TubeOhm Instruments
in cooperation with Rolf Degen
present:

Jeannie
polyphoner Synthesizer

the electronic part

DIY Manual V 1.07 / 15.09.22

TubeOhm 2021

Jeannie - DIY Synthesizer , technical data

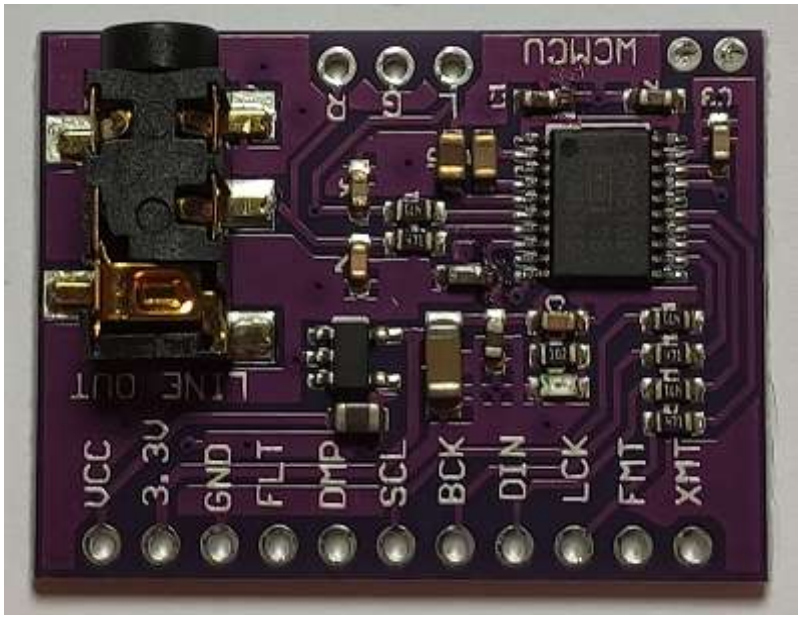
- **Power supply unit 12 V DC -minimum 1 A , plug 5.5 - 2.1 center positive**
- **micro SD RAM Karte 8..16 GB**
- **Power consumption 3.6 Watt**
- **8 Voices polyphonic**
- **multimode Filter**
- **2x ADSR**
- **2x LFO**
- **10 Waveshaper**
- **960 different waveforms per oscillator**
- **2 Oszillatoren pro voice**
- **xor, xmod, modulo, and, or and FM**
- **8-fold unison mode with up to 16 oscillators / one voice**
- **FX with Hall , Chorus, Delay , Pitchshifter, uvm.**
- **adjustable clock frequency for the FX DSP - dirty effects**
- **2048 programmes can be stored on micro SD card**
- **graphic LCD display**
- **Teensy 4.1**
- **DIY friendly**

Changes in V 1.01 : C 16,20 = now 100 nF, with this the encoder works better . Also C9 and C15 is now 1 uF instead 4,7 uF.
Changes in V 1.05 . The AD converter board is now soldered directly to the board with the pins.
The coils L1,2,3,4 and on the FX board L23 can have 3.3 uH or also 4.7 uH.
Changes in V 1.06 : bugfix in the DIY manual.. 107 picture of a tantal cap

To build Jeannie we need some basic parts

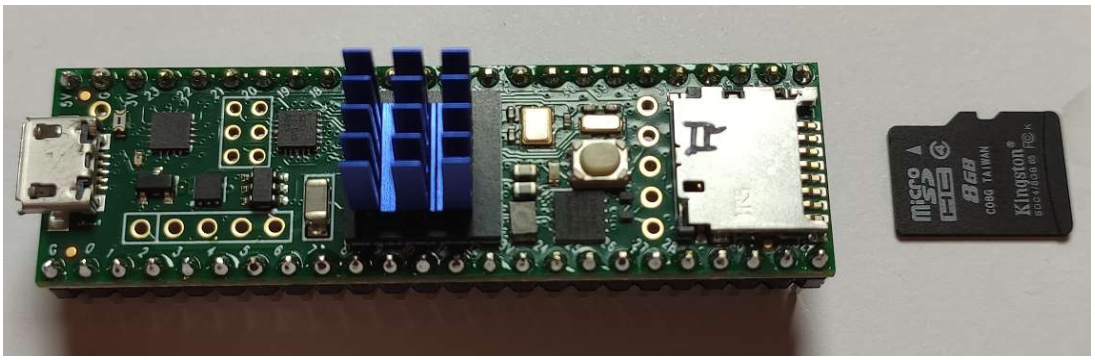
First of all, the PCM 5102A AD converter is ready build on a circuit board. This is included in the full kit and tested. You can also find it at various suppliers such as Amazon/Alibaba/Banggood.
The important thing is 11 pins on one side and three on the other. I mention it specifically because there are two designs. The pins are soldered on from below

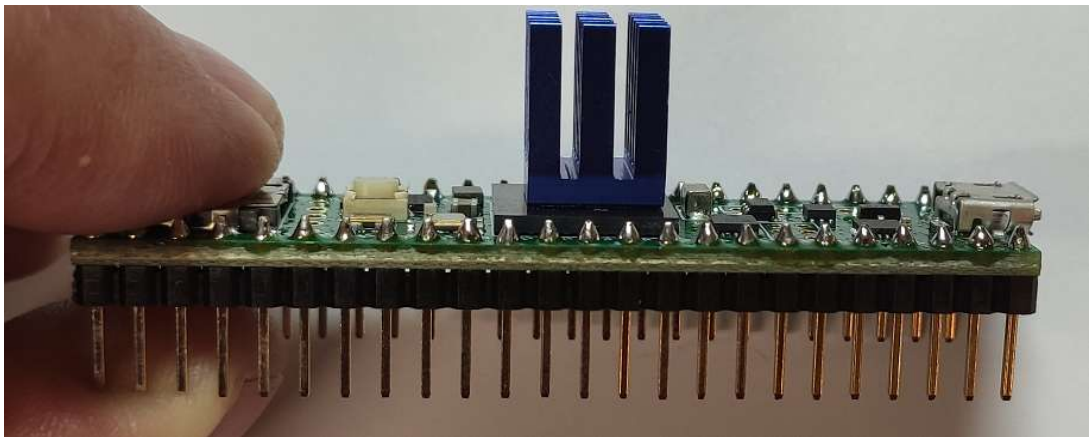
AD Wandler 24 Bit with PCM 5102A



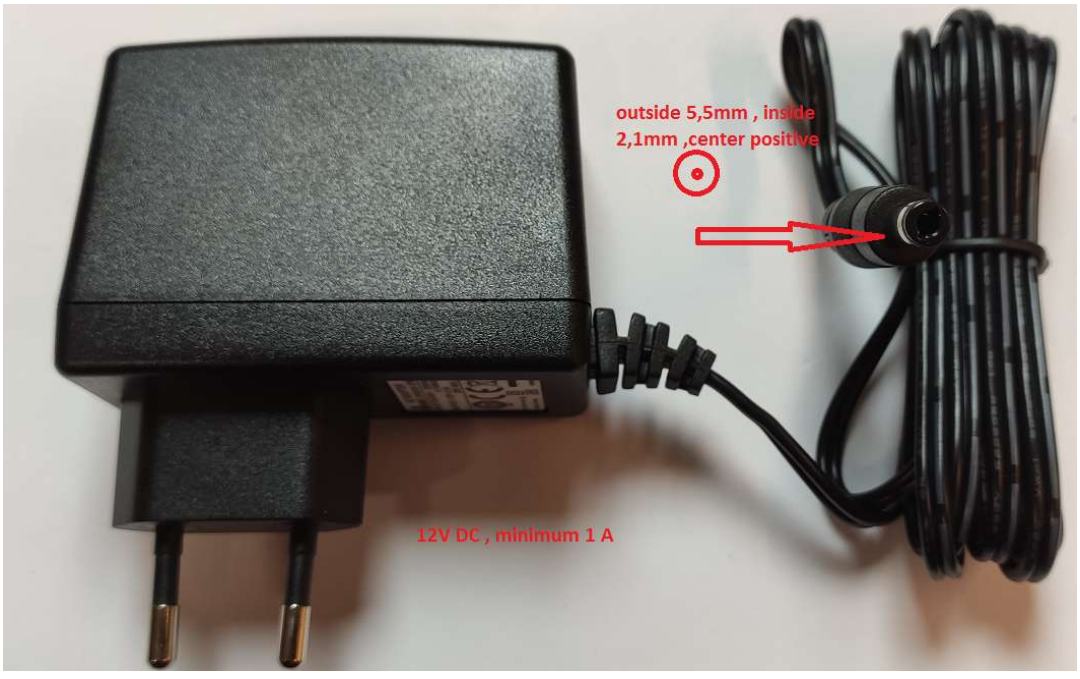
The most important element is of course the Teensy 4.1, which is available from various distributors such as Reichelt. A heat sink is glued to the Teensy and the USB 5 volt - described below - is changed. The two pin headers are soldered in from below.

Teensy - 800 MHZ clock





Switching power supply - DC 12 V , 1 A, Plug 5,5 – 2,1 center positiv-



Introduction:

It's amazing what has happened in electronics in recent years. Synthesizers that used to be as big as a cabinet have now shrunk to the size of a Eurocard and are even suitable for DIY.

I ask all interested DIY'ers to read this manual at least once BEFORE you build Jeannie.
Although it is relatively easy to build Jeannie, you should have some experience in soldering.
You should also be familiar with the component/part labels.

You have not only bought a DIY kit but also the experience of building your own synthesizer.
In this sense, have fun while building and even more fun while playing the instrument.

***Depending on the supplier, the components may look different or have a different colour. The important thing is the value and the pitch !

TubeOhm 28.08.21

1:) first works (IMPORTANT)

First of all, we should switch the Teensy 4.1 to external power supply so that it is not forgotten.

**** The Teensy is supplied with 5 V from the Jeannie. When programming a new firmware via USB, the Teensy also gets 5 V from the PC. Since there can be voltage differences between the PC and the Jeannie, a line on the Teensy MUST be disconnected.

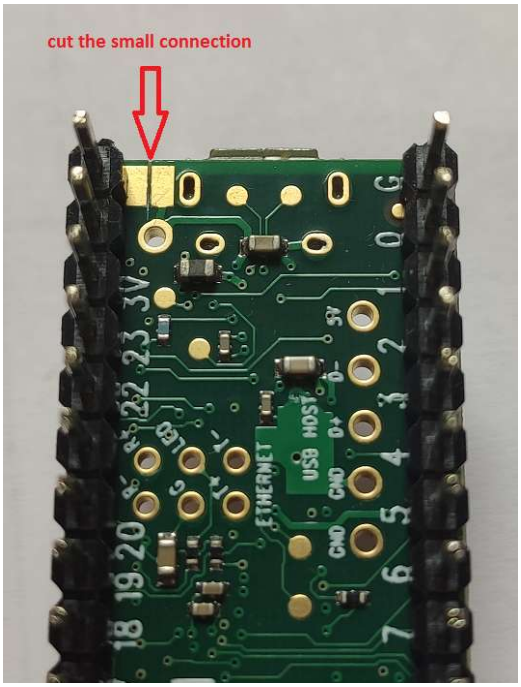


Bild 1

You can see the underside of the Teensy 4.1 here.

Both PADS are connected by a very narrow cable.

This wire is cut with a cutter
cutter knife or a razor blade or even a sharp knife.

Now the 5V USB power supply is interrupted. Only the USB data lines are still active.

**The Teensys supplied by TubeOhm are already switched to external and have the latest firmware.

The Teensy 4.1 also needs a heat sink (picture 2) because we are running it within the specifications but with a slightly higher clock speed.

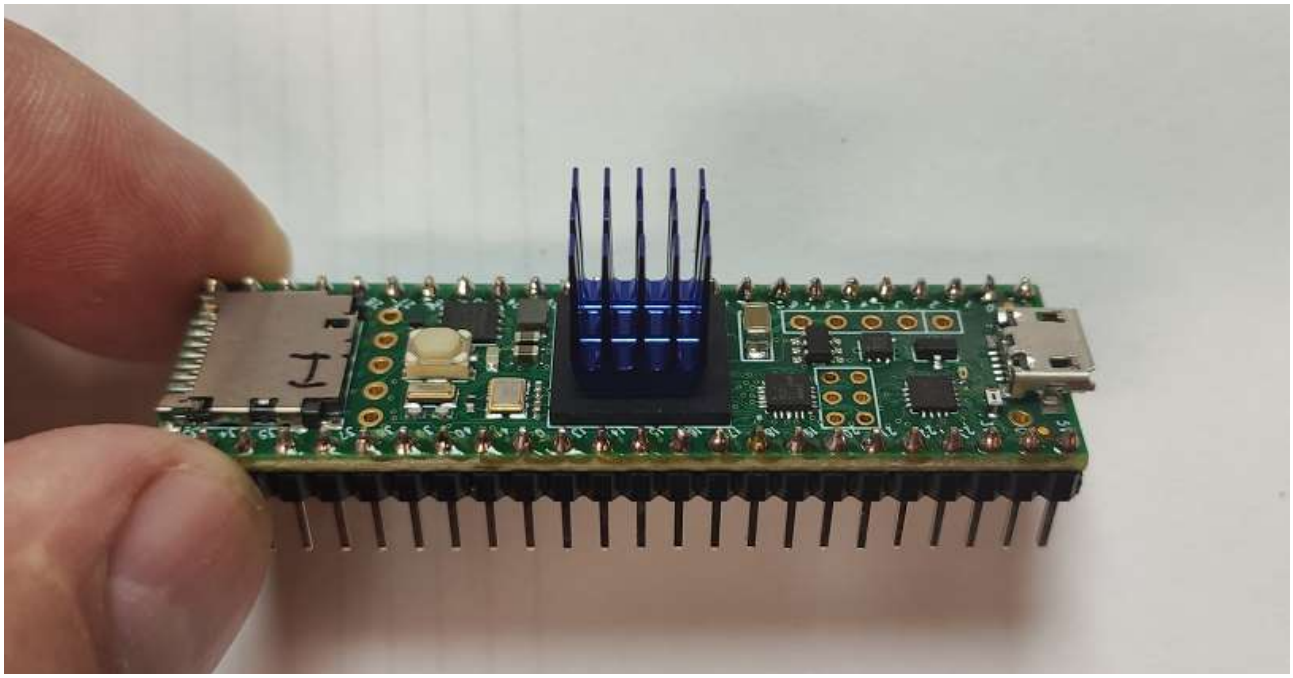


Figure 2
So the preliminary work is done for now.

2:) the SMD components

For a long time we have resisted using SMD components in DIY projects.
Unfortunately, certain components are only available as SMDs. To make soldering easier, we have lengthened the pads on the board considerably.

And, to be honest, with a little practice, the components can be soldered without any problems. Even by hand.
If you have ordered a board with soldered SMD ICs from TubeOhm, you can skip the next chapter 'Soldering the SMD components - procedure'. If not - please read on.

In the Jeannie we use SOIC (small outline ICs) 14,16,8. These are just solderable.
Pay attention to pin 1 on the IC. If the IC is soldered the wrong way round, the whole circuit will not work and in the worst case the board will be broken.

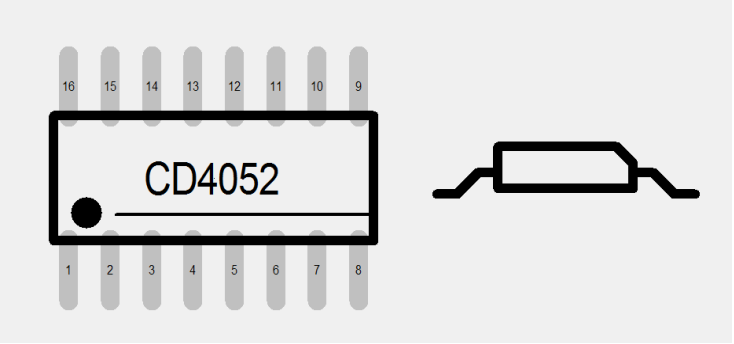


Figure 3

Figure 3 shows how pin 1 on the IC is marked.
Either:

a:) by a dot or indentation
b:) one side of the housing is slightly bevelled.
The counting is from the left ,pin 1, to the right, pin 7 SO 14 or pin 8 SO 16.

On the PCB, pin 1 of the ICs is marked, this with a dot and also with a symbolic bevel

Soldering the SMD components-Procedure

The connections pin 1 and pin 8 (SOIC 14) or pin 1 and pin 9 (SOIC 16) are tinned on the board. Place the IC as straight as possible on the pads and hold it firmly with your fingers. The soldering iron tip should be cleaned of solder residue. Use the soldering iron tip to quickly press pin 1 of the IC into the tinned pad 1 of the PCB.
Repeat the same procedure with pin 8/9. The IC is now soldered to pins 1 and 8/9 on the board.

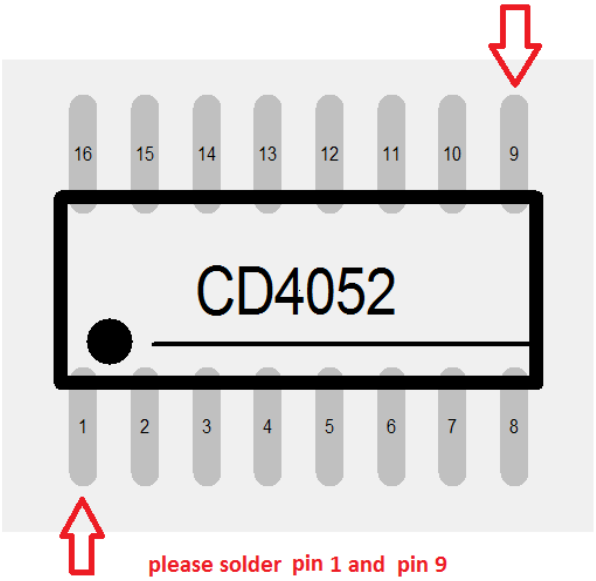


Figure 4

When the IC is straight, the other legs of the IC are carefully soldered on. If you use a little soldering honey as an additional flux, the solder flows from the pads directly to the legs.

