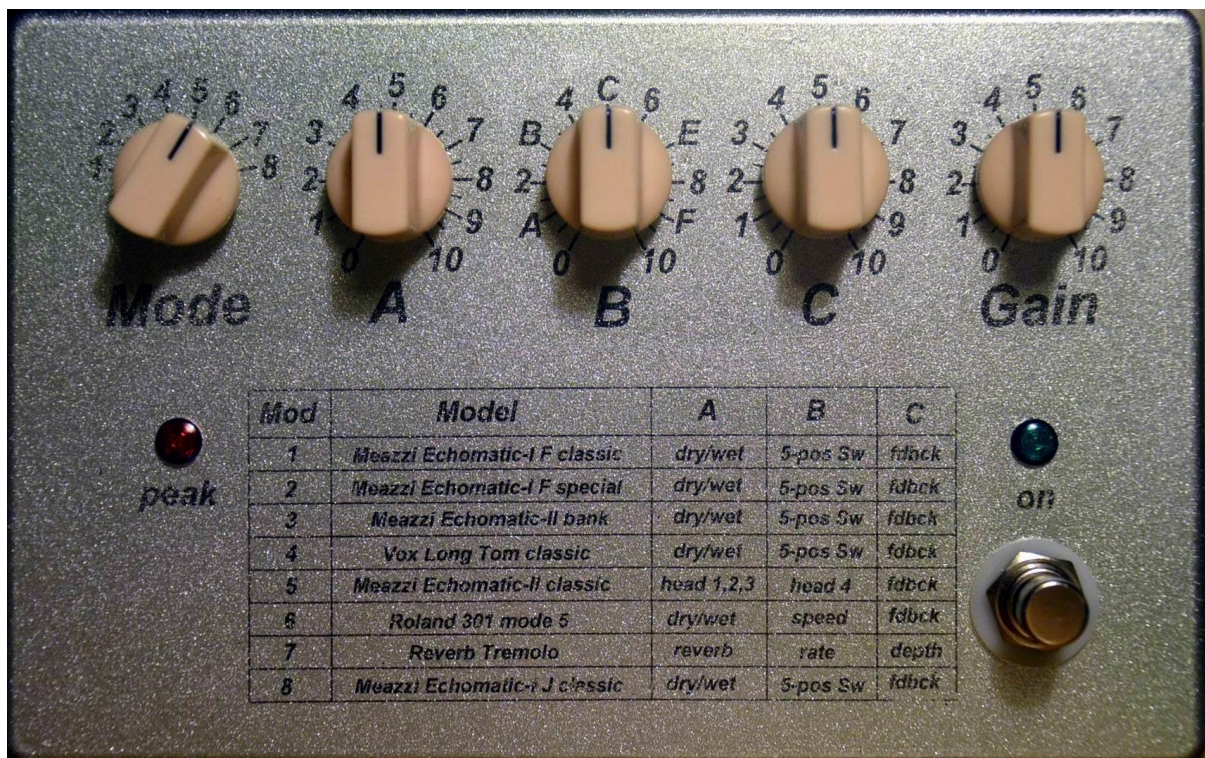


eTap2HW

Vintage echo emulation DIY kit



BUILDING INSTRUCTIONS

Newtone Version

UPDATE VERSION 2.0

©May 2015

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

CONTENTS

Update	3
Bill of Materials	4
Version 2.0 Layout of the motherboard	5
Please note for step 7 and 8	6
FET Optimalisation	8
Versie 2.0 Schematic drawing	10

Update

This V2 build instruction is based on improvements as being researched and developed by Stephen Mitchell in the UK and provides improvements over the basic unit:

- Less noise
- More audio headroom
- Improved low pass filtering
- Optimized FET triode characteristics

Compared to the original Bill of Materials:

- Resistors R22 en R26 are not used (location is to be kept empty)
- Resistors R9 and R17 are now 22K Ohm (was 10K Ohms)
- C11 and C12 are now 220pF capacitors (was 120pF)
- R13 and R24 are now 1K5 Ohm as an initial value (was 2K2)
- R12 and R23 are now 15K Ohm as an initial value (was 6K8)
- IC1 and IC2 are now TLC2272 (was NE5532P) IC3 is still NE5532P

The values for R12, R13, R23 and R24 have shown to be better initial values for the FET settings.