Soldering the SMD ICs is actually the most difficult part of the job.

Now let's turn to the board and get started .

Here is the front side of the board again

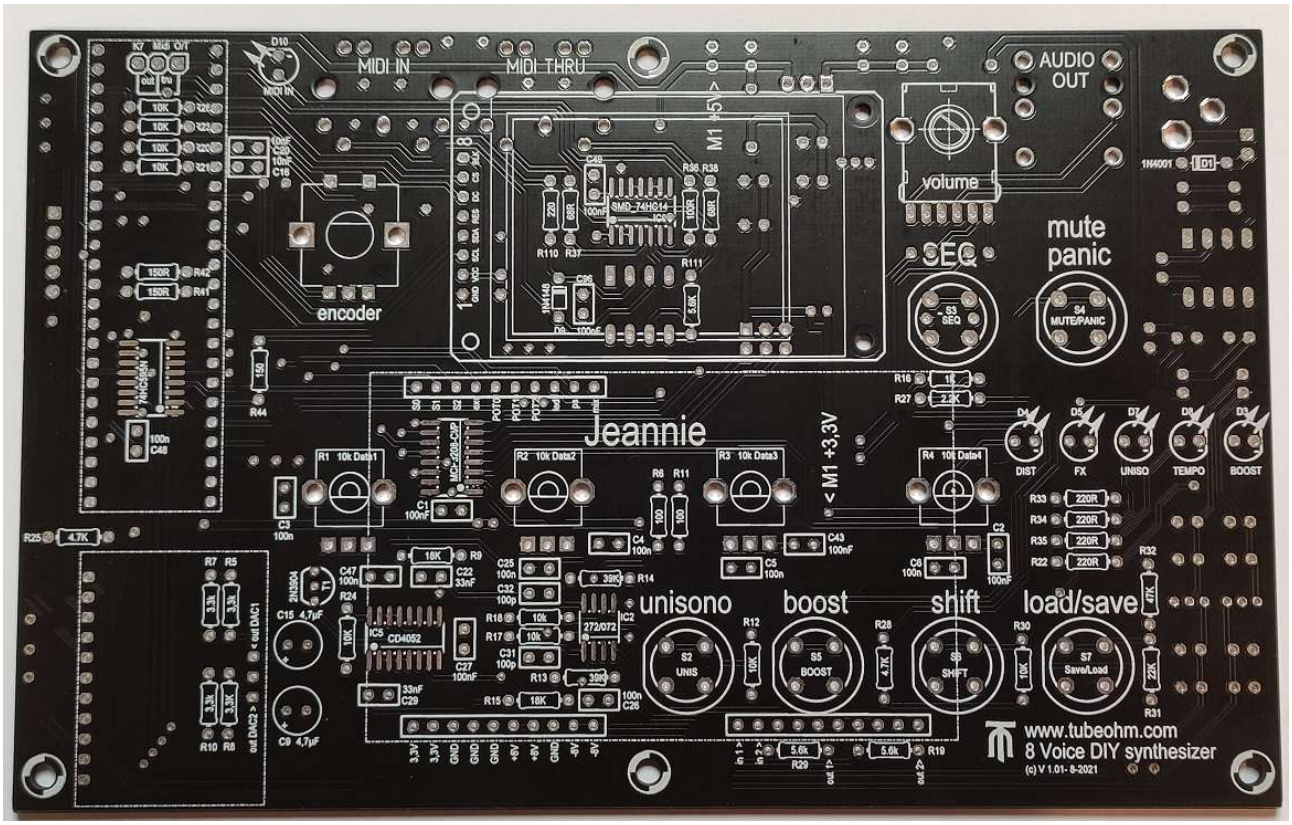


Figure 5

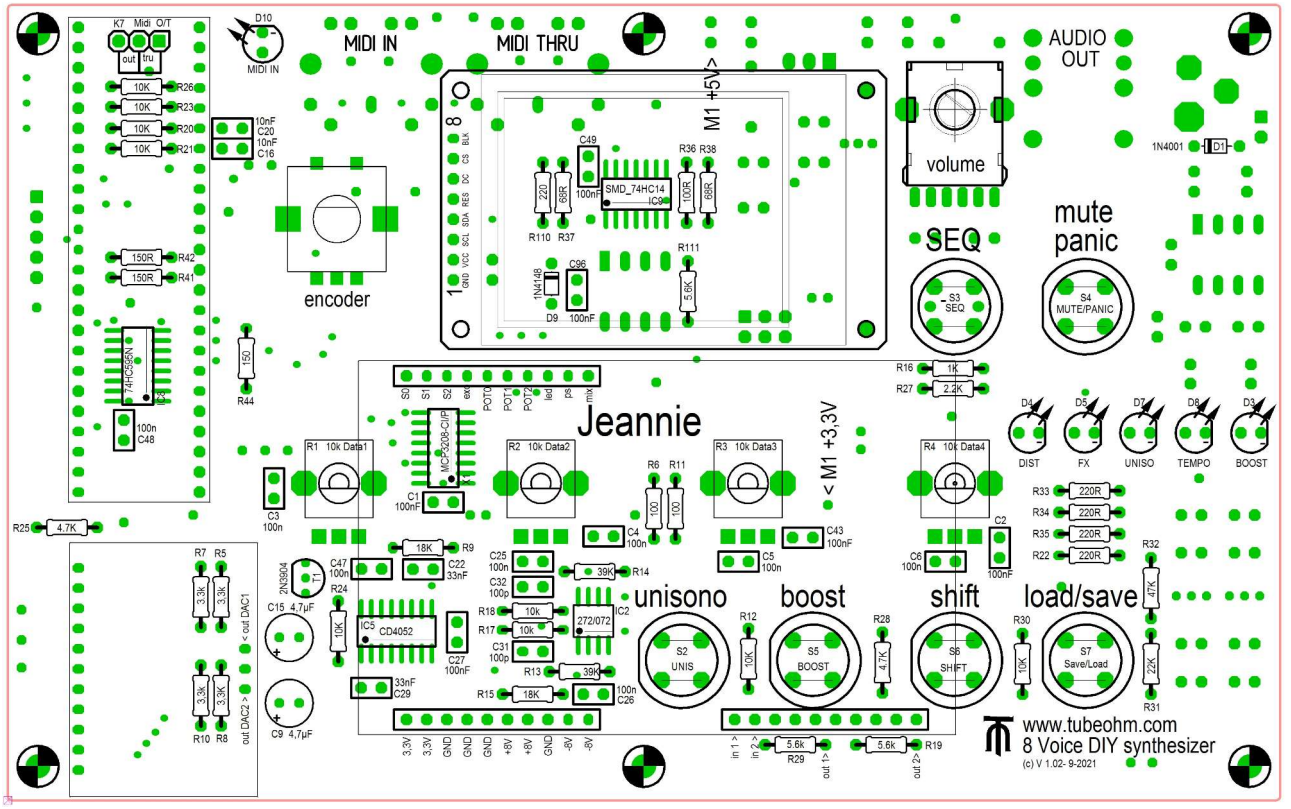


Figure 6

And here is the back

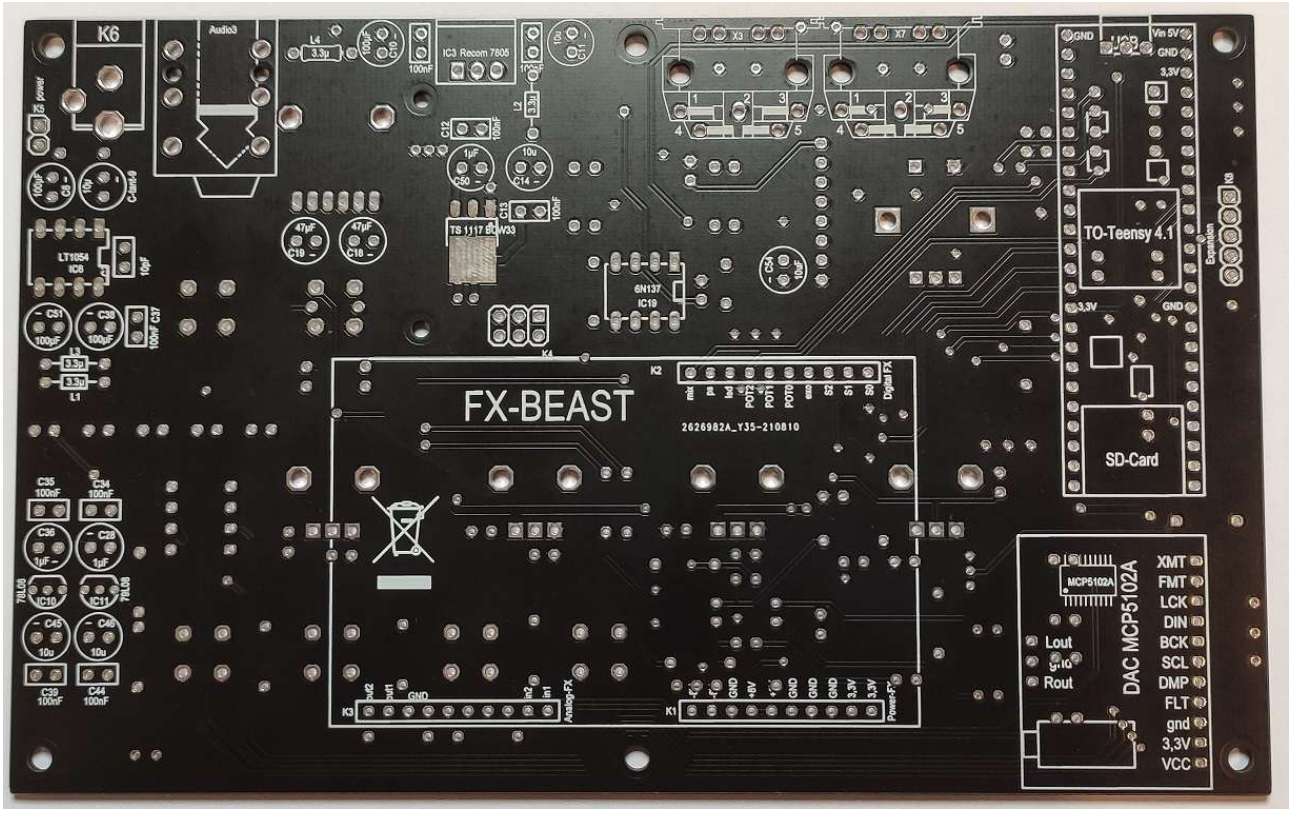


Figure 7

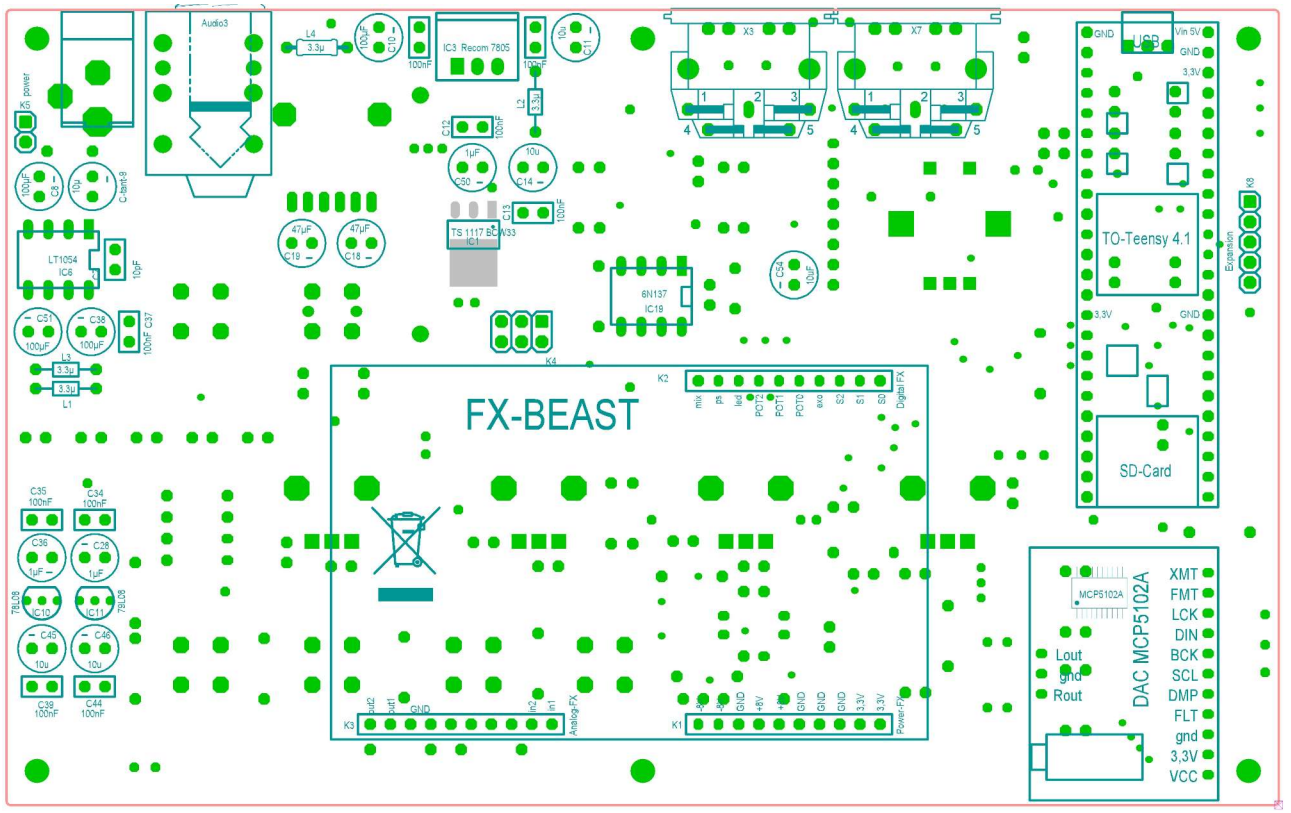


Figure 8

Now it's time for soldering.

We start with IC 5 CD 4052. Attention: This IC MUST be a CMOS IC. Because this IC gets a supply voltage of +/- 8 volts.

First we tin pin 8 on the board.

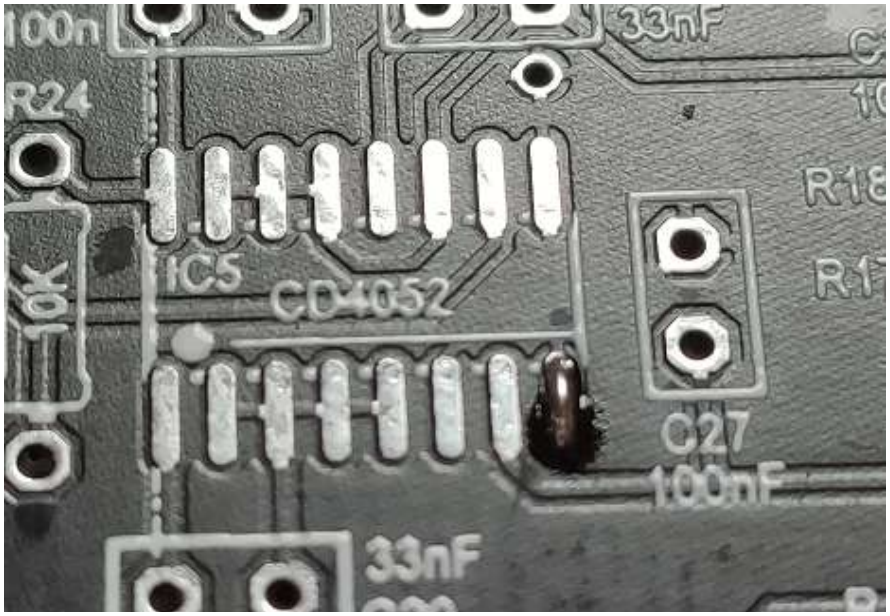


Figure 9

Then the IC is aligned and soldered to the leg.

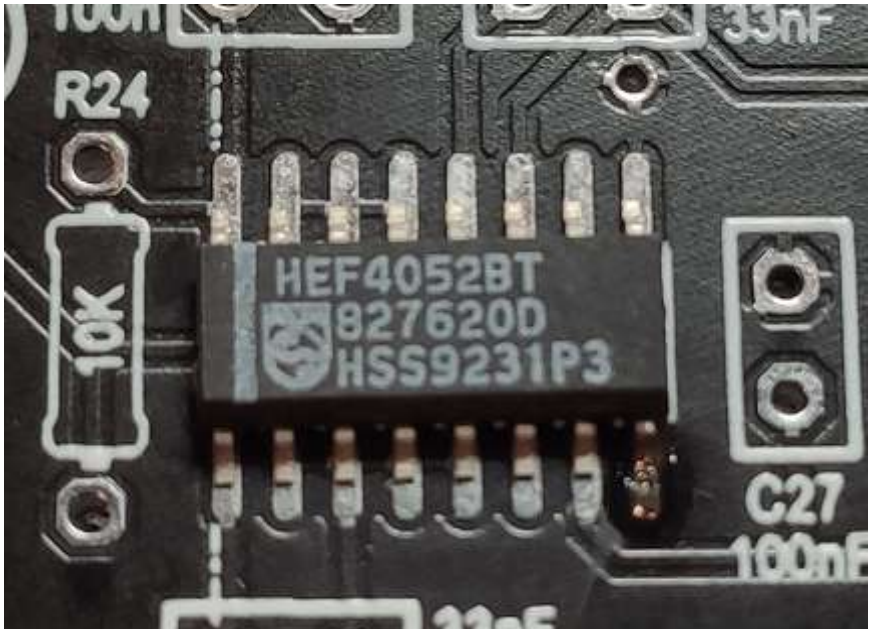


Figure 10

Then the IC is straightened and pin 16 is soldered.

Figure 11

Now the IC can be soldered completely. The connections of the IC are cleaned with a cotton swab and a cleaner and the solder pads are checked with a magnifying glass.

Figure 12

The remaining ICs are soldered using the same procedure.
*** First solder one pin, align it, solder the second pin and then solder it completely.
Attention, you should clean the tip of the soldering iron more often !

Now it should look like this.
Please clean the solder joints and look over them again with a magnifying glass.

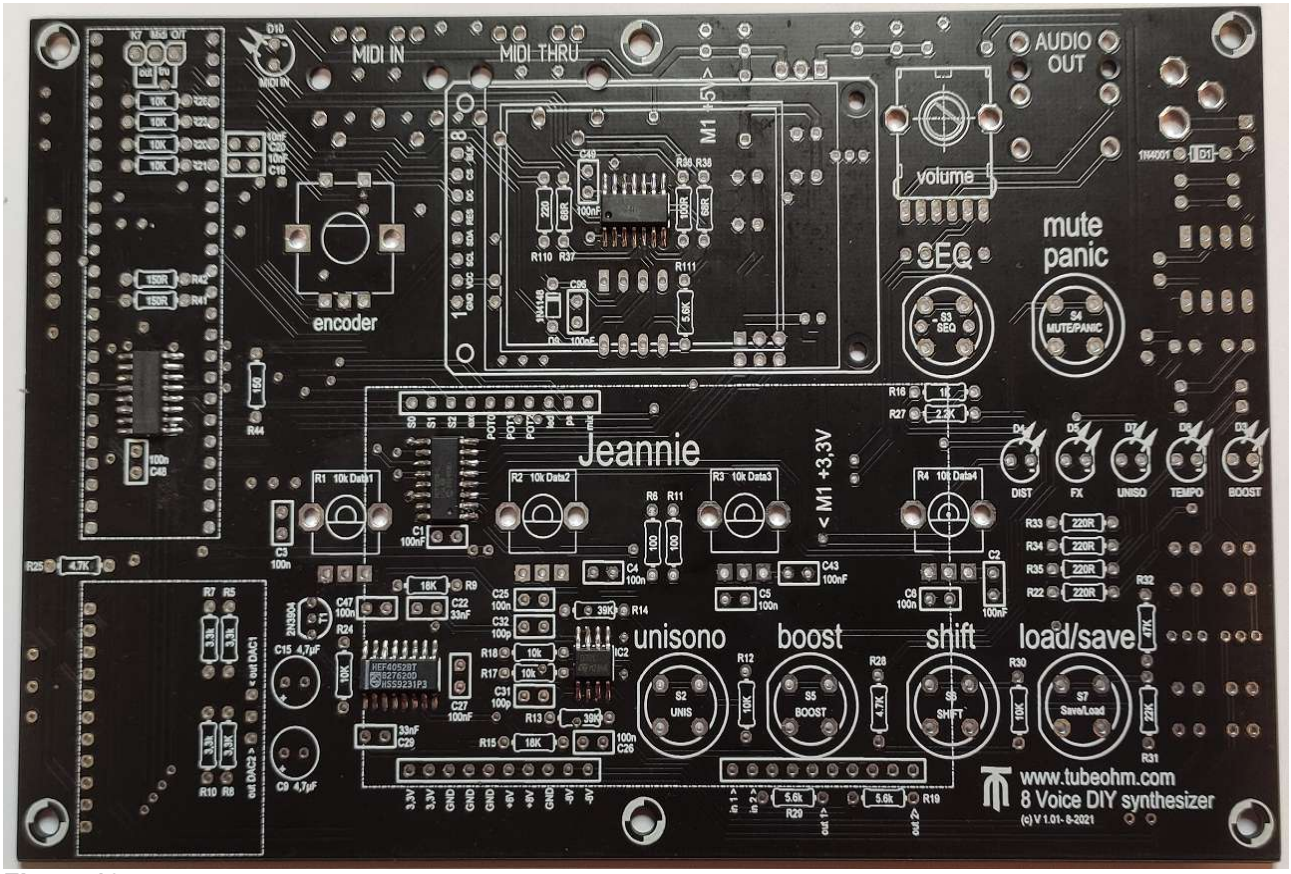
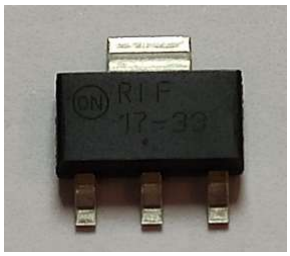


Figure 13

Once the SMD components are soldered, the most difficult part is done.

Now we turn the board over and solder the complete power supply.
We start with the TS 1117 3.3 Volt. This is the last SMD component.

TS 1117 33		
	3,3 Volt Regulator	SMT/SMD

3.3 Volt regulator soldered on . The soldered-on cooling fin serves for heat dissipation..

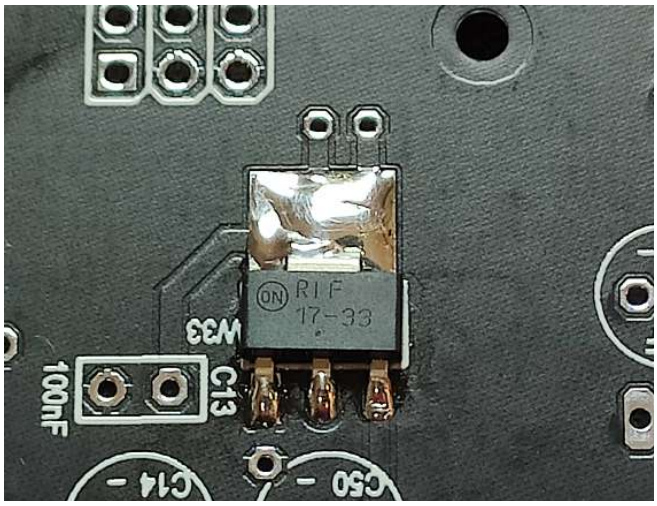
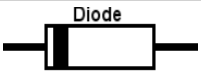


Figure 14

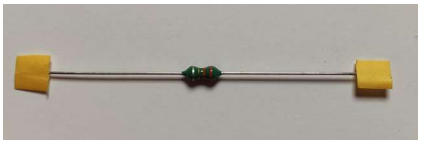
End of the SMD/SMT orgy.

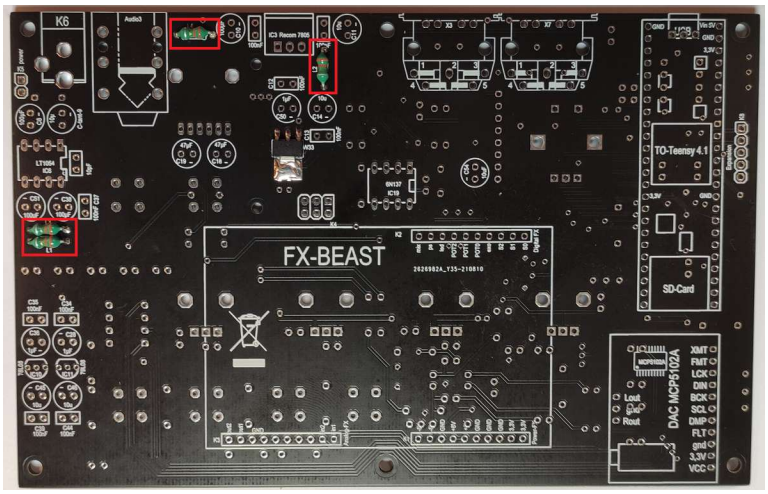
Building Jeannie with though hole components

Please solder the diode 1N4001 right away otherwise it will be quite fiddly later.

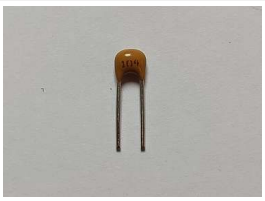
	D1= 1N4001 or 1N4004	-through hole polarized

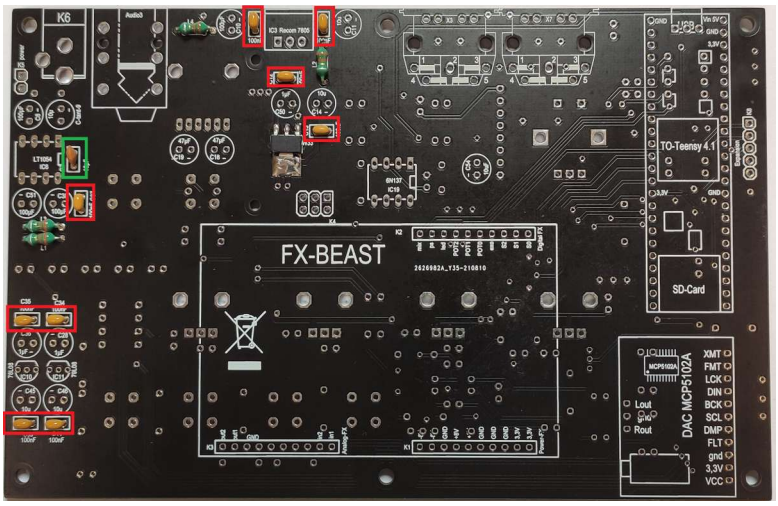
4x 3.3uH coils are soldered in the positions L1,L2,L3,L4

	L1,2,3,4= Coil 3,3 uH or 4,7uH Colorcode orange, orange,gold, silver=3,3 uH yellow, purple, gold, silver=4,7 uH	through hole *** Serves for interference pulse suppression



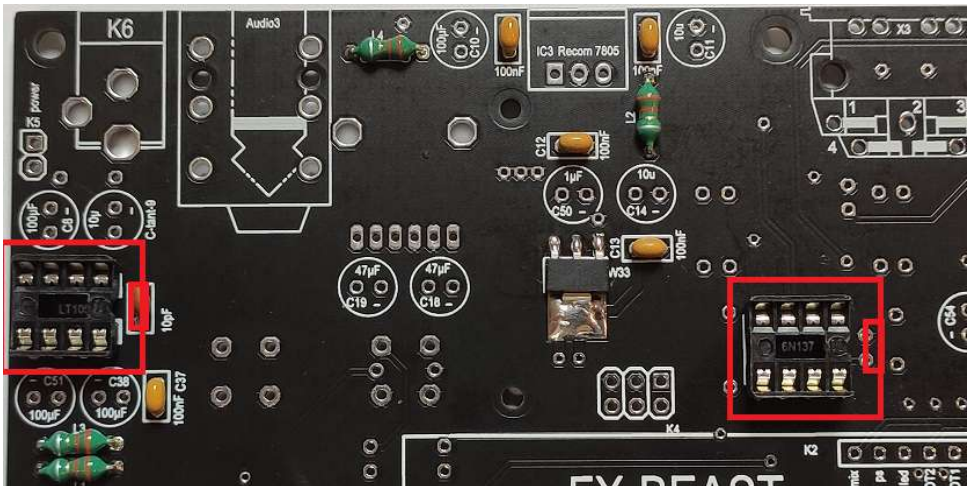
Now solder the 9x 100 nF and the 10 pF capacitor.

9x100n , 1x 10 pF		
	9x 100 nF marked 104 1x 10 pF marked 10	-through hole Interference suppression capacitors



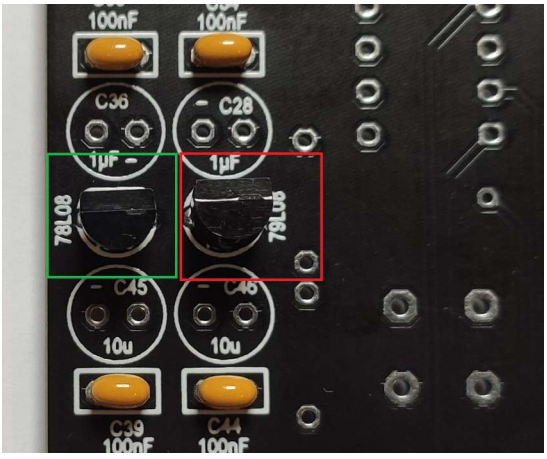
The next step is to solder in the IC sockets for ICs IC6 and IC 19.

2x8 pin IC sockets		
	sockets for IC6 and IC 19 Attention, this part is marked	-through hole

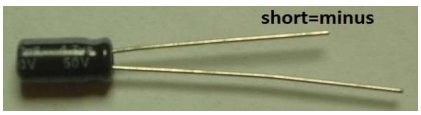


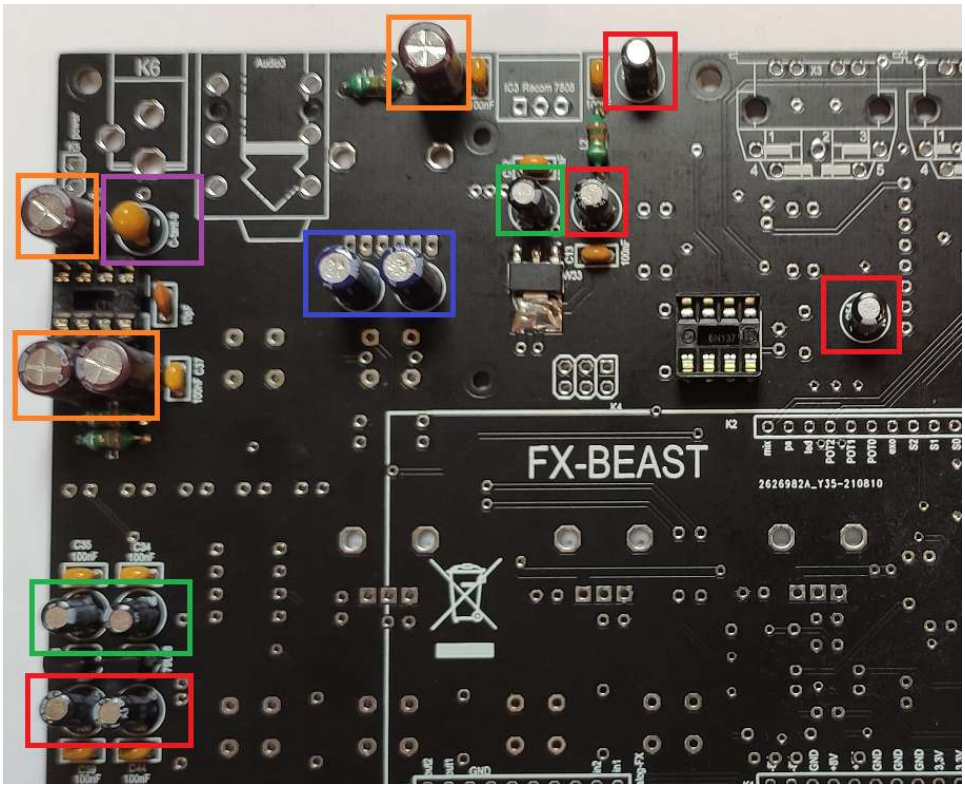
-8V =79L08 and +8V=78L08 Voltage stabilisers

78L08,79L08		
	2 voltage Regulator IC 10=78L08 =+8V IC 11=79L08=-8V	-through hole



electrolytic capacitors for the power supply

3x1uF,5x10uF,4x100uf, 1x Tantal 10uf		
 	C38,36,50= 1uF C11,14,45,46,54=10uF, C18,19= 47uF C8,10,38,51=100uF Ctant-9= 10uF tantal	-through hole -attention this parts are polarized minus(-) is the short leg Attention , the tantal is marked with plus

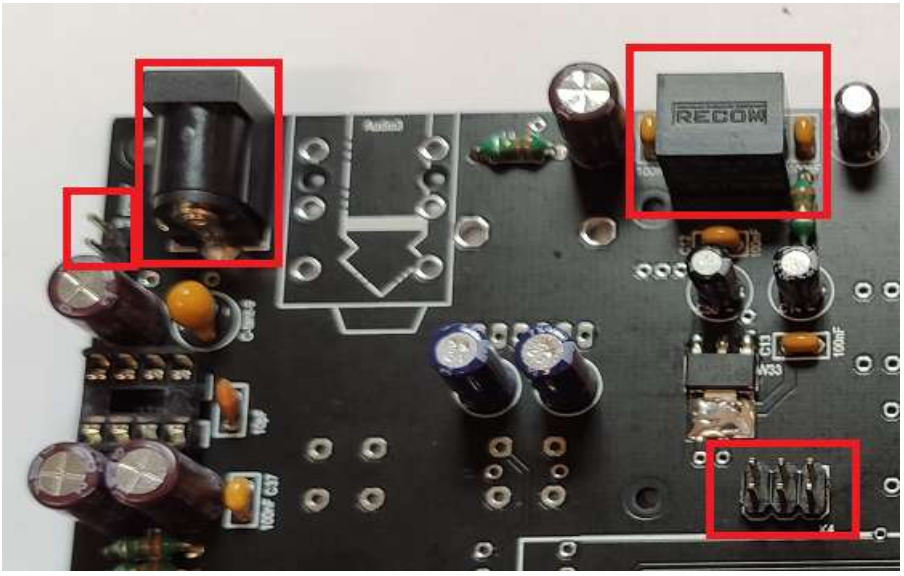


Now we solder in the 5V switching regulator, the DC socket and 2 pin headers
Attention, it has to be a switching regulator , a normal 7805 gets too hot !