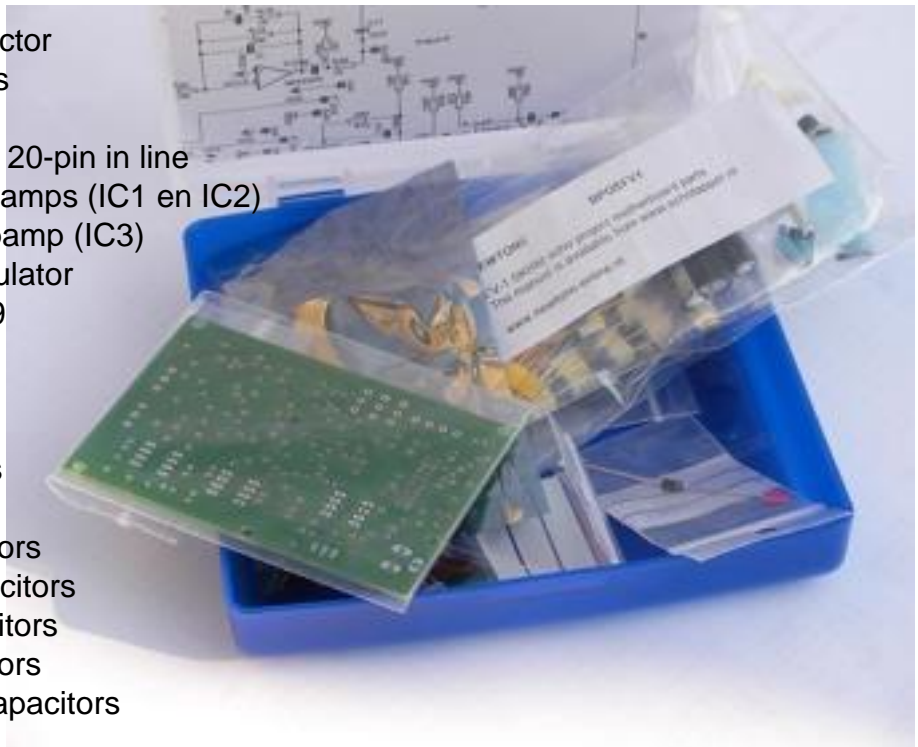
When you build the unit you can still use the detailed version 1.0 building manual. But please note that during the population of the PCB the following points are different:

- Step 1: R9 and R17 are now 22K so you now you place 6x10K and 2x 22K
- Step 3: R22 en R26 are not used, so now you place 2x 100K (the locations for R22 and R26 are kept empty)
- Step 7: The source resistors R13 and R24 are now 1,5K PLEASE NOTE these are now placed on solder pins
- Step 8: The drain resistors R12 and R23 are now 15K PLEASE NOTE these are now placed on solder pins t
- Step 14: C11 and C12 are now 220pF

On the following pages you can find the new Bill of Materials, the new motherboard layout and the new Schematic drawing.