K5 is used to connect a power switch and must be bridged

K4 is later used for additional functions .

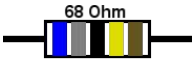
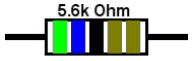
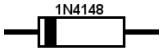
R78-E5.0-05, 12V DC, header K4,K5		
	IC3 =R78-E5.0-05 = 5 Volt K6= DC Plug 1x2 header K5 Power SW 2x3 header K4	-through hole Recom switching power regulator 5Volt

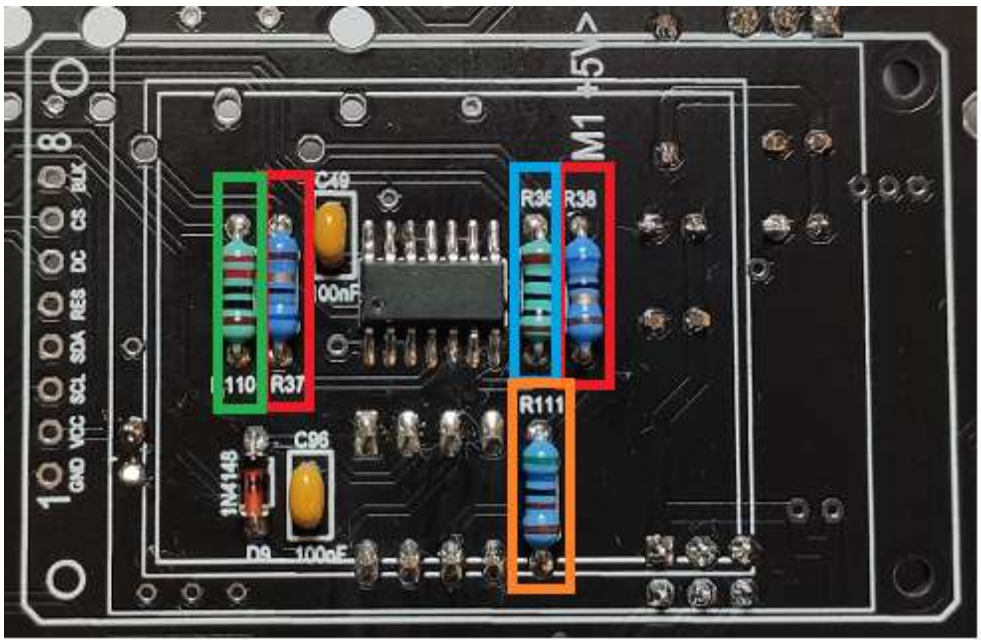


With this, we are done with the back of the board for now. The remaining components will be soldered in at the very end, otherwise the soldering of some parts may not be easy.

So people, now turn the board over.

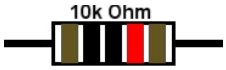
First we build the complete MIDI circuit.
For this we need :

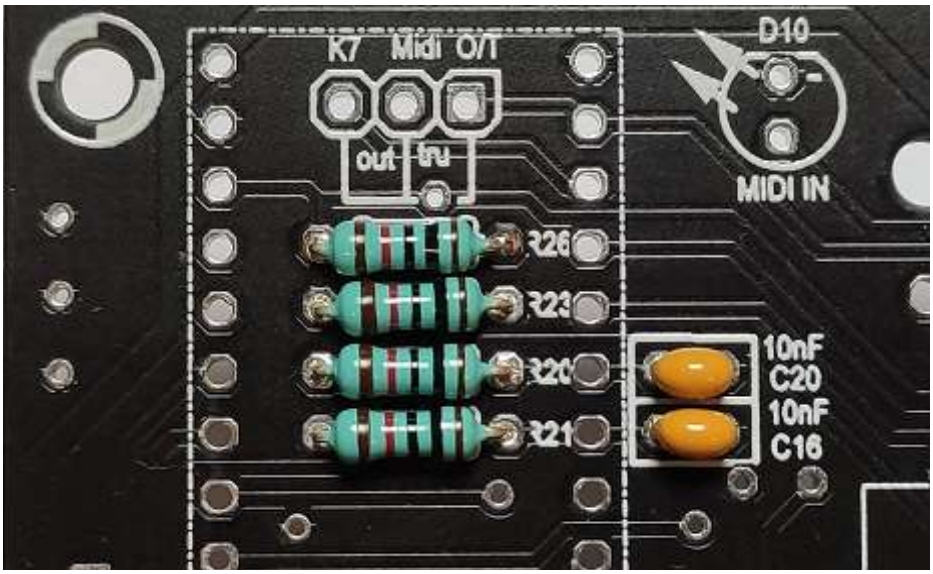
R110,R110,R36,R37,R38,D9 ,C49,C96		
	R37,38= 68R -blue,gray,black,gold,brown	-through hole metalloxide , 1%
	R36=100R -brown,black,black,black,brown	-through hole metalloxide , 1%
	R110=220R -red,red, black,black,brown	-through hole metalloxide , 1%
	R111 -green,blue,black,brown,brown	-through hole metalloxide , 1%
	D9=1N4148 Diode	-polarized, through hole
	C49,96 =100nF -marked 104	-through hole



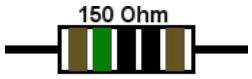
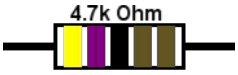
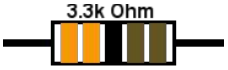
Now solder in the 4 resistors and 2 capacitors for the encoder.

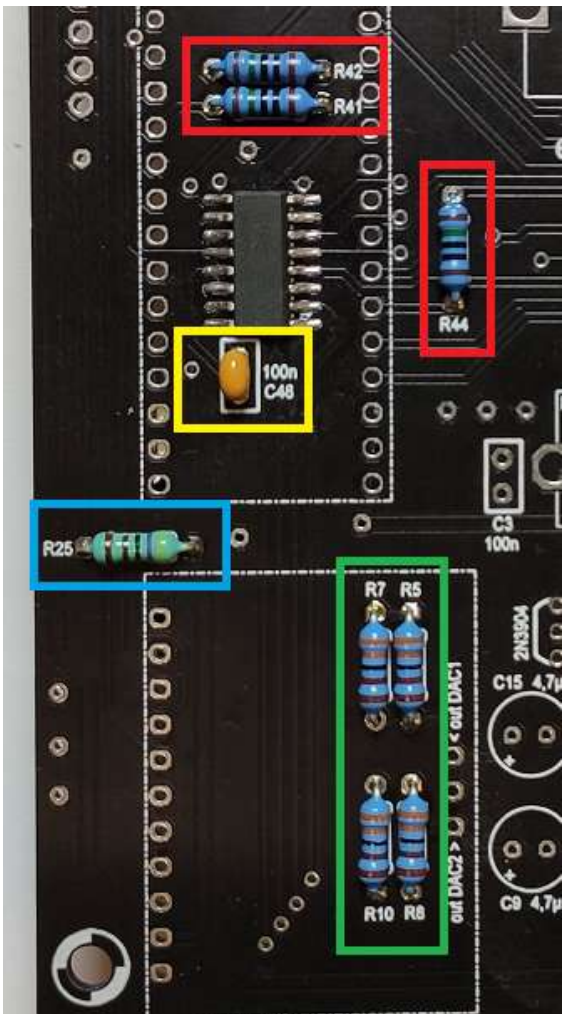
The resistors and capacitors are used for interference pulse suppression of the encoder.

R20,21,23,26 and C16,20		
	R20,21,23,26 =10kohm -brown,black,black,red,brown	-through hole metalloxide , 1%
	C16,20 = ~~10nF~~ = 100nF -marked 104	-through hole changings , take 100 nF for C16,20

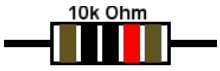
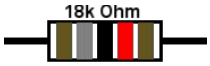
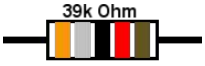


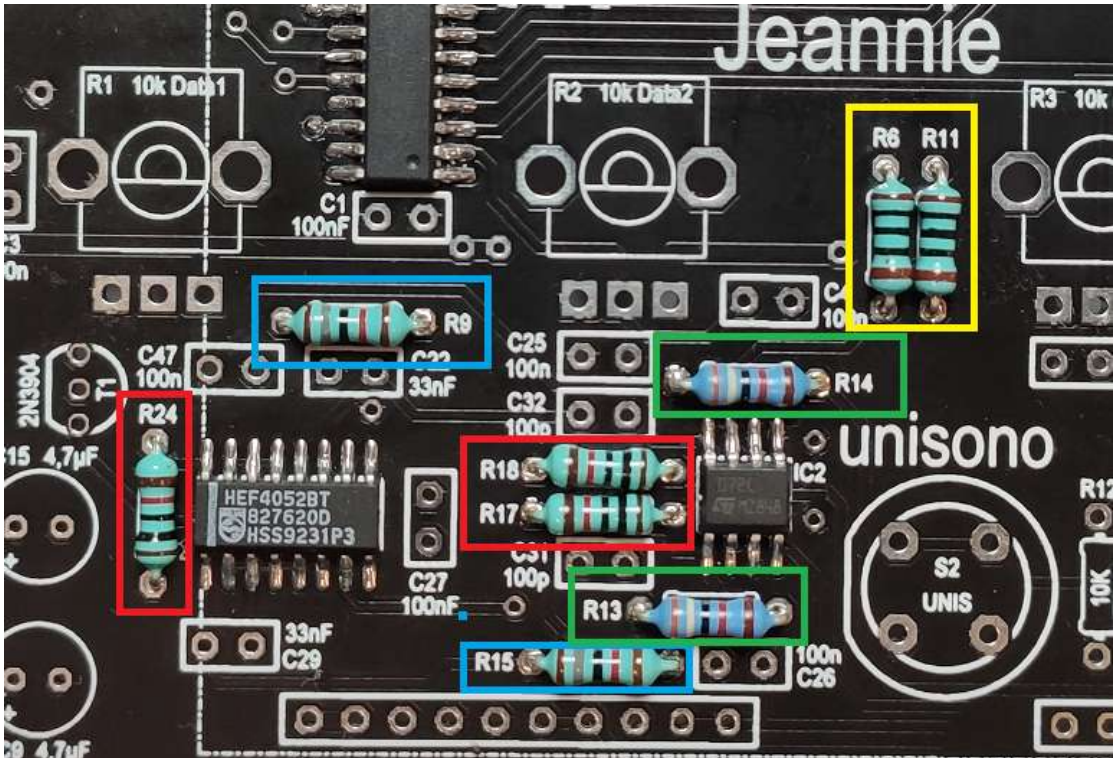
Now it's time for resistors R5,7,8,10,25,41,42,44 and C48.

R5,7,8,10,25,41,42,44 and C48		
	R41,42,44 =150 ohm -brown,green,black,black,brown	-through hole metalloxide , 1%
	R25=4,7 Kohm -yellow,purple,black,brown,brown	-through hole metalloxide , 1%
	R5,7,8,10=3,3 Kohm -orange,orange,black,brown,brown	-through hole metalloxide , 1%
	C48 = 100nF -marked 104	-through hole



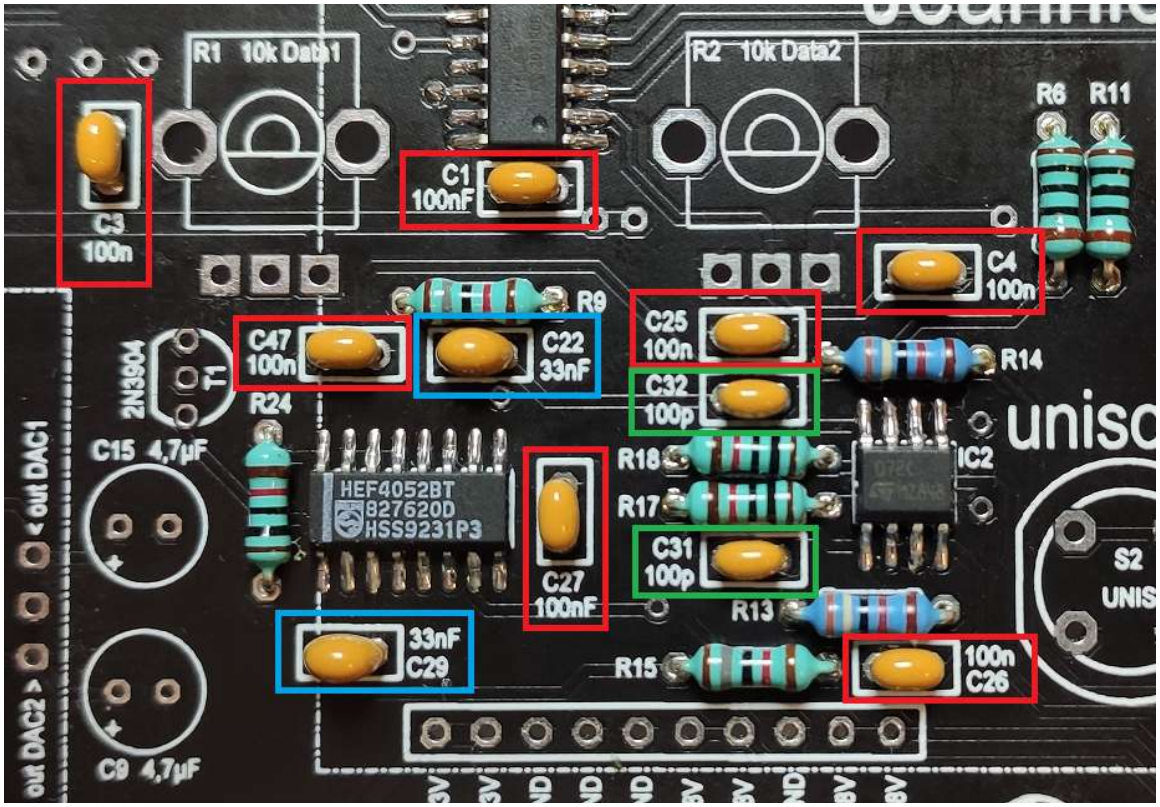
We install the resistors R6, 9, 11, 13,14,15,17,18, 24

R6,9,11,13,14,17,18,24		
	R17,18,24 =10 kohm -brown,black,black,red,brown	-through hole metalloxide , 1%
	R=9,15 =18 kohm -brown,gray,black,red,brown	-through hole metalloxide , 1%
	R13,14= 39Kohm -orange,white,black,red,brown	-through hole metalloxide , 1%
	R6,11=100R -brown,black,black,black,brown	-through hole metalloxide , 1%



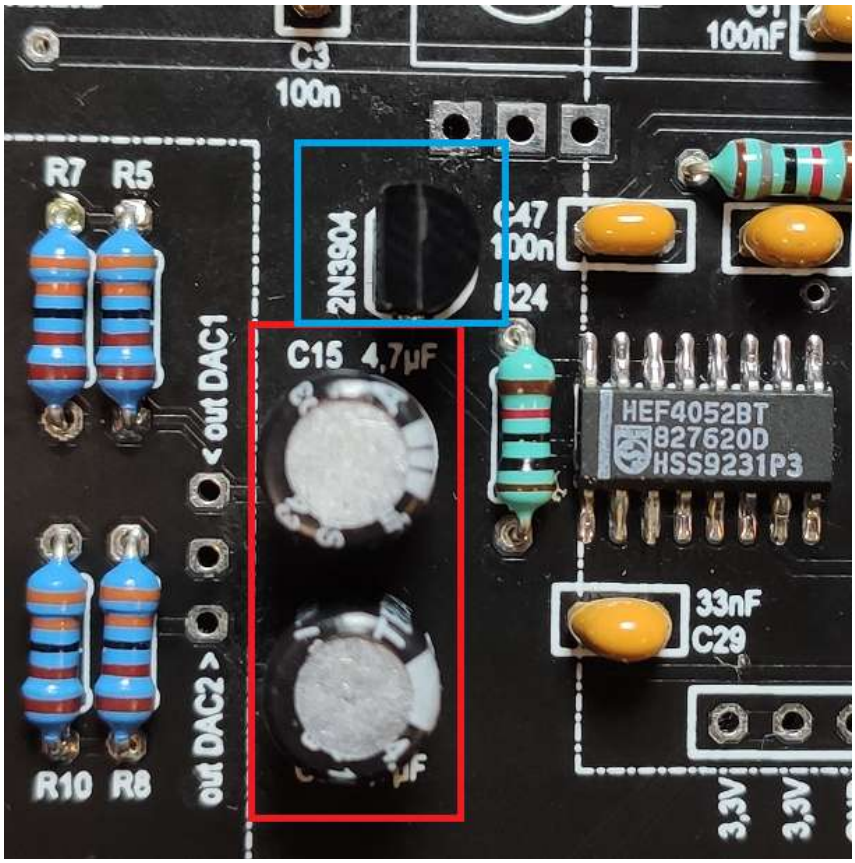
The capacitors C1,3,4,22,25,26,27,29,31,32,47 are soldered in.

C1,3,4,22,25,26,27,29,31,32,47		
	C1,3,4,25,26,27,47=100nF -marked 104	-through hole X7R
	C22,29=33nF -marked 333	-through hole for the bass circut
	C31,32=100pF -marked 101	-through hole



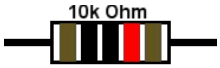
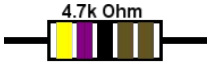
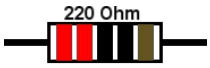
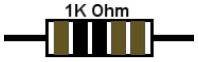
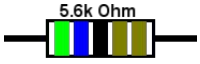
2 Elkos and the Transistor 2N3904

C9,15, T1	C9,C15 is 1uF or 4m7 uF	
	C9,15=1uF or 4,7uF small ** for both caps you can take also 1 uF !!	-through hole polarized, short leg is (-) minus *** max part high =8..9mm
	T1=2N3904 Transistor -marked 2N3904	-through hole



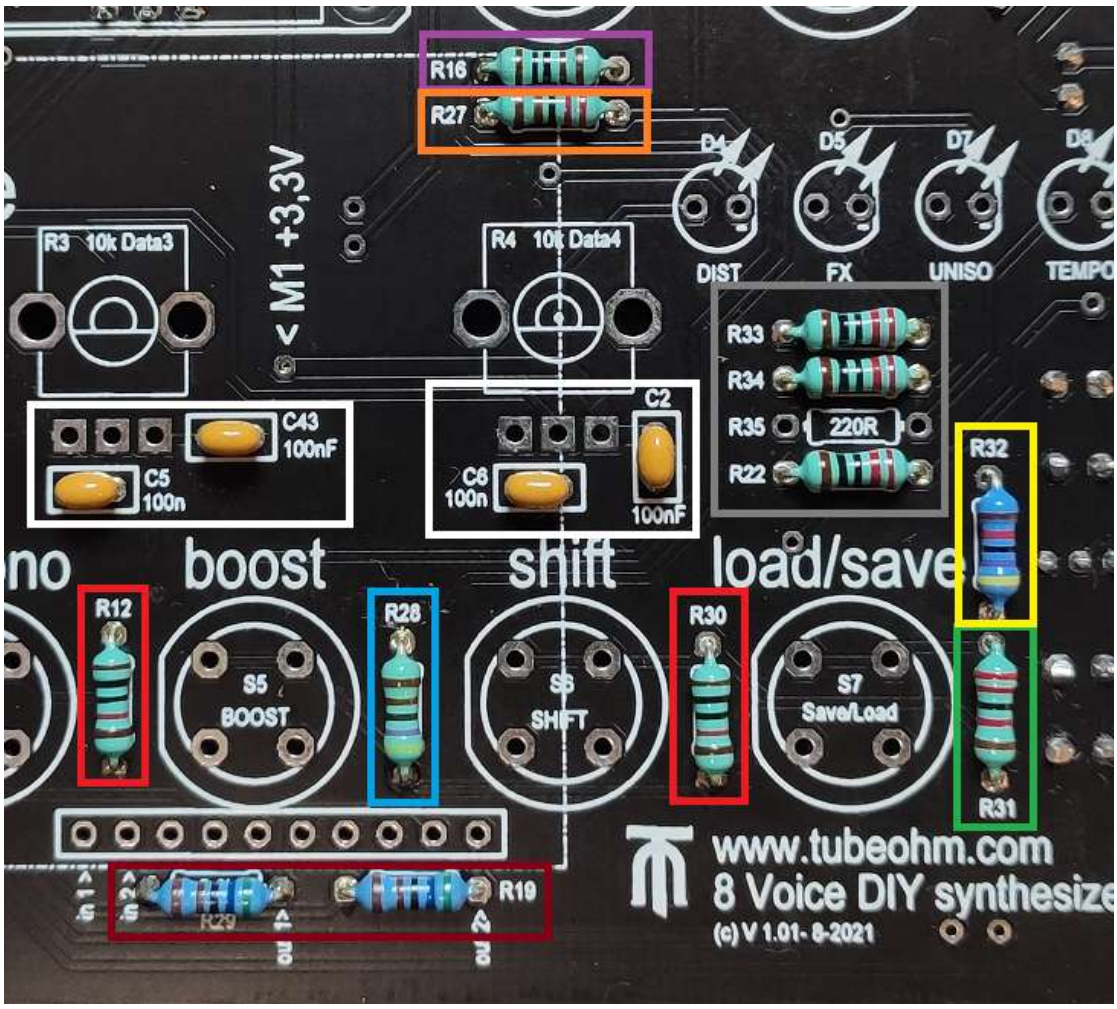
Now we turn to R12,16,19,22,27,28,29,30,31,32,33,34,35,
As well as C2,5,6,43 to

Attention, R12,16,27,28,30,31,32 is the resistor matrix for the buttons.
Please make sure that the correct resistors are installed.

R12,16,22,27,28,30,31, 32,19,29 33,34,35 C2,5,6,43		
	R12,30 =10 kohm -brown,black,black,red,brown	-through hole metalloxide , 1%
	R28 =4,7 kohm -yellow,purple,black,brown,brown	-through hole metalloxide , 1%
	R31= 22 kohm -red,red,black,red,brown	-through hole metalloxide , 1%
	R32=47 kohm -yellow,purple,black,red,brown	-through hole metalloxide , 1%
	R22,33,34,35=220 R ***R35 = 150R if SEQ-Switch with LED -red,red,black,black,brown	-through hole metalloxide , 1%
	R16=1 kOhm -brown,black,black,brown,brown	-through hole metalloxide , 1%
	R27=2,2 kohm -red,red,black,brown,brown	-through hole metalloxide , 1%
	R19,29=5,6 kohm -green,blue,black,brown,brown	-through hole metalloxide , 1%
	C2,5,6,43=100nF -marked 104	-through hole X7R

Attention, 2 buttons (depending on availability) can be used for the sequencer. One push-button without built-in LED or one push-button with built-in LED.

*** R35: if the LED and the push button are connected in parallel, R35 should be 150 ohm. If no illuminated push button is used in the kit, please use R35= 220 ohm.



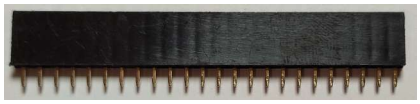
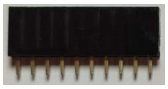
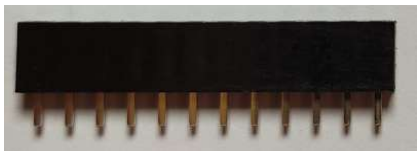
LEDs. No, I have not forgotten them. The LEDs are installed at the very end. For this we need the 4x 10mm spacers and the housing panel.
The LEDs are put on the positions. Then the panel is screwed on.
The LEDs are put through the holes in the panel so that they are flush with the surface and then soldered on.

LED D3,4,5,7,8,10, 1N4001 D1		
	LED D3,4,5,7,8,10,	-through hole polarized , short leg is (-) minus

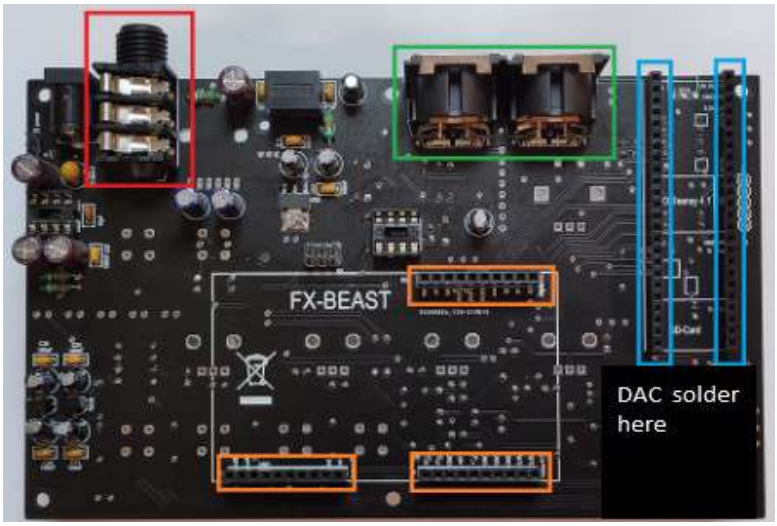
Hurray, half the battle, the electronic components are all in. Now it's time for the mechanical components.

We will now complete the lower side of the board.
The following components are needed.

Attention, board from the lower side !!!

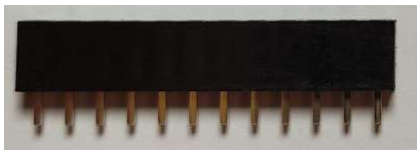
X3,X7 MIDI		
	X3,X7 MIDI in /through	-through hole MIDI connector
	Audio3 stereo 6,3mm connector	-through hole stereo jack socket
	2x24 pin socket h=8,5mm for the teensy module	-through hole
	3x10 pin socket h = 8,5mm for the FX module	-through hole
	~~2x 12 pin socket. h=5mm~~ ~~You need 1x11 and 1x3 pins .~~ ~~Cut it one time with 11 pins for the DAC~~ ~~cut from the second header 3 pins for the dac~~	-through hole not applicable . The DAC is soldered directly from behind onto the PCB

It should now look like this.

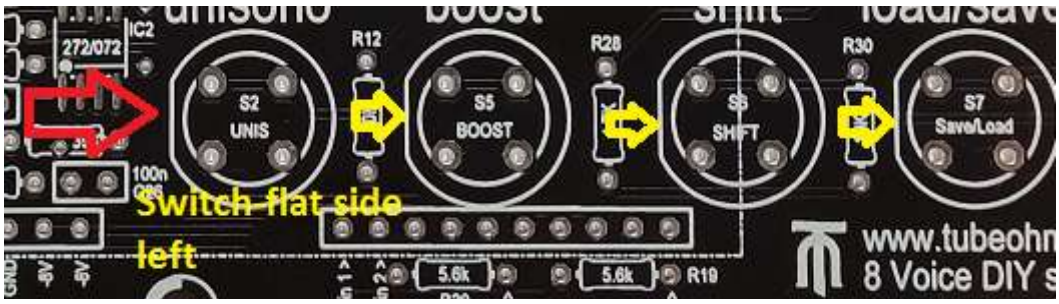


Attention, ~~the socket strip for the DAC is only 5mm high.~~ The DAC is a little lower so that the SD RAM card can be pulled out of the Teensy.
Now the bottom side is ready.
We solder the remaining components from above.

Please turn the board over again.
We need :

	2x 12 pin header. h=5mm Cut it to 8 pin for the LCD	-through hole LCD header 8 pin /cut
	R1,2,3,4=10k poti	-through hole 4x data pots
	Encoder with switch	-through hole 1x data encoder
	Volume poti	-through hole
	red switch/mute-panic	Mute-panic switch attention , polarized flat side to left
 	SEQ Switch. Can be with or without LED.	SEQ switch attention , polarized flat side to left
	4 switches in black,unisono, boost, shift , load/save	attention , polarized flat side to left
	K7 Header 1x3	Select MIDI through/out Set the jumper first in through mode.

ATTENTION, flat side on the button to LEFT

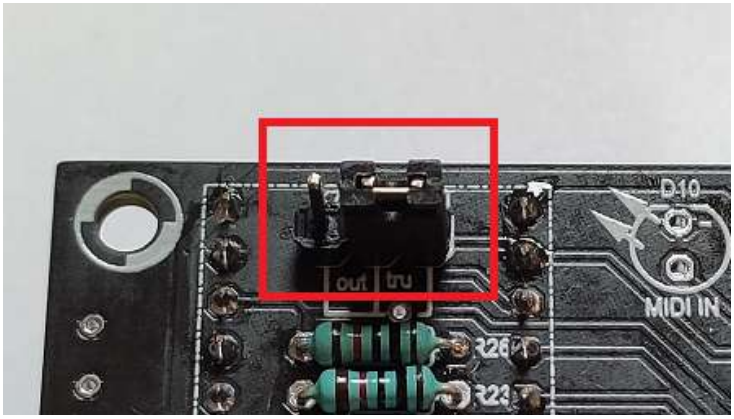


It should now look like this.

The LEDs are not soldered in for the time being, as they will have to be adapted to the housing later. For this purpose, the PCB is screwed to the front panel with the 10mm spacers. The LEDs are now pushed through the holes in the front panel so that they are flush with the front panel. The LEDs are soldered in this position.

Now the 4 jumpers are set and the voltages checked.

Jumper K7 switches the second MIDI socket (x3) either to MIDI out or through . to MIDI . Currently Jeannie can only do MIDI through.

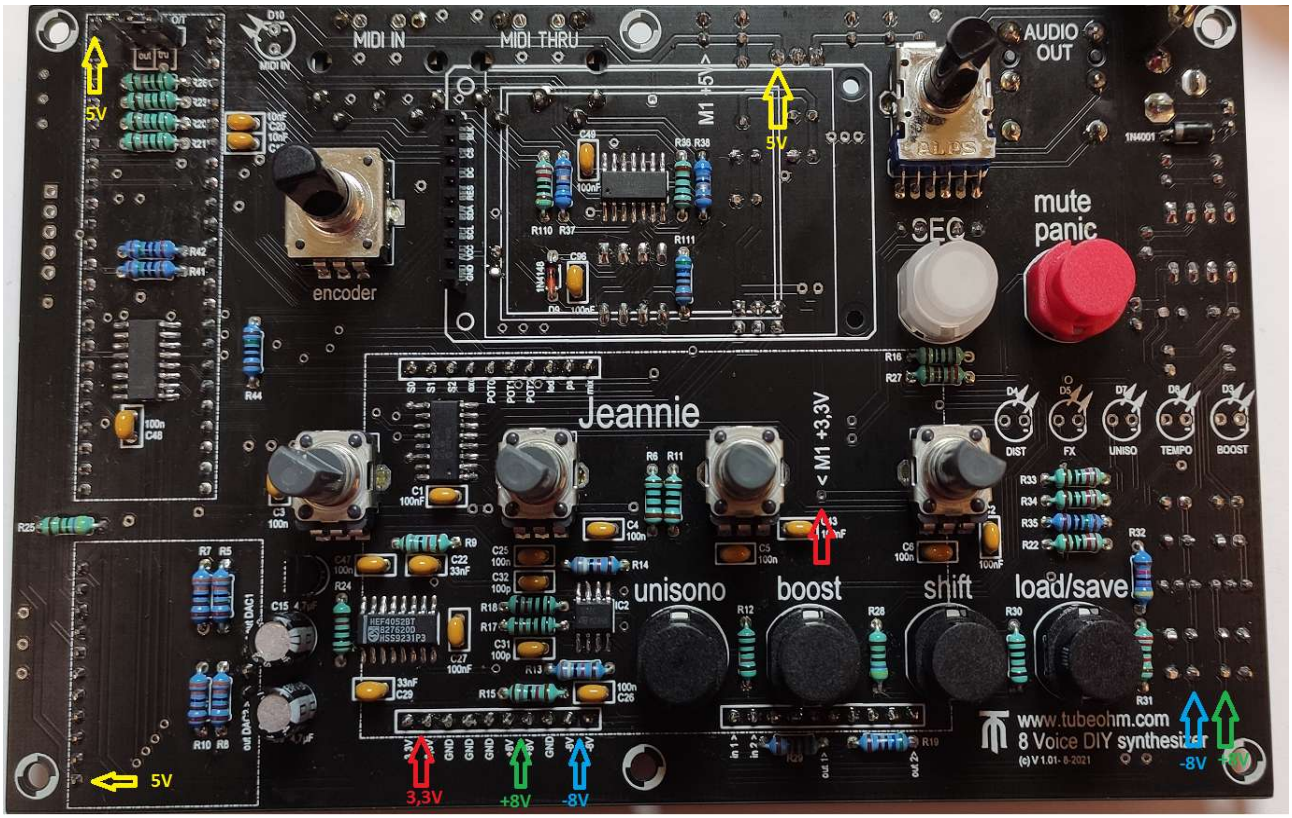


Jumper K4 is an expansion port. In order for these inputs to have a defined potential all three jumpers should be set.

Now the voltages are measured

To do this, the IC LT1054 is plugged into the socket on the back of the board. The IC LT1054 is a charge pump and generates the voltage for the negative (-8 V) regulator. Pay attention to the IC direction!!

K5 - the connection for the power switch is bridged with a jumper.
The ground can be taken from the audio output.
*** or you can solder a grounding point (pinched off leg directly to the 12 volt plug).



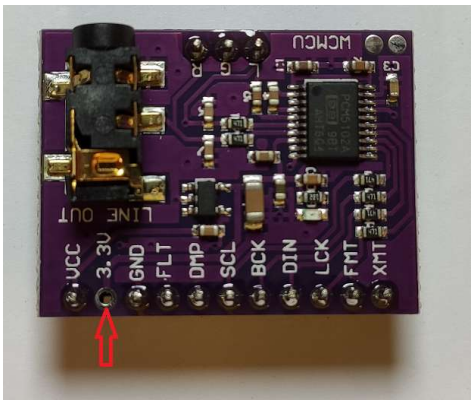
We have several Voltages

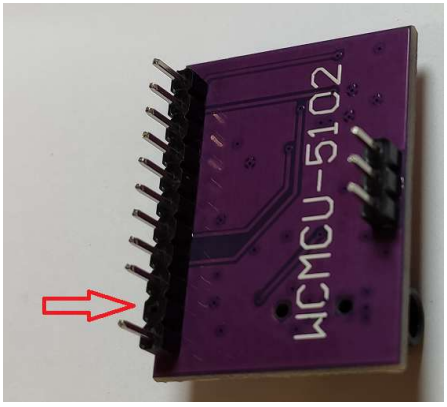
gelb	=	5V
rot	=	3,3V
grün	=	8V
blau	=	-8V

These voltages should be applied to the described points

** It can also be 4.95 V or 5.05 V for 5 volts, for example. - At 3.3V from 3.25..3.35.
+8 can also be 7.8 V. But the voltages should not differ dramatically.
If I only have 4.2 volts at 5 volts, for example, then something is wrong.
Remember that the measuring instruments have tolerances.