Version 2.0 Bill of Materials

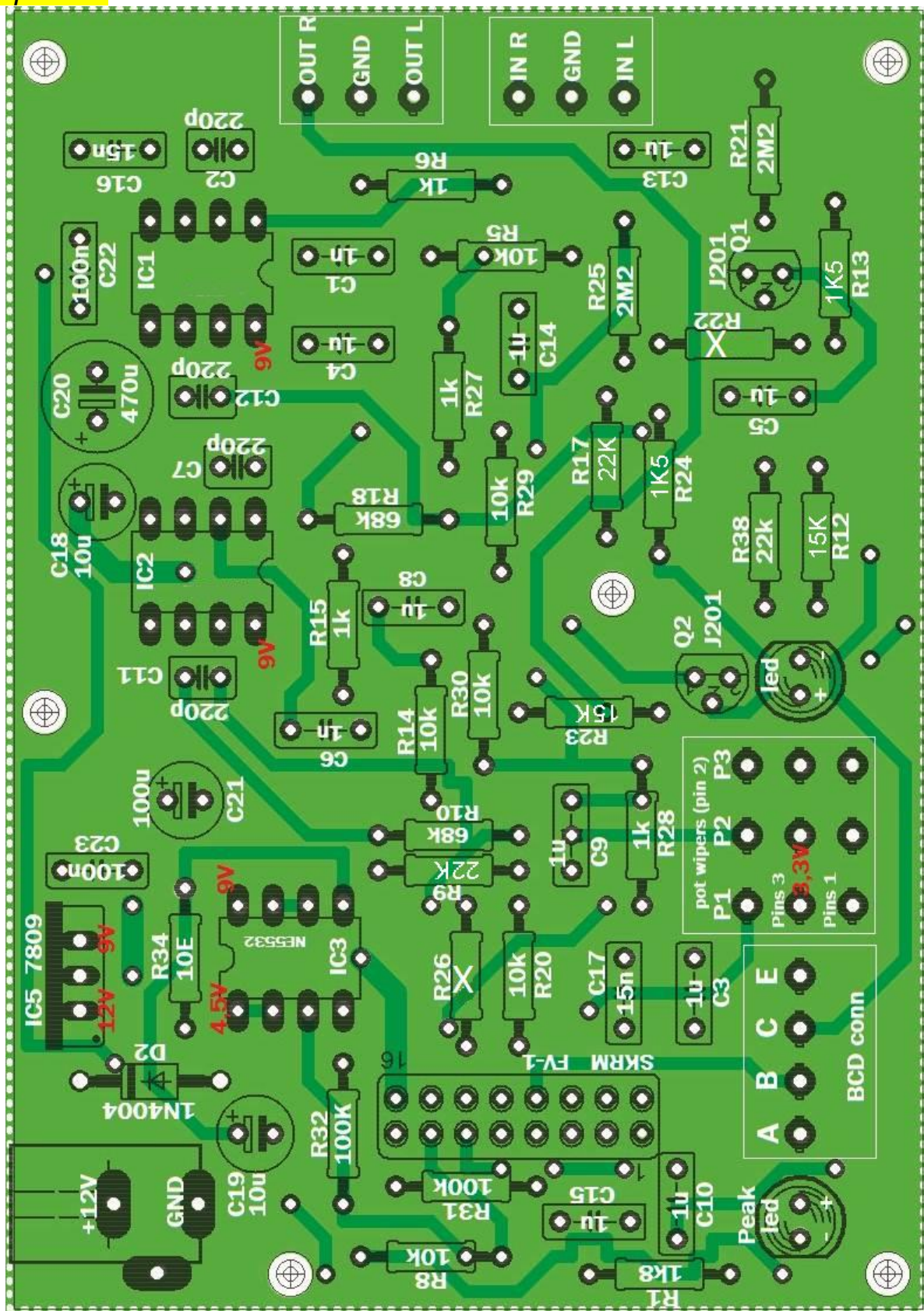
- 1 Aluminium enclosure type DD
- 1 PCB eTap2HW motherboard
- 1 PCB (populated) SKRM daughterboard by with eTap2hw pre-programmed
- 1 3PDT footswitch
- 2 Mono jacks
- 1 DC connector
- 3 10k lin potentiometers
- 1 1M log potentiometer
- 1 Lorlin BCD-schakelaar
- 2 Bright leds transparent
- 1 Fresnel lens red (peak)
- 1 Fresnel lens green (on)
- 4 Miniature spacers (15mm) to mount the PCB
- 1 set solderpins
- 1 2-pins pcb connector
- 3 IC sockets, 8-pins
- 1 2x8-pins header
- 1 Transistorsocket, 20-pin in line
- 2 TLC2272 dual opamps (IC1 en IC2)
- 1 NE5532p dual opamp (IC3)
- 1 7809 voltage regulator
- 1 Heatsink for 7809
- 1 1N4004 diode
- 2 J201 fets
- 1 470 μ F/25V elco
- 2 100 μ F/16V elco's
- 1 10 μ F/16V elco
- 9 1 μ F MKT capacitors
- 2 100nF MKT capacitors
- 2 15nF MKT capacitors
- 2 1nF MKT capacitors
- 5 220pF ceramic capacitors
- 1 10 Ω resistor
- 4 1k Ω resistors
- 2 1,5k Ω resistors (fet source resistors)
- 1 1,8k Ω resistor (peak LED)
- 6 10k Ω resistors
- 2 15k Ω resistors (fet drain resistors)
- 3 22k Ω resistors (on/bypass led and R9 and R17)
- 1 33k Ω resistor
- 2 68k Ω resistors
- 2 100k Ω resistors
- 2 2,2M Ω resistors