Now we turn to the DAC converter. A PCM5102A is used.
It is ready mounted on a circuit board. Note that the 3.3 volt pin is a voltage output and should not be connected. You can simply pull out the leg of the pin connector.
Than solder the DAC from behind to the PCB.

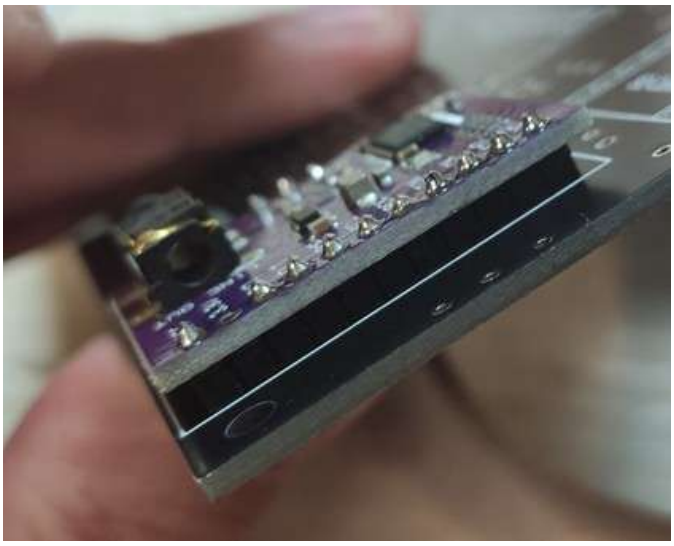




Here is the position of the AD-converter.

Here is the position of the AD-converter.

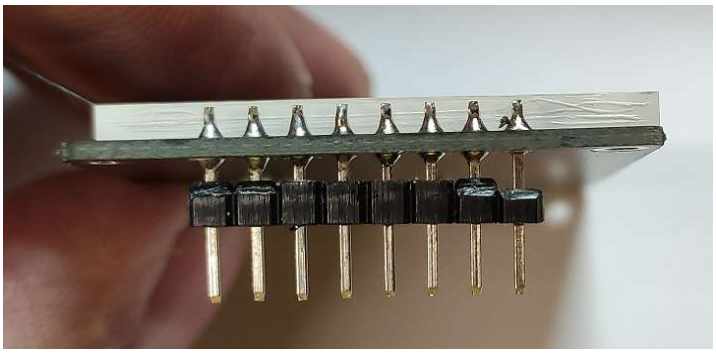
This picture shows how the pin header of the DAC is soldered to the board.

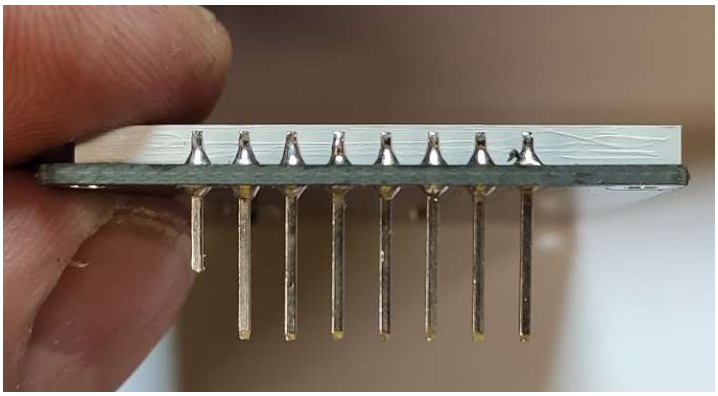


Now the display is installed

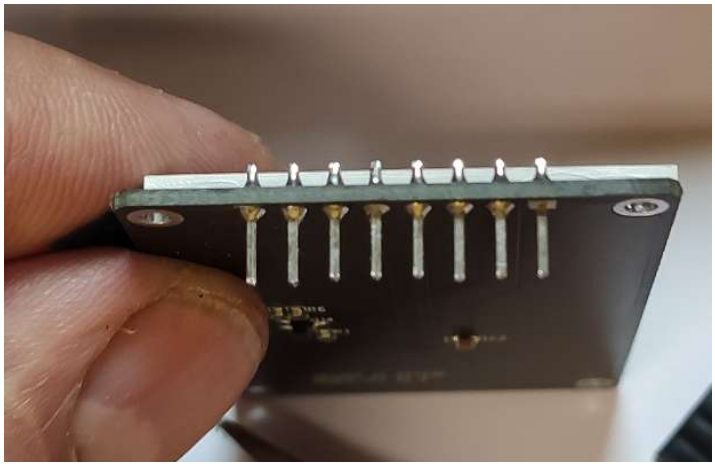
It is important that the display is approx. 9.5 mm high from the circuit board to the display glass, otherwise it will not fit into the housing.

First, carefully remove the plastic holder of the pins. This has already been done with displays from TubeOhm. Then the pins are shortened.

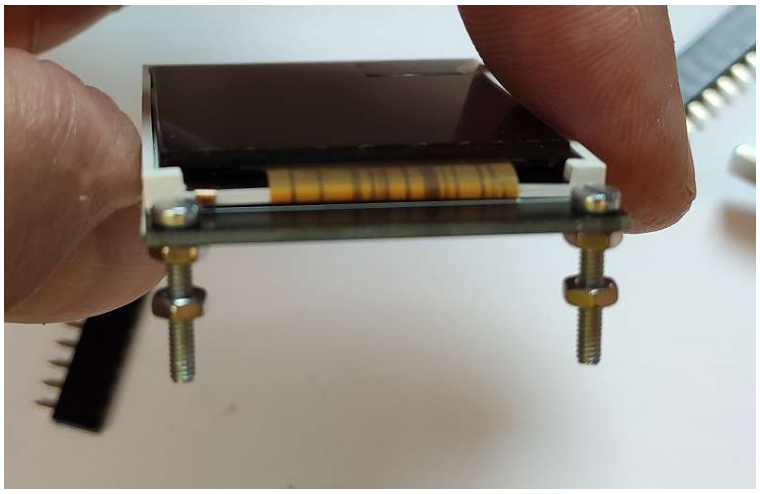




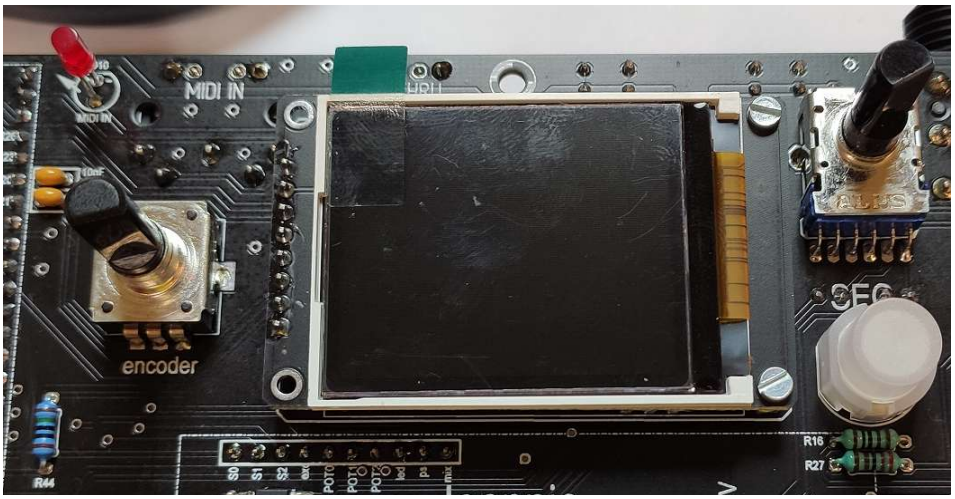
Now all the pins measured from the bottom are approx. 4..4.5 mm long. This is enough to insert the display flush into the socket.



To ensure that the display sits firmly and straight in the board, it is screwed into the board with two M2x12 screws and three nuts each.



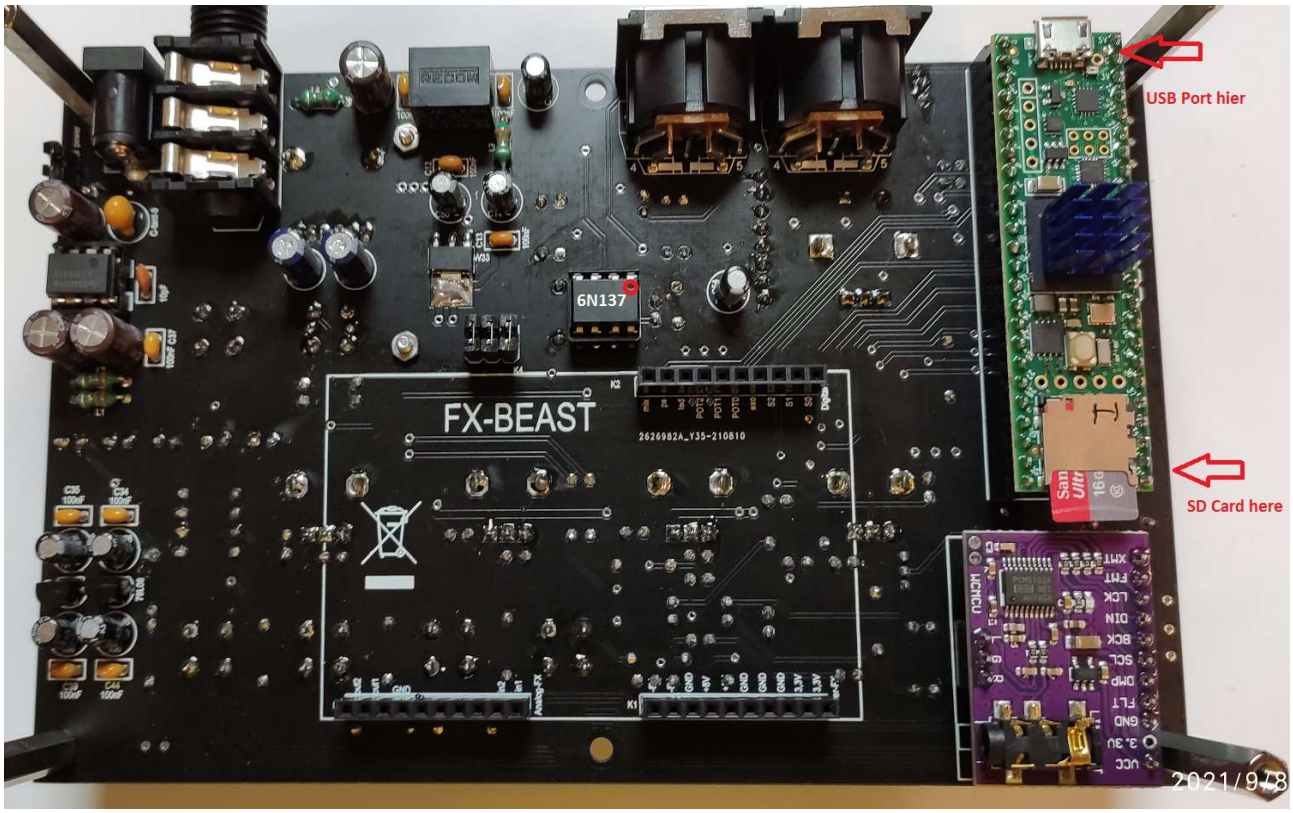
This is how the LCD looks when it is installed.



Now the IC 6N137 , the opto-coupler for MIDI and the previously prepared Teensy 4.1 are plugged onto the motherboard from below.

Teensy 4.1 Check.

5 Volt USB interrupted ?
Heat sink glued on ?

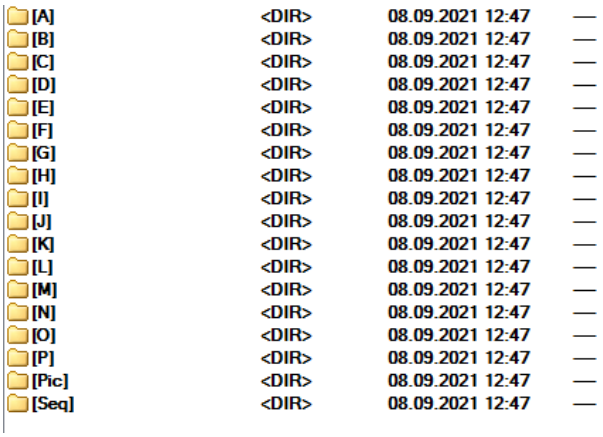


So the basic version without Jeannie's effect is ready and can be put into operation.
The excitement is rising - also for me because I have built the prototype board and documented it here at the same time.

First commissioning - without FX module

As just mentioned, the synthesiser unit is now ready for commissioning, but for now without the salt in the soup, the FX unit with various effects.

The following preparatory work must still be done.
The SD card 8.16 GB is formatted in Fat 32.
The directories A-P, Pic and Seq are copied directly into the root of the SD card.



A-P are the banks for the sound programmes, Pic contains the start images and Seq contains the stored sequences of the sequencer.

*** The zipped sound/seq files can be downloaded from TubeOhm.com.

For flashing the Teensy you need the programme ' teensy.exe '.
*** is provided on the page 'TubeOhm,com

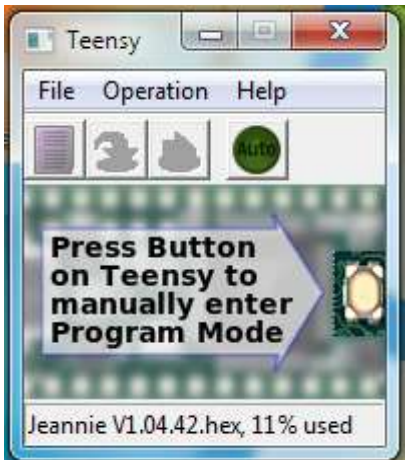
The first thing to do is to install the Teensy. Then we do a short test

1:) Is the Teensy properly inserted in the socket
2:) the DA converter is in the right way in the socket ?
3:) Is K5, the plug for the mains switch short-circuited (-jumper) ??
4:) I have a 12 V power supply unit, 5.5mm /2.1 mm centre positive, minimum 1 A. and , does voltage also come out ?
5:) Is the MIDI keyboard connected in MIDI IN ?
6:) Have I connected an audio cable with a 6.3 mm stereo jack plug?
7:) The SD card is formatted in Fat 32 and all files have been copied to the SD card.

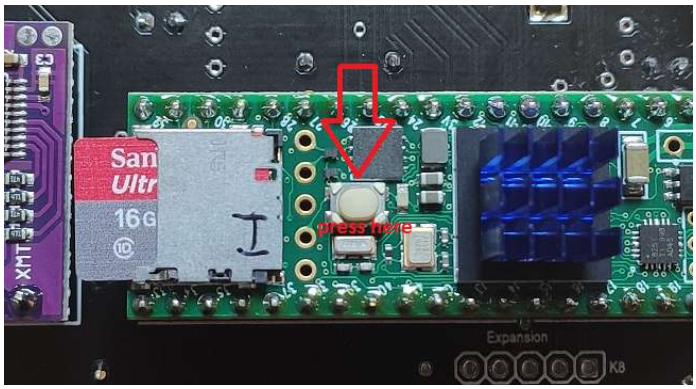
Jeannie is connected to 12 V. Jeannie's screen is white.

With a brand new Teensy, the Jeannie Synthesizer software must be transferred to the Teensy 4.1.

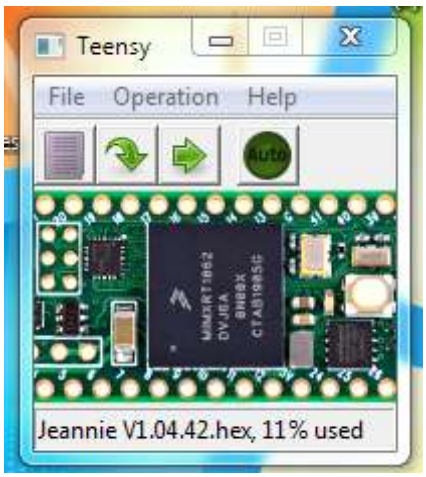
The Teensy is connected to the computer (Win PC) via a USB cable.
The Teensy.exe program is started



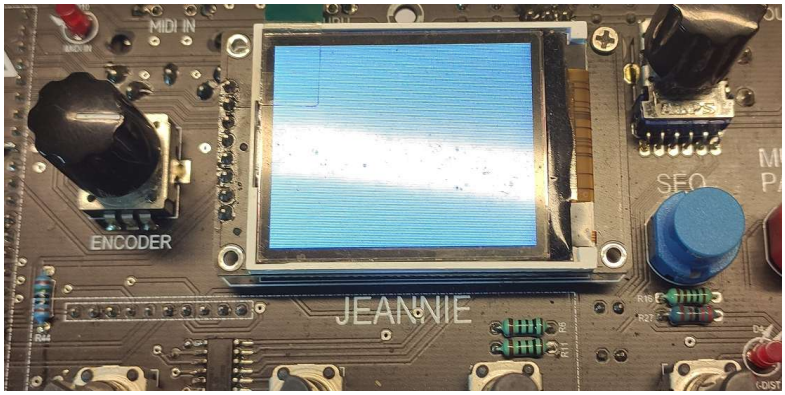
Now press the button on the Teensy,



The arrows ' download and restart' light up green and signal that the Teensy is connected to the PC

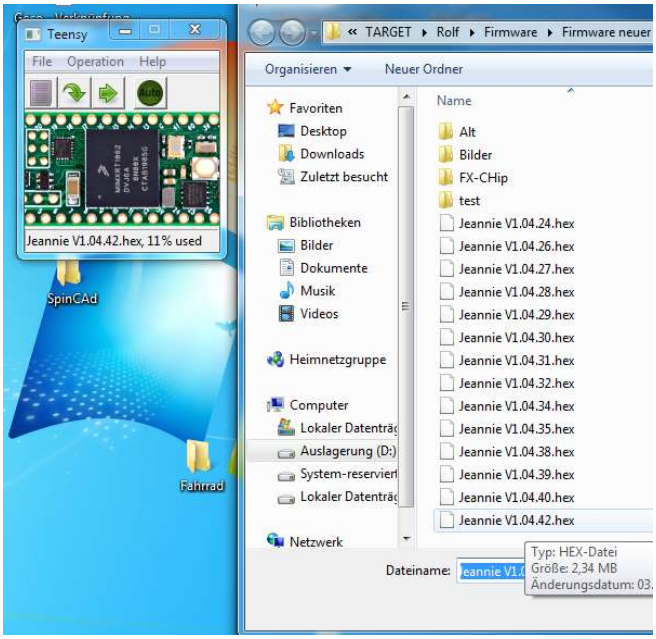


At the same time, the Jeannie's screen is white.