Check contents of kit before you start.

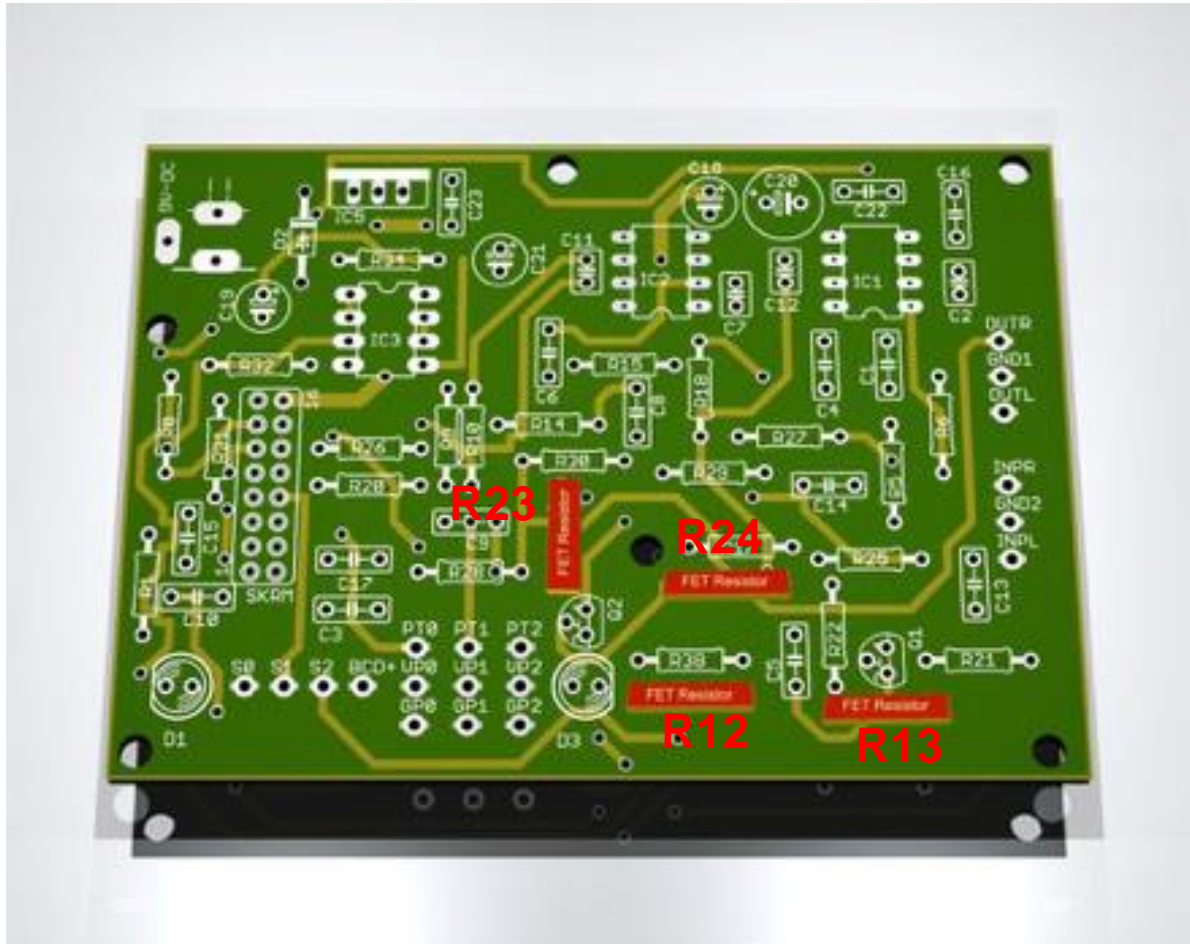
Version 2.0 Layout of the motherboard components