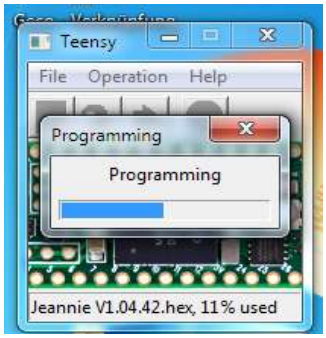


Via the menu 'File , open HEX File'.

the hex file ' Jeannie V1.0x.xx.hex' is loaded with a double click.

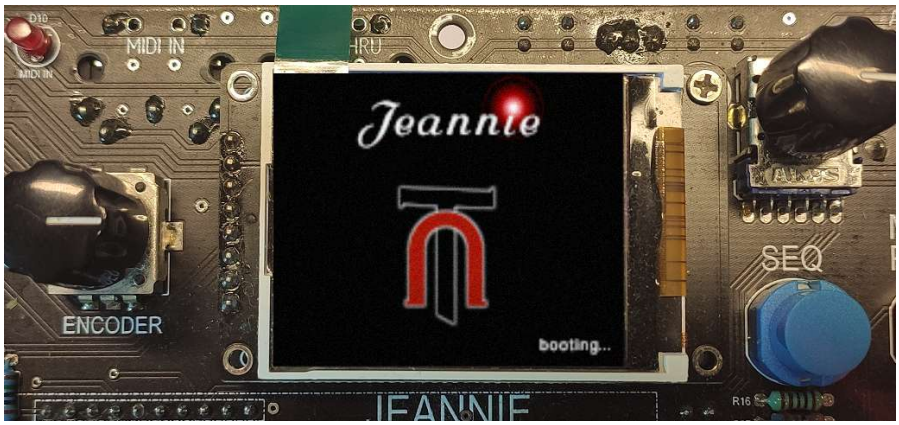


Now press the 'download' button in the Teensy.exe program.



The hex.file is transferred to the Teensy.
Now press the ' reboot ' button.

And Jeannie comes to life



After booting, please check if bank A has been loaded. Jeannie saves the last sound. With a new Teensy, nothing is saved yet. The sound bank can be set with the first potentiometer from the left.

The sound number and the name of the set sound appear.
Now Jeannie - without effects - is ready to play.

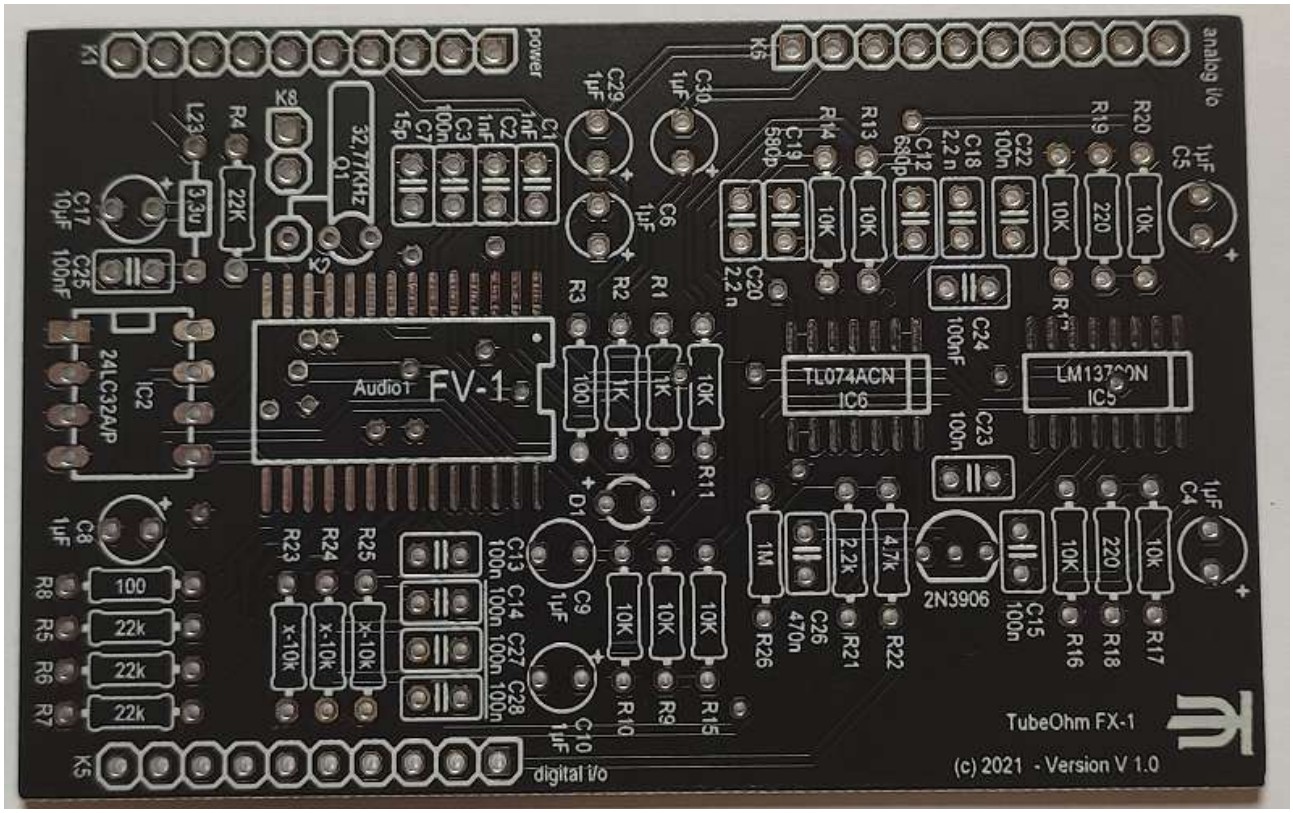
If MIDI and audio are connected, the sounds can be played.
Please check if all switches are working and if the pots and the encoder are working.
Do both audio channels emit a sound?

Congratulations, you have successfully built up 70% of Jeannie.
but now comes the salt in the soup. We are now building the FX module.

Many DIY devices comes without additional effects. But these can dramatically enhance a synthesiser sound.
That is why we have developed an effect board with the FV-1 which can simply be plugged into Jeannie and it enchants with reverberation rooms, long echoes, a chorus and much more. As a special effect, the clock frequency of the DSP can also be controlled. Of course, all parameters can be stored.

And, we'll get started right away.

Here is a look at the DSP board.



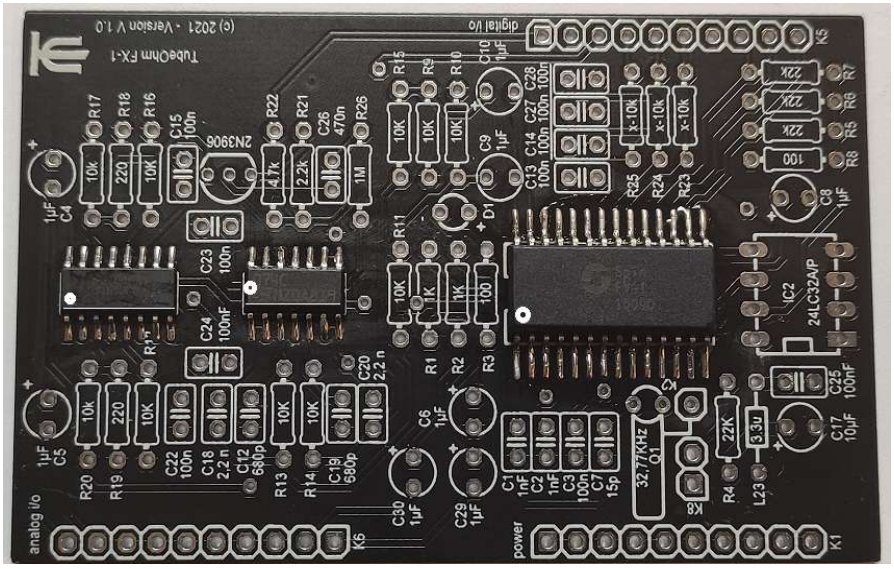
The PCB is quickly assembled.

As ICs we need

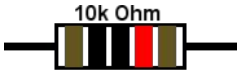
1:) 1x Spinsemi 1001 FV-1 SMD/SMT
2:) 1xTL074 SOIC 14 SMD/SMT
3:) 1x LM13700 SOIC 16 SMD/SMT
4:) 1x 24LC32 EE-Prom from Tubeohm . This contains the effects programmes.

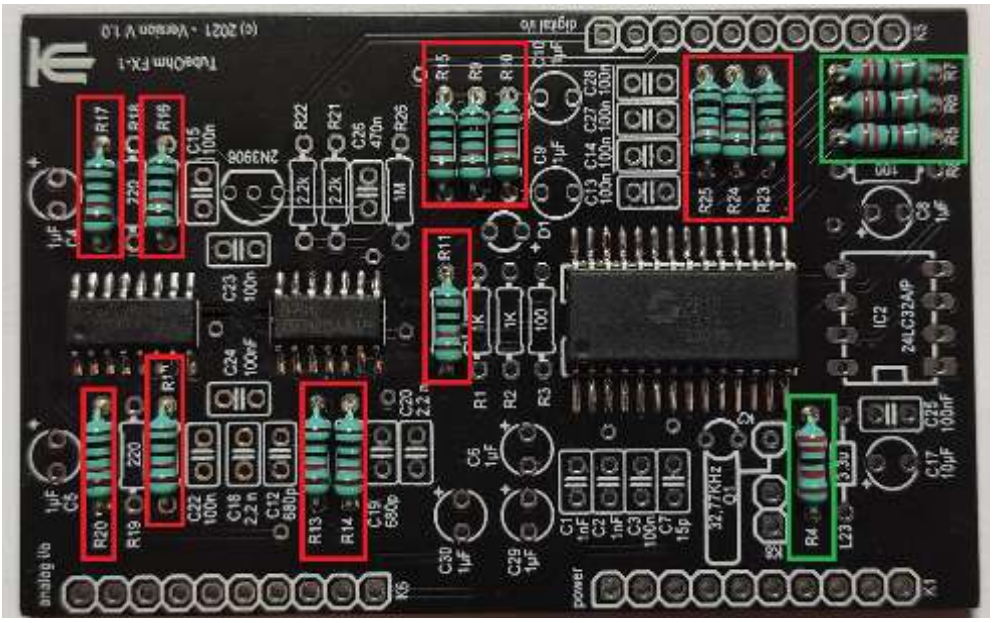
Procedure
First come the SMD ICs, then the resistors, then the semiconductors and finally the IC socket for the EE-Prom and the pin header.

The SMD components are soldered on first as described in ' Soldering the SMD components - Procedure'.
*** You can order the board with soldered ICs from TubeOhm.

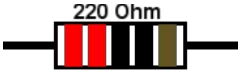


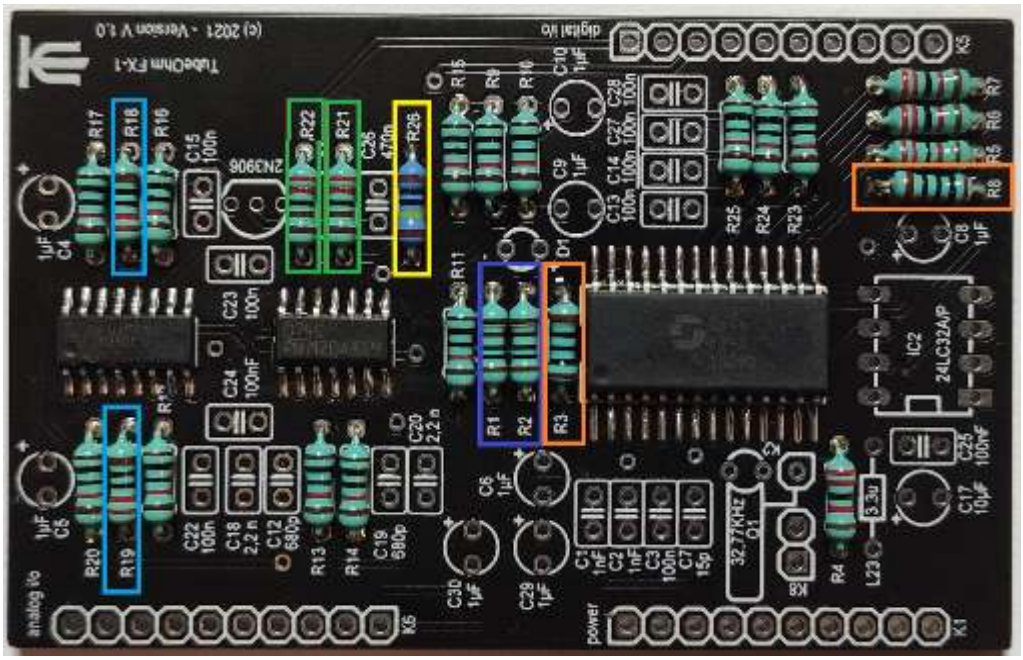
Now we solder 13x10k and 4x22k

R9,10,11,15,12,13,14,16,17,20,23, 24,25, R4,5,6,7		
	R9,10,11,15,12,13,14,16,17,20,23, 24,25, =10Kohm -brown,black,black,red,brown	-through hole
	R4,5,6,7 =22kohm -red,red,black,red,brown	-through hole

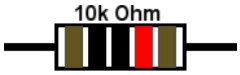


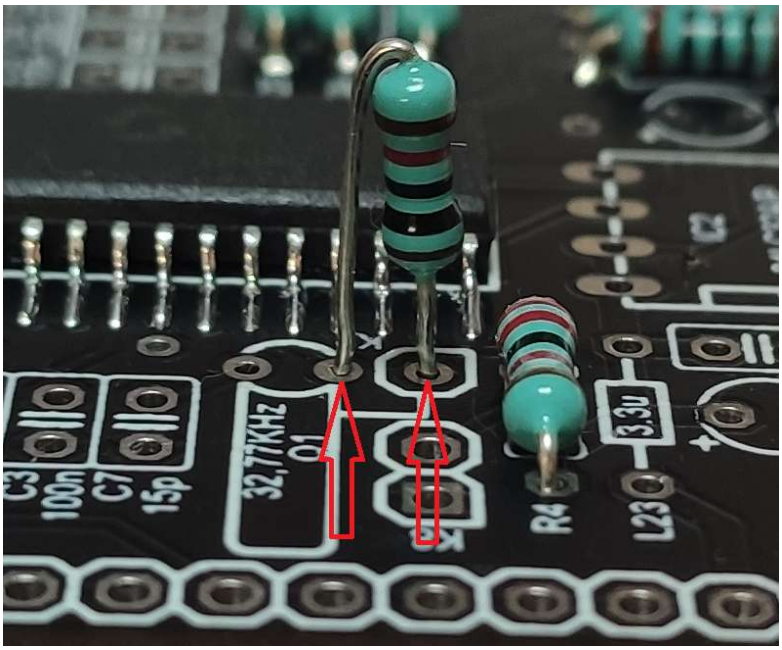
Than R 22,21,26,18,19,1,2,3,8

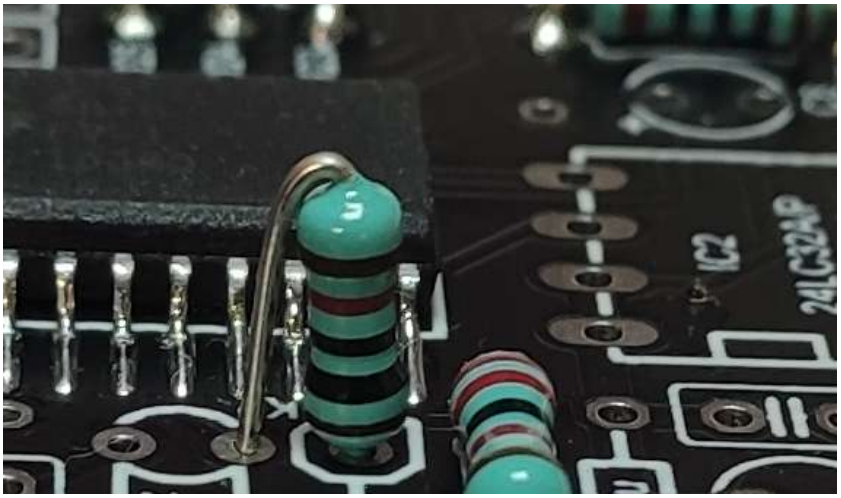
	R21,R22=2,2 Kohm -red,red,black,brown,brown	-through hole
	R26= 1M ohm -brown,yellow,black,black,brown	-through hole
	R18,19= 220ohm -red,red,black,black,brown	-through hole
	R1,2= 1kohm -brown,black,black,brown,brown	-through hole
	R3,8= 100R -brown,black,black,black,brown	-through hole



A 10K resistor for the external clock line MUST be soldered in --Important !!