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.. All values are for version 2.0, R22 and R26 marked with an X are not placed.



Please note for step 7 and 8

In this step, the values of the resistors are now 1.5K (for R13 and R24), and 15K (R12 and R23). The locations for these transistors are marked on the image below.



In Version 2 these resistors are placed on solderpins. Make sure the eye is positioned in alignment with the resistor lead so NO BENDING is required making replacement easy. The solderpin should be soldered on the motherboard and the resistor soldered on the pin.



With the straight short lengths the resistor can be placed and, if required, replaced with ease without damaging the motherboard

Install 1K5 into R13 and 15K into R12. it is sufficient to just connect with minimal soldering as they need to be possible replaced.



Install 1K5 into R24 and 15K into R23 it is sufficient to just connect with minimal soldering as they need to be possible replaced.



Create and install 2 Sockets with pin-array part of the kit



The provided 'start' resistor pairs (1.5K and 15K) for the FET transistors are acceptable but fine-tuning them to be fully optimized makes the harmonics being generated including a small amount of compression similar to an ECC83/12AX7 triode tube.

With optimal resistor values per FET transistor and a correct setting of the level potentiometer you will find the your guitars 'sweet spot' and experience 'that sound'!

FET Optimisation

As mentioned the resistors included in the kit should give acceptable results. However it is possible to further optimize the settings for the FETS.

Equipment used: DMM, Multimeter of a good quality with digital readout.

De procedure is:

1. Install the FET Q1 and apply power to the eTap2hw unit.
2. Measure the regulated voltage between GP2 pad and the left connection of the power LED (see picture) record voltage as Vss.
3. Measure voltage across R13 and report this voltage as Q1Vs
Email the Vss en Vs values to echotapper@ziggo.nl
4. Wait for response. in most cases another value or values is reported back.
5. Install resistors of recommended value or values
6. Measure the voltage over R13 again and report again.
7. Wait for conclusions
8. finalize
9. Repeat this procedure for Q2 and R24

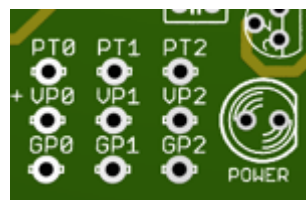
What happens when voltage measurements are sent?

These values are entered into a sophisticated calculation model that calculates by an iterative process the best fit for that FET transistor in the eTap2hw unit so it behaves optimal for the task of emulating triodes.