	Rext-CL (Extern Clock) -brown,black,black,red,brown *** the 10k reduce the amplitude of the external clock	-through hole important

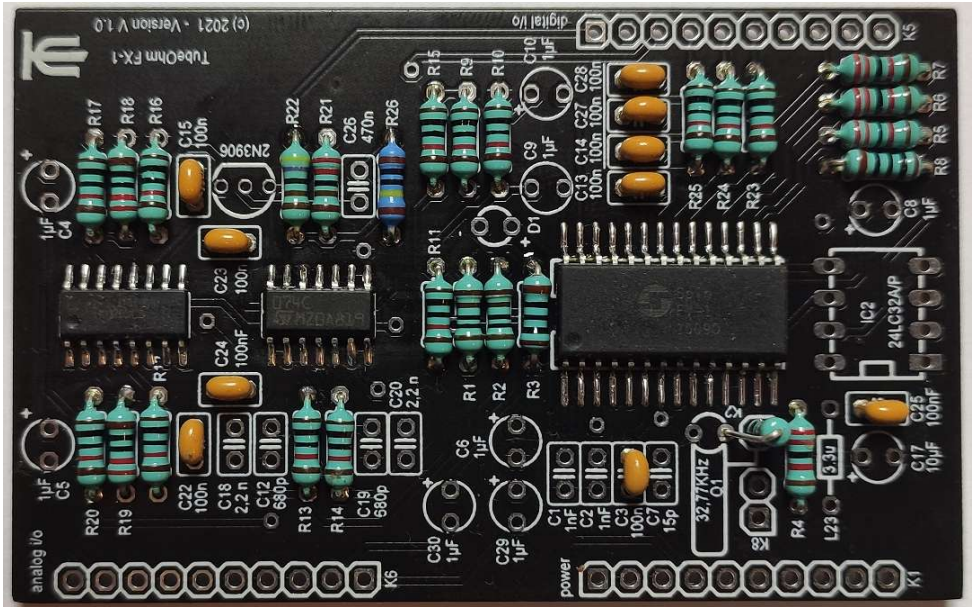




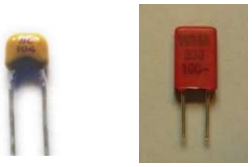
Resistor Rext-CL (external clock) is used to prevent reflections on the external clock line.

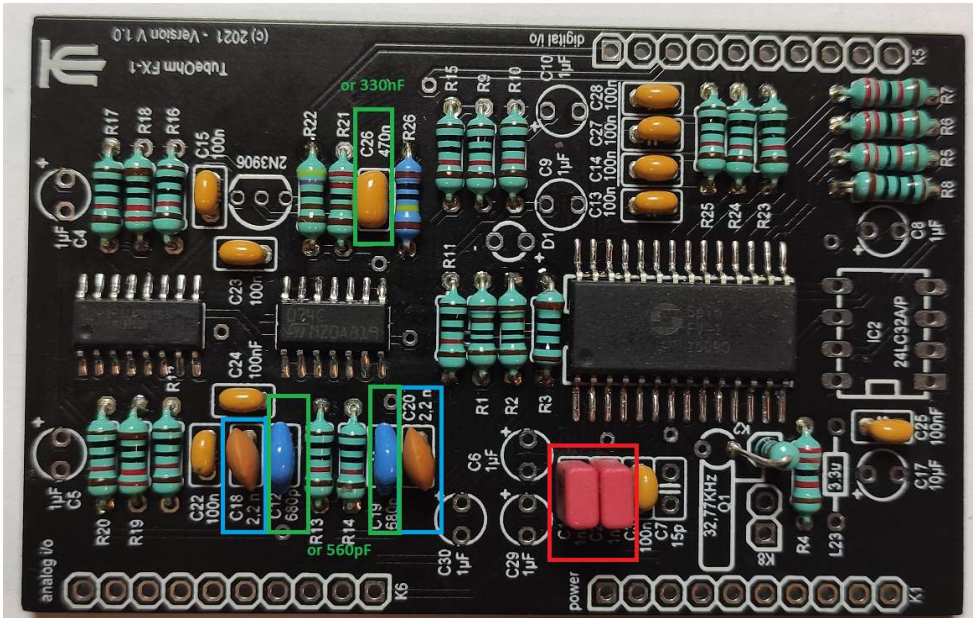
The resistors are now all soldered in. Now let's install the capacitors.

10x100nF		
	C3,13,14,15,22,23,24,,27,28, 25 =100nF -marked 104	-through hole X7R

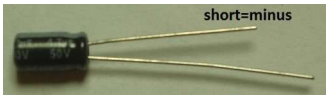
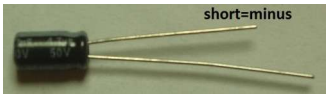
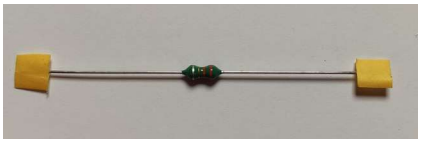


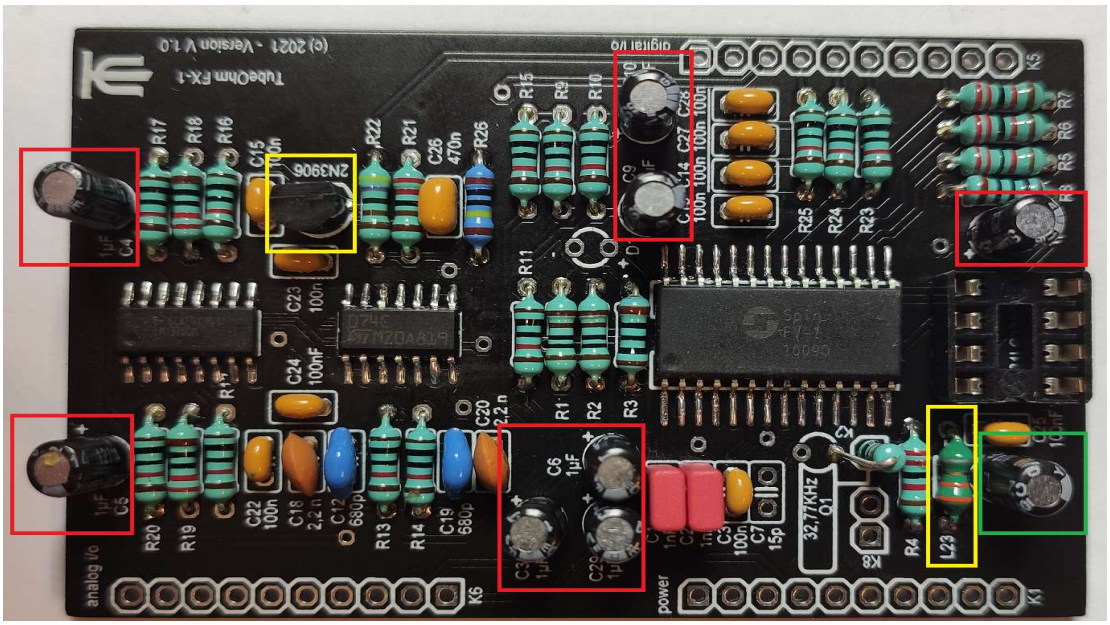
2x1 nF, 2x 2.2 nF, 1x 330 or 470 nF as well as 2x 560p or 680 pF are soldered in.

2x1nF		
	C1,2=1nF -marked 102 or 1000	-through hole
	C18,20=2,2nF -marked 222	-through hole
	C26= 330 nF or 470 nF Marked 334 or 474	-through hole smooth the PWM for Volume (PWM=100kHz)
	C12,19 =560pF...680 pF ***560pF makes the sound a little brighter	-through hole



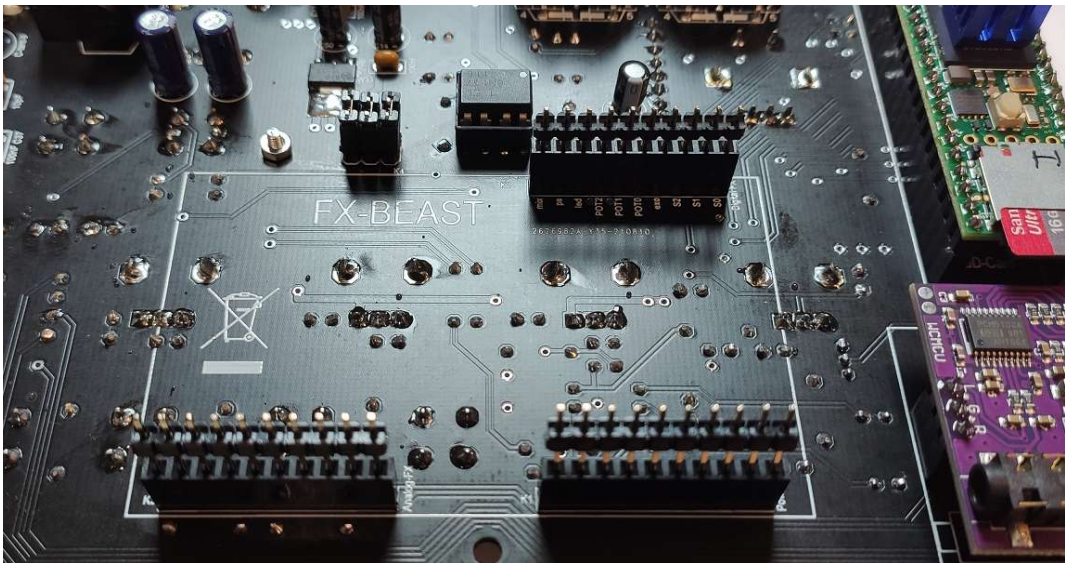
Now the last parts are added, 8x 1uF electrolytic capacitors, 1x 10 uF electrolytic capacitor, 3.3uH coil, the 2N3906 transistor and the IC socket.

	C4,5,6,8,9,10,29,30=1uF -makted 1uF attention- small Elko	-through hole
	C17=10uF attention- small Elko	-through hole
	Transistor 2N 3906	-through hole
	1x Coil 3,3 uH or 4,7uH Colorcode orange, orange, gold, silver yello , purple, gold silver	-through hole
	2x8 pin IC socket for the EE-Prom	-through hole

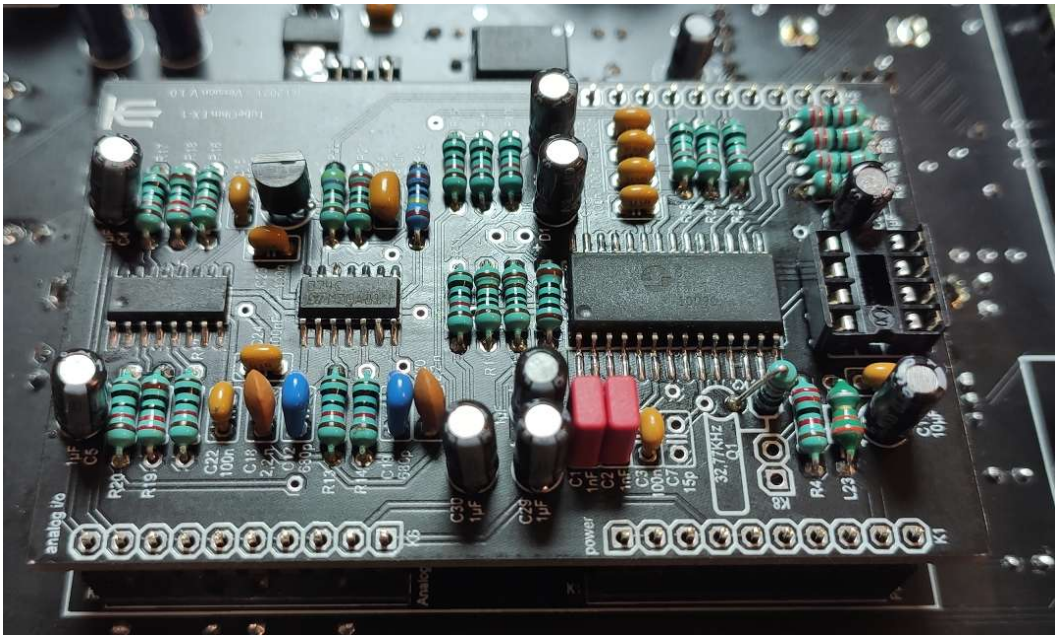


Now only the 3 pin headers have to be soldered.

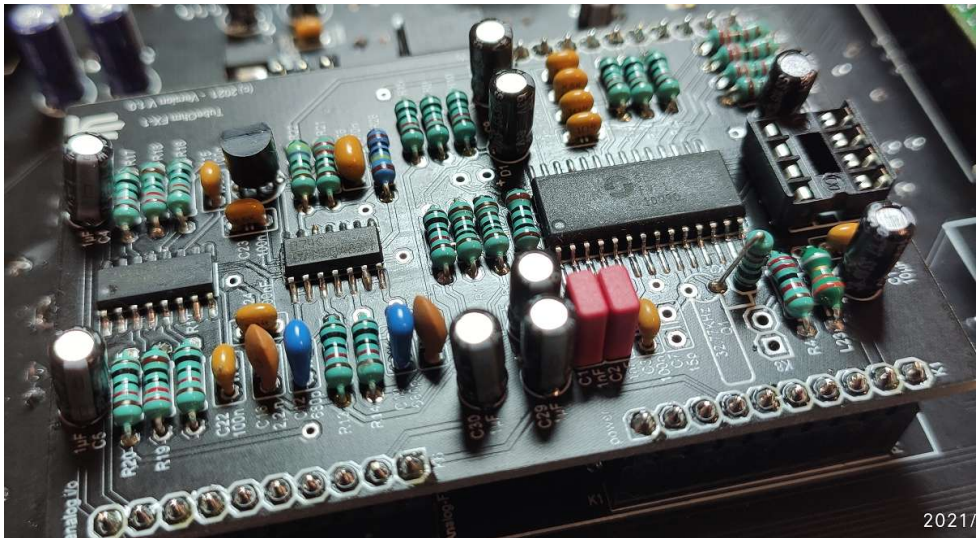
To do this, insert the pin headers into the Jeannie motherboard and press the pin headers flush into the socket headers.



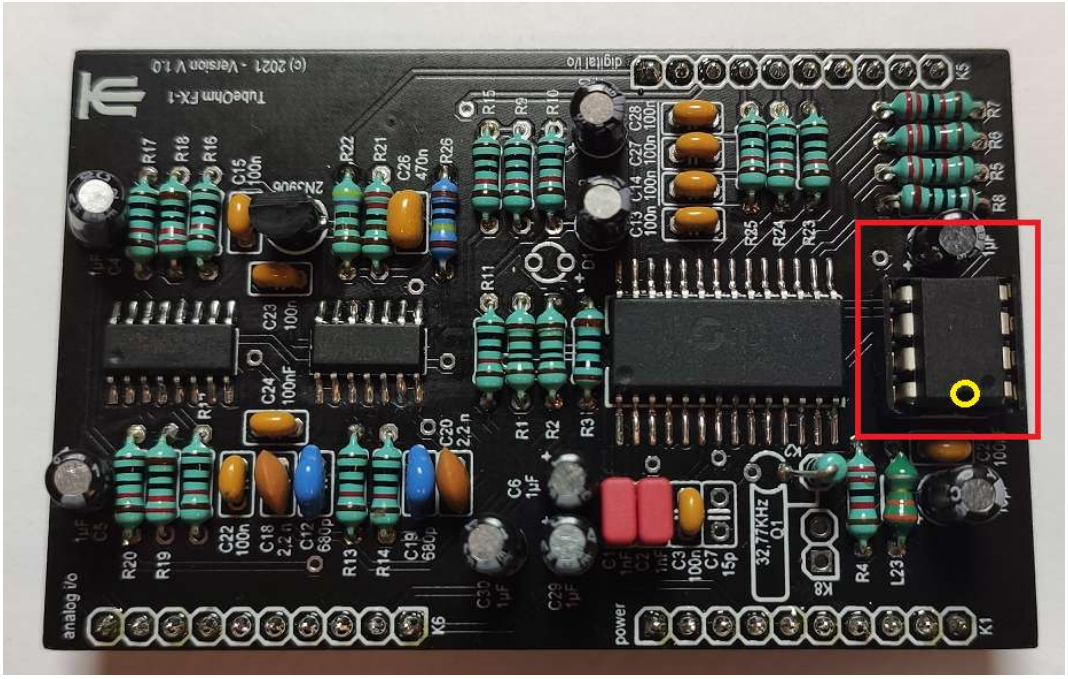
The effect module is placed on the pin headers .



And soldered.



Finally, the EE-Prom (-in the TubeOhm kit-) should be used.



As you can see, some components are not assembled. A quartz crystal is not needed. Furthermore, there is no 15 pF. The same board can later be used as an effect for a Eurorack module (in progress).

So, now it should also echo nicely, (reverben, chirpen- joke) and - well, I think you hear what I mean.

Before the device is installed in the housing, test it again.
There are enough sounds. Now it's time for the sound design or just to make music.

Software updates :

Well, the software is in work. We will add some new features and fix bugs - as far as they are not fixed yet.

Please have a look at our website from time to time.
The current software version is 1.42A / Date 01.04.22
I now refer you to the user manual where you will find some useful tips and tricks.

Andre'

TubeOhm Instruments 2021

Special thanks to:

Rolf, the programmer
T-Synth for giving ideas
Julian Schmidt