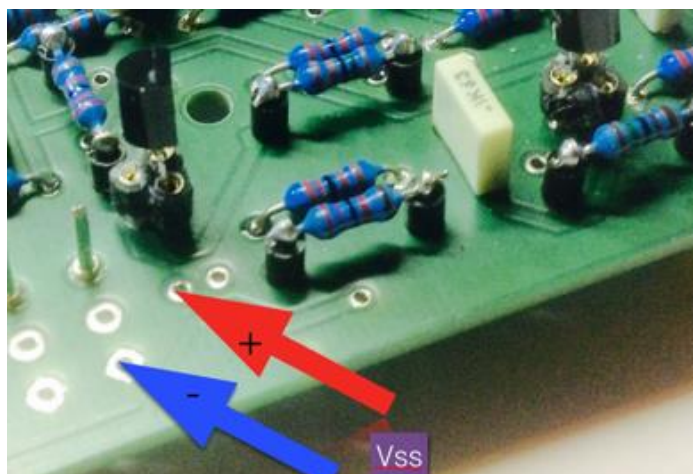
Digital multimeter



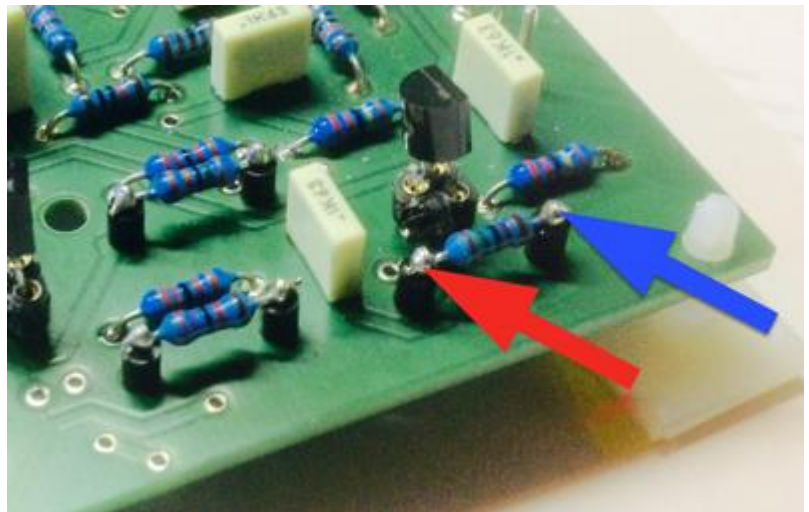
solder pads



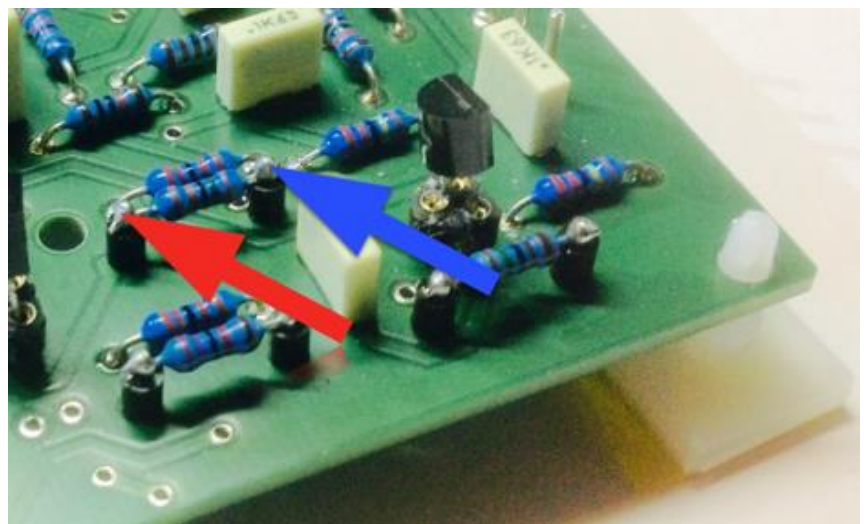
Measure the (Step 2) voltage here and report it as the Vss voltage. It is the regulated voltage for your motherboard and is around 8.80-9.0 volts D.C.



Measure the Vs voltage
across resistor R13 and report as
Q1Vs



Measure the Vs voltage
across resistor R24 and
report as Q2Vs



The report would look something like this but then with your measured values.....

*Vss= 8.88 Volt
Q1Vs = 0.3 Volt
Q2Vs = 0.26 Volt*

